

Verification Continuum™ VCS User Guide

Version V-2023.12-SP2, June 2024

SYNOPSYS®

Contents

Customer Support	53
Synopsys Statement on Inclusivity and Diversity	54
<hr/>	
1. Getting Started	55
Simulator Support with Technologies	56
Simulation Preemption Support	57
Setting Up the Simulator	57
Verifying Your System Configuration	57
Obtaining a License	58
Setting Up Your Environment	59
Creating a synopsys_sim.setup File	60
The Concept of a Library in VCS	61
Library Name Mapping	61
Including Other Setup Files	62
Using the SYNOPSYS_SIM_SETUP Environment Variable	62
Supporting VHDL Non-Locally Static Aggregates	62
Displaying Setup Information	63
Displaying Design Information Analyzed Into a Library	64
Using the Simulator	65
Two-step Flow	65
Compiling the Design	66
Simulating the Design	66
Three-step Flow	66
Analyzing the Design	66
Elaborating the Design	66
Simulating the Design	66
Basic Usage Model	67
Two-step Flow	67
Three-step Flow	67
Default Time Unit and Time Precision	68
Searching Identifiers in the Design Using UNIX Commands	68
Examples	70
UTF-8 Unicode File Format	70

2. VCS Flow	71
Three-step Flow	71
Analysis	71
Using vhdlan	72
Using vlogan	75
Disabling Environment Variables Checks on Different Machines	80
Analyzing the Design to Different Libraries	81
Elaboration	81
Using VCS	82
Simulation	84
Interactive Mode	84
Batch Mode	84
Commonly Used Runtime Options	84
Two-step Flow	85
Compilation	85
Using vcs	86
Simulation	89
Interactive Mode	89
Batch Mode	90
Commonly Used Runtime Options	90
Partition Compile Flow	90
3. Modeling Your Design	91
Avoiding Race Conditions	91
Using and Setting a Value at the Same Time	92
Setting a Value Twice at the Same Time	92
Flip-Flop Race Condition	93
Continuous Assignment Evaluation	94
Counting Events	94
Time Zero Race Conditions	95
Race Detection in Verilog Code	95
The Dynamic Race Detection Tool	96
Introduction to the Dynamic Race Detection Tool	96
Enabling Race Detection	98
The Race Detection Report	98
Post-Processing the Report	101
Debugging Simulation Mismatches	102
The Static Race Detection Tool	103

Contents

Race Detection Tool to Identify Race between Clock and Data	104
Use Model	105
Examples	106
Limitations	108
Optimizing Testbenches for Debugging	108
Conditional Compilation	109
Enabling Debugging Features at Runtime	110
Combining the Techniques	112
Creating Models That Simulate Faster	112
Unaccelerated Data Types, Primitives, and Statements	113
Inferring Faster Simulating Sequential Devices	114
Modeling Faster always Blocks	117
Using Verilog 2001 Constructs	117
Case Statement Behavior	119
Precedence in Text Macro Definitions	119
Memory Size Limits in the Simulator	119
Memory Size Calculation of Bit Data Type	120
Memory Size Calculation of Reg Data Type	121
Using Sparse Memory Models	121
Obtaining Scope Information	122
Scope Format Specifications	122
Returning Information About the Scope	125
Avoiding Circular Dependency	127
Translating VHDL Package to SystemVerilog Package	128
Use Model	128
VHDL Type Mapping	130
Limitations	131
VITAL2000 Negative Constraint Calculation	132
Using VITAL2000 NCC	132
Disabling VITAL2000 Conformance Checks	133
<hr/>	
4. Compiling/Elaborating the Design	134
Compiling/Elaborating the Design in the Debug Mode	134
Optimizing Simulation Performance for Desired Debug Visibility With the -debug_access Option	135

Contents

Use Model	135
Key Points to Note	138
Incrementally Removing Debug Capabilities	138
Assertion Debug Support	139
Verdi One Search Support	139
Reporting Global Debug Capability Diagnostics	139
Specifying Design Regions for -debug_access Capabilities	141
Example	143
Key Points to Note	143
Region Debug Enhancements	143
Automatic Debug Capability Addition Using \$fsdbDumpvars(level,path)	145
Enabling Additional Debug Capabilities	146
Driver/Load Debug Capability	146
Statement Debug Capability	146
Force/Deposit Debug Capability	146
Class Debug Capability	147
Reduction in the Objects Being Dumped	147
Using -debug_access With Tab Files (With -P Option)	147
Unused Tab File Calls	147
Including Tab Files	147
Dumping FSDB	147
Interaction With Other Debug Options	148
Dynamic Loading of DPI Libraries at Runtime	148
Use Model	148
Dynamic Loading of PLI Libraries at Runtime	150
Key Compilation or Elaboration Features	150
Initializing Verilog Variables, Registers, and Memories	151
Initializing Verilog Variables, Registers, and Memories for an Entire Design	151
Initializing Verilog Variables, Registers, and Memories in Selective Parts of a Design	152
Selections for Initialization of Registers or Memories	153
Reporting the Initialized Values of Variables, Registers, and Memories	154
Overriding Generics and Parameters	154
Overriding Verilog Parameters	155
Overriding VHDL Generics	156
Usage Model	157
Checking for x and z Values In Conditional Expressions	158
Enabling the Checking	159
Filtering Out False Negatives	160
Cross Module References (XMRs)	161

Contents

The hdl_xmr Procedure and the \$hdl_xmr System Task	162
Data Types Supported	163
Using the hdl_xmr Procedure	164
Using the \$hdl_xmr Task	165
Use Model	166
Examples	167
\$hdl_xmr Support for VHDL Variables	171
Data Type Support and Usage Examples	171
Support for Native XMR force and release	173
Verilog Configurations and Libmaps	177
Library Mapping Files	178
Configurations	178
Hierarchical Configurations	181
The -top Compile-Time Option	181
Limitations of Configurations	182
Use Model	182
Example	182
Using the -liblist Option	186
Design Cells and Library Cells	188
Library Search Order Rules	189
Example Testcase Files	195
Usage Examples for Library Search Order Rules for Verilog or SystemVerilog Designs	197
Lint Warning Message for Missing 'endcelldefine'	211
Error/Warning/Lint Message Control	214
Controlling Error/Warning/Lint Messages Using Compile-Time Options . .	214
Controlling Error/Warning/Lint Messages Using a Configuration File . .	225
Extracting the Files Used in Elaboration/Compilation	231
XML File Format	232
Limitations	237
Reducing Compile Time for Post-Process Only Debug	238
Use Model	238
5. Simulating the Design	239
Using Verdi	239
Using UCLI	240
-ucli2Proc Option	242
Options for Debugging Using Verdi and UCLI	242
Reporting Forces/Injections in a Simulation	243
Use Model	244
Key Points to Note	245

Contents

Reporting Force/Deposit/Release Information	245
Handling Forces on Bit/Part Select and MDA Word	246
Handling Forces on Concatenated Codes	247
Output Format	247
Usage Examples	249
Generating Force List Report for Desired Instance Hierarchies	254
Examples	255
Displaying Different ID for Each Verilog Force	257
Enhanced Force List Report	257
Use Model	258
Displaying Source Information for VHDL Forces	258
Use Model	259
Displaying Source Information for External Forces	259
Enhanced Force List Report	260
Use Model	261
Viewing Force Information in Interactive Debug Mode	261
Use model	261
Example	262
Reporting \$deposit Value Changes	264
Limitations	264
Key Runtime Features	266
Overriding Generics at Runtime	266
Use Model	267
Passing Values from the Runtime Command Line	269
Using -f Runtime Option	270
Limitations	270
Saving and Restarting the Simulation	271
Save and Restart Example	272
Save and Restart File I/O	273
Saving and Restoring Files During Save and Restore	273
Restoring the Saved Files from the Previous Saved Session	274
Save and Restart With Runtime Options	274
Additional Save and Restore Options	275
Specifying Long Time Before Stopping the Simulation	276
Preventing Time 0 Race Conditions	277
Resolving RTL Simulation Races in Mixed HDL Designs	278
Recommended Approach to Resolve Race Conditions	278
Resolving RTL Simulation Races in Verilog Designs	279
Recommended Approach to Resolve Race Conditions	280
Supporting Simulation Executable to Return Non-Zero Value on Error Results	284

Contents

Use Model	285
Limitation	285
Supporting Memory Load and Dump Task Verbosity	285
Use Model	286
<hr/>	
6. The Unified Simulation Profiler	288
The Use Model	288
Compile Time Option	288
Runtime Options	289
Running the profrpt Profile Report Generator	292
Specifying Views	294
The Snapshot Mechanism	296
Specifying Timeline Reports	297
Recording and Viewing Memory Stack Traces	298
The Caller-Callee Views	300
The Time FlameGraph View	305
HTML Profiler Reports	312
Display of Parameterized Class Functions and Tasks in Profiling Reports	337
Hypertext Links to the Source Files	339
SystemC Views	342
Reporting PLI, DPI, and DirectC Function Call Information	348
Compiling and Running the Profiler Example	348
Profiling Time Used by Various Parts of the Design	350
Profiling Memory Used by Various Parts of the Design	351
Constraint Profiling Integrated in the Unified Profiler	352
The Time Constraint Solver View	352
The Memory Constraint Solver View	359
Performance/Memory Profiling for Coverage Covergroups	362
Default Summary View	363
Time/Memory Summary View	363
Time/Memory Module View	363
Time/Memory Construct View	364
Time/Memory Covergroup View	365
Generating Simprofile Dynamic Report	366
Home	367
Memory Summary	369
Dynamic Memory	371
Verilog	375
Memory Construct Summary View	377
Memory Search View	378

Contents

Advanced Simprofile Usages	380
The Accumulative Views	380
The Comparative View	385
Reporting Debug Capabilities for Each Module	387
Use Model	388
HTML Reports	388
Text Reports	390
Limitations	391
Simulation Time Slice Based Profiler	391
Use Model	391
Diagnostics	393
Limitations	393
Line-Based CPU Time Profiler	393
Use Model	394
Limitations	395
Isolating the Cost of Garbage Collection	396
Use Model	396
Isolating the Cost of Loading Design Database	396
Use Model	396
Third-Party Shared Library Profiler Report	397
Use Model	397
VHDL Unified Simulation Profiler Report	398
Limitations	399
The HSIM View With the Simulation Profiler Report	399
Benefits	399
Limitations	400
<hr/>	
7. Diagnostics	401
Using Diagnostics	402
Using -diag Option	402
Compile-time Diagnostics	403
Libconfig Diagnostics	403
Example	404
Timescale Diagnostics	405
Example	406
Generating Information on Unused Libraries at vlogan	408
Use Model	408

Contents

Usage Example	408
Generating Information on Unused Libraries at VCS	409
Use Model	409
Obtaining Statistics on Package Utilization	410
Use Model	410
Runtime Diagnostics	412
Diagnostics for VPI/VHPI PLI Applications	412
Keeping the UCLI/Verdi Prompt Active After a Runtime Error	415
UCLI Use Model	415
Verdi Use Model	416
UCLI Usage Example	417
Limitations	419
Diagnosing Quickthread Issues	419
Diagnosing Quickthread Issues in DPI	419
Diagnosing Quickthread Issues in SystemC	420
Post-Processing Diagnostics	423
Using the vpdutil Utility to Generate Statistics	423
The vpdutil Utility Syntax	423
Options	423
Sparse Memory Diagnostics	424
Compile Time Options	425
Example	426
Runtime Options	427
Sparse Disable Options	429
Event Order Diagnostics	430
Flop Data Race Rules	430
Use Model	431
Diagnostic Report Details	431
Limitations	432
<hr/>	
8. VPD, VCD, and EVCD Utilities	433
Advantages of VPD	434
Dumping a VPD File	434
Using System Tasks	434
Enable and Disable Dumping	434
Override the VPD Filename	437
Dump Multi-Dimensional Arrays and Memories	437
Using \$vcdplusmemorydump System Task	439

Contents

Capture Delta Cycle Information	440
Dumping an EVCD File	440
Using \$dumpports System Task	441
Analyzing Direction Only for inout Ports	441
Dumping EVCD File for Mixed Designs Using UCLI dump Command	441
Use Model	441
Use Model for Dumping CCN Driver Through INOUT	442
Limitations	443
Unsupported Port Types	443
Unsupported DUT Types	444
SystemC Support	444
Post-processing Utilities	444
The vcdpost Utility	445
Scalarizing the Vector Signals	445
Uniquifying the Identifier Codes	446
The vcdpost Utility Syntax	446
The vcdiff Utility	447
Syntax	447
The vcdiff Utility Output Example	452
The vcat Utility	453
The vcat Utility Syntax	453
Generating Source Files From VCD Files	457
Writing the Configuration File	458
The vcsplit Utility	461
The vcsplit Utility Syntax	461
The vcd2vpd Utility	463
Options for Specifying EVCD Options	464
The vpd2vcd Utility	464
The Command File Syntax	468
The vpdmerge Utility	471
Restrictions	471
Limitations	472
Value Conflicts	472
The vpduutil Utility	472
9. Compile Time Productivity	474
Partition Compile	474
Autopartitioning	476
Specifying Partitions Manually	477

Contents

Specifying Partitions in a V2K or SystemVerilog Configuration	478
Specifying Partitions in an +optconfigfile File	479
Specifying Partitions in a VHDL Configuration File	479
Cell or Instance Based Partitions	480
Instance or Cell Based Partitions in a V2K/SV Configuration	481
Package Based Partitions	482
Partition Compile Use Model	483
The Partition Compile Two-Step Flow	483
The Partition Compile Three Step Flow	484
Commonly Used Partition-Compile Options	485
Limiting Partitions	485
Specifying a Location for the Partition Data Generation	486
Partition Compile Example	486
Parallel Compilation With Partition Compile	488
Adaptive Scheduling Using fastpartcomp	488
Turbo Compile Flow	488
Multiple Test Model	489
Multiple User Model	491
Diagnostics	493
Relocation Methods with Partition Compile	494
Using -simcopy Compile Time Options	495
Using simcopy Utility	496
Turbo Compile Example	497
Cross-Module References (XMRs)	500
Support for Skipping Intra Position Rewrite in Partition Compile Flow	501
Usage	501
Examples	501
Limitation	502
Coding Guidelines	502
Scenarios Causing a Recompilation of a Partition	504
Achieving the Best Turnaround Time with Partition Compile	505
Profiling of Compilation Time	506
Improving Partition Compile Performance for SDF Designs	508
Use Model	508
Usage Example	508
Limitations	510
Rewriting XMRs to VPI Calls	510
Use Model	511
Configuration File	511
Example	511
Limitations	512
Partition Compile Limitations	512

10. VCS Distributed Simulation	515
VCS Distributed Simulation	515
Distributed Simulation Setup	518
Defining Communication Among Client Simvs	518
Key Points to Note	521
RTL Connection	523
Key Points to Note	524
Multiple Sync Signals/Intervals	524
Testbench Phase Synchronization	525
Testbench Phase Synchronization Based on User Task	526
UVM Testbench Phase Synchronization	526
Key Points to Note	527
Example	528
Testbench Data Transfer of Class Objects	529
Save Replay Support	532
Save Restore Support	534
Save	535
Restore	537
Limitations	538
User Task to Print Class Objects	538
VCD Dump per Client	538
Bidirect Support Through Wired OR/AND	539
Limitations	541
Include Syntax in Configuration File	542
Configuration File Syntax	542
Server Launch as Independent Process	542
HDL Access Across Simvs from Verilog Source	543
Distsim Release Recv Call for TB Data Transfer	545
TB Data Open Channels	545
Distributed Simulation Profile	546
Configuration File Checker Utility	547
Specifying User-Defined TCP Port for Server Launch	547
Key Points to Note	548
Specifying Early Simulation Finish Mechanism Across Simvs	548
Cygnus PDF Profile Summary	548
TB Sync Comp Class	548
Packed Structure Support	549
Key Points to Note	549
Constant Support in the Configuration File	549

Contents

Support for Multiple TB Data Recv Requests	549
Compile Time Configuration File Support	550
Key Points to Note	551
Distsim Flow Without Shared File System	551
Early Finish Behavior	552
Limitations	553
Quick Reference: VCS Distributed Simulation Options	553
<hr/>	
11. Performance Tuning	557
Compile-time Performance	558
Incremental Compilation	558
Compile Once and Run Many Times	559
Parallel Compilation	559
Improving VCS Compile Performance and Capacity	559
Use Model	560
Runtime Performance	560
Using Radiant Technology	561
Compiling With Radiant Technology	561
Applying Radiant Technology to Parts of the Design	561
Improving Performance When Using PLIs	568
Use Model	569
Limitations	571
Enabling TAB File Capabilities in UCLI Using -debug_access	571
Use Model	572
Example	572
Impact on Performance	573
Obtaining VCS Consumption of CPU Resources	574
Use Model	574
Compile Time	574
Simulation Time	575
Dumping Design Statistics	575
Use Model	576
Usage Example	576
Limitations	578
<hr/>	
12. Using X-Propagation	579
Introduction to X-Propagation	579

Contents

Guidelines for Running X-Propagation Simulations	580
Using the X-Propagation Simulator	582
Runtime Options	584
Specifying X-Propagation Merge Mode	585
Compile Time Diagnostic Report	587
Querying X-Propagation at Runtime	588
X-Propagation Instrumentation Report	589
Automatic Hardware Inference of Flip-Flops Enabled by Default	590
X-Propagation Configuration File	591
X-Propagation Configuration File Syntax	592
Xprop Instrumentation Control	595
Limitation	597
Disabling Xprop on Library Cells	597
Process Based X-Propagation Exclusion	598
Support for XIndex Element Merging	599
X-Index	600
Index BSpace	600
Addressing Models	601
Merge Modes	602
Index Selection Methods	602
Disabling XIndex Merging for Read or Write Operations	603
Use Model	604
Examples	605
Limitations	608
Bounds Checking	608
Detecting Unknown Values in Type Conversion Functions	609
Time Zero Initialization	609
Enabling X-propagation on RTL Block Containing Asserts Having System Task Usage	609
Handling Non-pure Functions Due to Static Lifetime	610
Supporting UCLI Commands for X-Propagation Control Tasks	611
Use Model	611
UCLI Command to Specify the Merge Mode	611
UCLI Command to Control Error Messages or Warning Messages	612
VHDL Two-State Objects in X-Propagation	613
Supported XValues	613
Supported Objects	613
Supported Expressions	614
Supported Attributes	618
Limitations	618

Contents

X-Propagation Code Examples	619
If Statement	619
Verilog Example	619
VHDL Example	620
Case Statement	621
Verilog Example	621
VHDL Example	622
Unique/Priority Case Variants	624
Edge Sensitive Expression	625
Verilog Example	625
VHDL Example	626
Latch	627
Verilog Example	627
VHDL Example	628
Support for Active Drivers in X-Propagation	629
Combinational Logic	629
Latches	631
Flip-flops	633
Key points to Note	635
Support for Ternary Operator	635
Renaming xprop.log File	635
Limitations	636
Limitations	636
<hr/>	
13. Gate-Level Simulation	639
SDF Annotation	639
Using the Unified SDF Feature	640
Compilation	640
Simulation	640
Analysis	640
Elaboration	640
Simulation	640
Using the \$sdf_annotation System Task	641
Using the -xlrn Option for SDF Retain, Gate Pulse Propagation, and Gate Pulse Detection Warning	642
Using the Optimistic Mode in SDF	643
Using Gate Pulse Propagation	644
Generating Warnings During Gate Pulses	644
Enhancing SDF Annotation to Support Nets Through SPICE	645

Contents

Use Model	645
Use Model	646
Limitations	647
Precompiling an SDF File	647
Creating the Precompiled Version of the SDF File	648
Precompiling SDF Without Compiling Design Files	648
Writing Precompiled SDF to a Different Directory	648
SDF Configuration File	649
Delay Objects and Constructs	649
SDF Configuration File Commands	650
The INTERCONNECT_MIPD Command	651
The MTM Command	651
The SCALE Commands	652
An SDF Example With Configuration File	652
Delays and Timing	655
Transport and Inertial Delays	655
The Inertial Delay Implementation	656
Enabling Transport Delays	657
Pulse Control	657
Pulse Control With Transport Delays	658
Pulse Control With Inertial Delays	660
Specifying Pulse on Event or Detect Behavior	663
Specifying the Delay Mode	665
Support for Delayed Annotation When No Conditions Have Matched	667
Support for Delay Selection Options for Recent Active Paths by Ignoring Condition Checking	668
Specifying Timing Control Attributes in the +optconfigfile File	670
Using the Configuration File to Disable Timing	671
Using the timopt Timing Optimizer	672
Editing the timopt.cfg File	673
Editing Potential Sequential Device Entries	673
Editing Clock Signal Entries	674
Using Scan Simulation Optimizer	674
ScanOpt Configuration File Format	675
ScanOpt Assumptions	676
Combinational Path Delays	676
Length of Test Cycles	676
Improving the ScanOpt for Debug Support	676
Use Model	677

Contents

Usage Example	677
Negative Timing Checks	679
The Need for Negative Value Timing Checks	680
The \$setuphold Timing Check Extended Syntax	684
Negative Timing Checks for Asynchronous Controls	686
The \$recrem Timing Check Syntax	686
Enabling Negative Timing Checks	687
Other Timing Checks Using the Delayed Signals	688
IOPATH Delay Annotation Using Delayed Signals	691
Checking Conditions	692
Toggling the Notifier Register	693
SDF Back-Annotation to Negative Timing Checks	693
How VCS Calculates Delays	693
Using VITAL Models and Netlists	695
Validating and Optimizing a VITAL Model	695
Validating the Model for VITAL Conformance	696
Verifying the Model for Functionality	696
Optimizing the Model for Performance and Capacity	696
Re-Verifying the Model for Functionality	697
Understanding Error and Warning Messages	697
Distributing a VITAL Model	698
Simulating a VITAL Netlist	699
Applying Stimulus	699
Overriding Generic Parameter Values	699
Understanding VCS Error Messages	700
Viewing VITAL Subprograms	701
Timing Back-annotation	701
VCS Naming Styles	701
Negative Constraints Calculation (NCC)	701
Simulating in Functional Mode	702
Understanding VITAL Timing Delays and Error Messages	703
Negative Constraint Calculation (NCC)	703
Conformance Checks	703
Error Messages	706
Additional Diagnostics	716
Support for Identifying Non-Annotated Timing Arc and Timing Check Statements	716
Usage Example	716
14. Coverage	720

Contents

Code Coverage	720
Functional Coverage	721
Options For Coverage Metrics	721
<hr/>	
15. Using OpenVera Native Testbench	722
Usage Model	723
Example	723
Usage Model	726
Importing VHDL Procedures	727
Exporting OpenVera Tasks	728
Using Template Generator	729
Example	730
Key Features	738
Multiple Program Support	739
Configuration File Model	739
Configuration File	739
Usage Model for Multiple Programs	740
NTB Options and the Configuration File	741
Class Dependency Source File Reordering	742
Circular Dependencies	743
Dependency-based Ordering in Encrypted Files	744
Using Encrypted Files	744
Functional Coverage	744
Using Reference Verification Methodology	745
Compilation	745
Simulation	745
Analysis	745
Limitations	746
<hr/>	
16. Using SystemVerilog	747
Use Model	747
Using UVM With VCS	748
Update on UVM-1.2	749
Update on UVM-ieee	749
Update on UVM-ieee-2020	749
Update on UVM-ieee-2020-2.0	749
Natively Compiling and Elaborating UVM-1.1d	750
Natively Compiling and Elaborating UVM-1.2	750

Contents

Natively Compiling and Elaborating UVM-ieee	751
Natively Compiling and Elaborating UVM-ieee-2020	751
Compiling the External UVM Library	751
Using the -ntb_opts uvm Option	752
Explicitly Specifying UVM Files and Arguments	752
Accessing HDL Registers Through UVM Backdoor	753
Generating UVM Register Abstraction Layer Code	753
Recording UVM Transactions	754
Recording UVM Phases	754
UVM Template Generator	755
Using Mixed VMM/UVM Libraries	755
Migrating from OVM to UVM	757
Where to Find UVM Examples	757
Where to Find UVM Documentation	758
UVM-1.1d Documentation	758
UVM-VMM Interop Documentation	758
Using VMM with VCS	758
Using OVM with VCS	759
Native Compilation and Elaboration of OVM 2.1.2	759
Compiling the External OVM Library	760
Using the -ntb_opts ovm Option	760
Explicitly Specifying OVM Files and Arguments	760
Recording OVM Transactions	761
Debugging SystemVerilog Designs	762
Functional Coverage	762
SystemVerilog Constructs	763
Extern Task and Function Calls through Virtual Interfaces	764
Modport Expressions in an Interface	766
Limitations	767
Interface Classes	768
Difference Between Extends and Implements	769
Cast and Interface Class	771
Name Conflicts and Resolution	772
Interface Class and Randomization	774
Package Exports	775
Severity System Tasks as Procedural Statements	776
Width Casting Using Parameters	777
The std::randomize() Function	779
Syntax	779

Contents

Description	779
Example	780
SystemVerilog Bounded Queues	780
wait() Statement with a Static Class Member Variable	781
Support for Consistent Behavior of Class Static Properties	782
Parameters and Local Parameters in Classes	783
SystemVerilog Math Functions	783
Streaming Operators	784
Packing (Used on RHS)	784
Unpacking (Used on LHS)	785
Packing and Unpacking	785
Propagation and force Statement	785
Error Conditions	785
Structures with Streaming Operators	785
Support for with Expression	786
Constant Functions in Generate Blocks	788
Support for Aggregate Methods in Constraints Using the “with” Construct ..	789
Debugging During Initialization SystemVerilog Static Functions and Tasks in Module Definitions	790
Example	791
Explicit External Constraint Blocks	792
Using an Empty Constraint Block	794
Generate Constructs in Program Blocks	795
Error Condition for Using a Genvar Variable Outside of its Generate Block ..	796
Randomizing Unpacked Structs	797
Using the Scope Randomize Method <code>std::randomize()</code>	797
Using the Class Randomize Method <code>randomize()</code>	800
Disabling and Re-enabling Randomization	802
Using In-Line Random Variable Control	804
Limitation	807
Using a Package in a SystemVerilog Module, Program, and Interface Header	807
Disabling DPI Tasks and Functions	809
Use Model	809
Support for Overriding Parameter Values through Configuration	809
Example	810
Precedence Override Rules	810
Limitations	811
Support for Inclusion of Dynamic Types in Sensitivity List	811
Usage Example	812
Support for Assignment Pattern Expression in Non-Assignment Like Context	812

Contents

Usage Example	813
User-Defined Nettypes	813
The Resolution Function	814
Limitations	814
Example of User-Defined Nettype	814
Example of User-Defined Nettype in Arrays	815
Example of Nettype MDAs of Type Real	816
Example of Nettype MDAs of Type Unpacked Struct	817
Support for Connecting Nettypes through Tranif Gates	817
Limitations	818
Generic Interconnect Nets	818
Limitations	819
Support for Associative Array With Unpacked Structure as Key	819
Limitation	819
Specifying a SystemVerilog Keyword Set by LRM Version at Command Line	819
The -sv Compile-Time Option	820
The `begin_keywords and `end_keywords Compiler Directives	821
Support for .triggered Property with Clocking Block Name	822
Usage Examples	822
Support for Intra Assignment Delay With Non-Blocking Assignments in Program Block	823
Limitations	823
Support for Array Query Functions	823
Array Query Functions With Associative Array	823
Limitations	824
Support for Nested Randsequence	824
Use Model	824
Usage Example	824
Support for Typed Constructor Call as an Argument to Task or Function	825
Usage Example	825
Limitations	826
Support for Select of Unpacked Structure Array as an Argument to Task or Function	826
Usage Example	826
Limitation	826
Support for Function Returning Unpacked Structure in Conditional Operator in a Continuous Assignment Statement	827
Usage Example	827
Support for Built-in Method Which Returns Work Library Name	828
Usage Example	828
Limitations	829

Contents

Support for Function call in Clocking Block Hierarchical Expression	829
Usage Example	830
Limitations	831
Extensions to SystemVerilog	831
Unique/Priority Case/IF Final Semantic Enhancements (-xlmr uniq_prior_final Compile-Time Option)	832
Using Unique/Priority Case/If with Always Block or Continuous Assign ..	832
Using Unique/Priority Inside a Function	835
System Tasks to Control Warning Messages	837
LRM Compliant Behavior for the atohex Method	838
Controlling Runtime Warning Messages Generated Using Unique/Priority If Constructs	838
Support for Unique0 in Conditional Statements	839
Usage Example	840
Enhancements to the -xlmr uniq_prior_final Compile-Time Option	841
Single-Sized Packed Dimension Extension	845
Covariant Virtual Function Return Types	846
Self Instance of a Virtual Interface	847
UVM Example	849
Support for Shuffle Method for Multi-Dimensional Arrays	849
Use Model	850
Usage Example	850
Enhanced Clocking Block Behavior When Skew is negedge/posedge	851
Usage Example	852
Support for Slice of String Variable	854
Support for Randomization of Floating Point Variables	854
Use Model	854
Usage Example	854
Support for Unpacked Array Concatenation in the HighConn of Inout Port ..	855
Use Model	855
Usage Example	856
Limitation	856
Support for Index Locator Methods for Multi-Dimensional Arrays	856
Use Model	856
Usage Example	857
Limitation	857
Support for String Method Substr() with a Single Argument	857
Usage Example	858
Support for Assignment Pattern with Single-Bit Scalar Nets or Variables ..	858
Use Model	858
Usage Example	858

Contents

Limitations	859
Support for Reading Value of any Clock Variable	859
Use Model	859
Usage Example	859
<hr/>	
17. Aspect Oriented Extensions	860
Aspect-Oriented Extensions in SystemVerilog	861
Processing of Aspect-Oriented Extensions as a Precompilation Expansion	862
Weaving Advice Into the Target Method	866
Precompilation Expansion Details	869
Precedence	869
Adding of Introductions	871
Weaving of advices	871
Symbol Resolution Details	878
The hide_list Details	880
<hr/>	
18. Using Constraints	888
Support for Array Slice in Unique Constraints	889
Limitation	889
Support for Object Handle Comparison in Constraint Guards	890
Limitations	893
Support for Pure Constraint Block	893
Support for SystemVerilog Bit Vector Functions in Constraints	897
\$countones Function	899
\$onehot Function	899
\$onehot0 Function	900
\$countbits Function	901
\$bits Function	902
Inconsistent Constraints	903
Constraint Debug	904
Partition	905
Randomize Serial Number	906
Solver Trace	907
Constraint Profiler	910
Test Case Extraction	910
Using multiple +ntb_solver_debug arguments	912

Contents

Summary for the +ntb_solver_debug Option	912
+ntb_solver_debug=serial	912
+ntb_solver_debug=trace	912
+ntb_solver_debug=profile	912
+ntb_solver_debug=extract	912
+ntb_solver_debug=verbose	912
Support for Save and Restore Stimulus	913
Use Model	913
Limitations	914
Constraint Guard Error Suppression	914
Error Message Suppression Limitations	915
Flattening Nested Guard Expressions	915
Pushing Guard Expressions into Foreach Loops	916
Support for Array and Cross-Module References in std::randomize()	916
Syntax	917
Example	917
Error Conditions	917
Support for Cross-Module References in Constraints	918
XMR Function Calls in Constraints	919
State Variable Index in Constraints	920
Runtime Check for State Versus Random Variables	920
Array Index	920
Using DPI Function Calls in Constraints	920
Invoking Non-pure DPI Functions from Constraints	921
Using Foreach Loops Over Packed Dimensions in Constraints	924
Memories with Packed Dimensions	924
Single Packed Dimension	924
Multiple Packed Dimensions	925
MDAs with Packed Dimensions	925
Single Packed Dimension	925
Multiple Packed Dimensions	925
Just Packed Dimensions	925
The foreach Iterative Constraint for Packed Arrays	926
Randomized Objects in a Structure	927
Support for Typecast in Constraints	928
Syntax	929
Description	929
Examples	929

Contents

Strings in Constraints	931
Support for Dynamic MDA Elements in the Randomize Call Arguments	931
Supported Dynamic MDA Types	932
Key Points to Note	933
Supported Usage Scenarios	933
Limitations	934
Support for Dynamic MDA in the Inside Expression Used in Constraints	934
Supported Dynamic MDA Types	934
Supported Usage Scenarios	935
Limitations	936
Support for Array Reduction Methods over MDAs in Constraints	936
Limitations	937
SystemVerilog LRM 1800™-2012 Update	937
Using Soft Constraints in SystemVerilog	938
Using Soft Constraints	938
Soft Constraint Prioritization	939
Soft Constraints Defined in Classes Instantiated as rand Members in Another Class	940
Soft Constraints Inheritance Between Classes	941
Soft Constraints in AOP Extensions to a Class	941
Soft Constraints in View Constraints Blocks	944
Discarding Lower-Priority Soft Constraints	944
Unique Constraints	948
Enhancement to the Randomization of Multidimensional Array Functionality	949
Limitation	950
Supporting Random Array Index	951
Random Array Index Enhancements	951
Function Call in Array Index Expression	952
Dynamic MDA With Loop Index Variable in Array Index Expression	952
Random Array Indexed Sub-Array In Unique/Inside Constraints	953
Limitation	954
Supporting System Function Calls	954
\$size() System Function Call	954
\$clog2() System Function Call	955
Usage Example	955
Supporting Foreach Loop Iteration over Array Select	956
Support for Enumerated Type Methods in a Constraint Expression	957
Usage Example	957

Contents

Limitation	957
Improved Support for Function Evaluation in Constraints	958
<hr/>	
19. VCS Intelligent Coverage Optimization (ICO)	960
<hr/>	
20. Extensions for SystemVerilog Coverage	963
Support for Reference Arguments in get_coverage() and get_inst_coverage() ..	963
get_coverage() method	963
get_inst_coverage() method	964
<hr/>	
21. OpenVera-SystemVerilog Testbench Interoperability	965
Scope of Interoperability	966
Importing OpenVera Types Into SystemVerilog	966
Data Type Mapping	968
Mailboxes and Semaphores	969
Events	970
Strings	971
Enumerated Types	971
Integers and Bit-Vectors	973
Arrays	973
Structs and Unions	975
Connecting to the Design	975
Mapping Modports to Virtual Ports	975
Virtual Modports	975
Importing Clocking Block Members Into a Modport	976
Semantic Issues With Samples, Drives, and Expects	980
Notes to Remember	980
Blocking Functions in OpenVera	980
Constraints and Randomization	980
Functional Coverage	981
Usage Model	982
For Two-Step Flow:	982
Compilation	982
Simulation	982
Three-Step Flow	982
Elaboration	982

Contents

Simulation	982
Limitations	983
<hr/>	
22. Using SystemVerilog Assertions	984
Using SVAs in the HDL Design	985
Using VCS Checker Library	986
Instantiating SVA Checkers in Verilog	986
Instantiating SVA Checkers in VHDL	987
Binding SVA to a Design	988
Compilation	989
Simulation	989
Binding SVA to a Design in VCS	989
bind Statements In Your Verilog Source Files	990
Encapsulating Bind Statements	990
Supported Data Types	991
Extensions	992
Salient Features	992
Limitations	993
Inlining SVAs in the Verilog Design	993
Use Model	994
Inlining SVA in the VHDL design	995
Use Model	995
Number of SystemVerilog Assertions Supported in a Module	996
Controlling SystemVerilog Assertions	997
Compilation/Elaboration and Runtime Options	997
Concatenating Assertion Options	999
Assertion Monitoring System Tasks	999
Using Assertion Categories	1002
Using System Tasks	1002
Using Attributes	1003
Starting and Stopping Assertions Using Assertion System Tasks	1004
Viewing Results	1007
Using a Report File	1008
Enhanced Reporting for SystemVerilog Assertions in Functions	1008
Introduction	1008
Use Model	1009
Name Conflict Resolution	1010
Checker and Generate Blocks	1010

Contents

Controlling Assertion Failure Messages	1010
Introduction	1010
Options for Controlling Default Assertion Failure Messages	1011
Using <code>-assert no_default_msg[=SVA OVA PSL]</code> Option	1011
Using <code>-assert quiet</code> and <code>-assert quiet1</code> Options	1012
Options to Control Termination of Simulation	1012
Option to Enable Compilation of OVA Case Pragmas	1014
Reporting Values of Variables in the Assertion Failure Messages	1014
Limitations	1015
Using <code>\$uniq_prior_checkon</code> and <code>\$uniq_prior_checkoff</code> System Tasks	1016
Reporting Messages When <code>\$uniq_prior_checkon/\$uniq_prior_checkoff</code> System Tasks are Called	1016
Assertion and Unique/Priority Re-Trigger Feature	1018
Flushing Off the Assertion Re-Trigger Feature	1019
Enabling Lint Messages for Assertions	1020
Fail-Only Assertion Evaluation Mode	1022
Key Points to Note	1023
Limitations	1024
Treating <code>x</code> as true on an Assertion Precondition	1024
Use Model	1025
Usage Example	1025
Using SystemVerilog Constructs Inside <code>vunits</code>	1026
Limitations	1027
Calling <code>\$error</code> Task When Else Block is Not Present	1027
Disabling Default Assertion Success Dumping in <code>-debug_access</code> Option	1028
Support for Success Count With <code>-assert summary</code> Option by Default	1028
Use Model	1028
Usage Example	1028
Disabling Success Callbacks	1029
Use Model	1030
Enhancement to Assertion Diagnostics	1030
Use Model	1030
Usage Example	1031
Support of String Variable to Assertion Control System Tasks	1032
Use Model	1032
Example	1032

Contents

Limitation	1034
Support for Automatic Variables in Sampling Functions	1034
Use Model	1034
Usage Examples	1034
Limitation	1035
List of supported IEEE Std. 1800-2012 Compliant SVA Features	1035
Support for \$countbits System Function	1038
Support for Real Data Type Variables	1038
Support for \$assertcontrol Assertion Control System Task	1038
Limitations	1039
Enabling IEEE Std. 1800-2012 Compliant Features	1039
Limitations	1039
Support for Strong Operators in Assertions	1039
Suppressing the Run-time out of Bound Access Messages	1041
Use Model	1041
Example	1041
Output	1042
Reporting Runtime Violations for Unique/Unique0/Priority For and Foreach Statements	1042
Example	1043
Limitation	1044
Using -assert errmsg Runtime Option	1044
Typed Formal Support for Sequence and Property Using -assert typed_formal Option	1044
Disabling Sequence Debugging Using -assert no_seqdebug Option	1047
Assertion Summary Output With -assert global_finish_maxfail=<N> and -error=SVART-NAFRLTS Options	1049
SystemVerilog Assertions Limitations	1052
Debug Support for New Constructs	1052
Note on Cross Features	1052
<hr/> 23. Using Property Specification Language	1053
Including PSL in the Design	1053
Examples	1053
Use Model	1054
Examples	1055

Contents

Examples	1056
PSL Assertions Inside VHDL Block Statements in Vunit	1056
Introduction	1056
Example	1057
Use Model	1057
Limitations	1058
PSL Macro Support in VHDL	1058
Using the %for Construct	1058
Using the %if Construct	1060
Using Expressions with %if and %for Constructs	1061
PSL Macro Support Limitations	1061
Using SVA Options, SVA System Tasks, and OV Classes	1062
Limitations	1063
<hr/>	
24. Using SystemC	1064
<hr/>	
25. Dynamic Test Loading	1066
Introduction	1066
Advantages of DTL	1066
How DTL Works	1067
Use Model	1068
Dynamic Test Loading Modes	1071
Regression Model	1071
Development Model	1072
Guidelines for Using DTL	1073
Debug Support for Dynamic Test Loading	1074
Limitations	1074
<hr/>	
26. C Language Interface	1075
Using PLI	1075
Writing a PLI Application	1076
Functions in a PLI Application	1077
Header Files for PLI Applications	1077
PLI Table File	1078

Contents

Syntax	1078
Using the PLI Table File	1091
Enabling ACC Capabilities	1091
Using the Configuration File	1092
Selected ACC Capabilities	1094
Using VPI Routines	1097
Support for VPI Callbacks	1098
Integrating a VPI Application With VCS	1098
PLI Table File for VPI Routines	1099
Virtual Interface Debug Support	1100
Example	1100
Limitations	1102
Unimplemented VPI Routines	1102
Modified VPI Features	1103
Example	1104
Error Message	1104
Solution	1105
Example	1106
Warning Message	1106
Solution	1106
Using VHPI Routines	1106
Diagnostics for VPI/VHPI PLI Applications	1106
Using DirectC	1107
Using Direct C/C++ Function Calls	1108
Functioning of C/C++ Code in a Verilog Environment	1109
Declaring the C/C++ Function	1110
Calling the C/C++ Function	1114
Storing Vector Values in Machine Memory	1115
Converting Strings	1117
Avoiding a Naming Problem	1119
Using Pass by Reference	1120
Using Direct Access	1120
Example 1	1122
Example 2	1123
Example 3	1123
Example 4	1123
Example 5	1124
Example 6	1124
Example 7	1124
Example 8	1125
Using the vc_hdrs.h File	1125

Contents

Access Routines for Multi-Dimensional Arrays	1126
Using Abstract Access	1127
Using vc_handle	1127
Using Access Routines	1128
Summary of Access Routines	1158
Enabling C/C++ Functions	1163
Mixing Direct And Abstract Access	1164
Specifying the DirectC.h File	1164
Extended BNF for External Function Declarations	1164
Using DPI Open Array	1165
Using Multi-Dimensional Array as an Actual Argument for DPI Function	1165
Limitations	1167
<hr/>	
27. Support for VHDL 2002, 2008 and 2019	1168
VHDL 2002 Protected Type	1168
Use Model	1168
Limitations of VHDL 2002 Protected Type	1169
VHDL 2008 Constructs	1169
Array Types and Operators	1170
Adding Comments	1171
Use Clause and Aliases	1171
Support for Bit String Literals	1172
Support for TO_STRING Conversion	1174
Support for External Names	1176
Specifying The all Keyword in the Process Sensitivity List	1177
Support for Logical Unary Reduction Operator	1177
Support for Matching Relational Operators for Bit and std_ulogic	1178
Including Non-Static Expressions in Port Map	1179
Standard Environment Package	1180
Package Declaration and Instantiations	1180
Limitation	1183
Referencing Interface Lists	1183
Limitations	1184
Overriding the Value Assigned to a Signal	1186
Forcing and Releasing Values of Signals	1187
Forcing and Releasing Ports of a Design	1188
Assigning Composite Value to a Collection of Signals	1189
Forcing and Releasing Assignment Written in a Subprogram	1189

Contents

Forcing and Releasing Multiple Concurrent Assignments	1189
Debugging the Force and Release Assignments	1190
Matching Case Statements	1190
Conditional Elaboration	1192
Condition Operator in an Expression	1198
Reading Output Port	1200
2008 IEEE Packages	1202
Overview of Additional IEEE Packages	1202
Resolved Elements	1206
Usage Example	1207
Conditional and Selected Assignments	1210
Use Model	1211
Usage Example	1212
Context Declaration	1215
Usage Example	1215
Improved I/O	1217
Usage Example	1218
Support for Implicitly Constrained Array Elements	1220
Usage Example	1220
Support for Unconstrained Element Types	1221
Usage Example	1223
Support for Enhanced Generics in Entity Interfaces	1226
Usage Example	1226
Limitation	1232
Support for Slices in Array Aggregates	1232
Usage Example	1232
Limitation	1235
Support for Type Conversion in VHDL 2008	1236
Usage Example	1236
Limitation	1237
Support for Case Expression Subtype	1237
Example for Case Expression	1238
Support for Subtypes of Ports and Parameters	1241
Example for Subtypes of Ports and Parameters	1241
Support for Static Composite Expressions	1242
Usage Examples	1243
Support for VHDL 2008 Enhanced Generics in Components	1249
Examples	1250
Support for VHDL 2008 Local Packages	1255
Limitation	1257

Contents

Support for Unconstrained Elements in Generics and Generic Types	1258
Limitations	1259
VHDL 2019 Conditional Analysis Tool Directives	1259
Use Model	1259
Usage Example	1260
Limitations	1262
<hr/>	
28. SAIF Support	1263
Using SAIF Files with VCS	1263
SAIF System Tasks for Verilog or Verilog-Top Designs	1264
The Flows to Generate a Backward SAIF File	1266
Generating an SDPD Backward SAIF File	1266
Generating a Non-SPDP Backward SAIF File	1267
SAIF Calls That Can Be Used on VHDL or VHDL-Top Designs	1267
SAIF Support for Two-Dimensional Memories in v2k Designs	1268
UCLI SAIF Dumping	1268
Criteria for Choosing Signals for SAIF Dumping	1269
Improving Simulation Time by Reducing the Overhead due to SAIF File Dumping	1269
Use Model	1269
Example	1270
Limitations	1271
<hr/>	
29. Encrypting Source Files	1272
IEEE Verilog Standard 1364-2005 Encryption	1272
The Protection Header File	1273
Unsupported Protection Pragma Expressions	1275
Other Options for IEEE Std 1364-2005 Encryption Mode	1275
How Protection Envelopes Work	1276
VCS Public Encryption Key	1277
Creating Interoperable Digital Envelopes Using VCS - Example	1278
Discontinued -ipkey Option	1281
IEEE VHDL Standard 1076-2008 Encryption	1282
VHDL 1076-2008 Encryption Use Model	1282
Encrypting the Entire VHDL Source Files	1283
Encrypting the Parts of VHDL Source Files	1283

Contents

Using the Protection Header File	1283
Options for VHDL 1076-2008 Encryption Mode	1285
Protection Envelopes	1286
The VCS Public Encryption Key	1286
Usage Example	1287
Example for Full Encryption	1287
Example for Partial Encryption	1288
Debug Protection	1289
Combining Encrypted and Unencrypted Code	1290
Assertion and Report Statements	1291
Hierarchy Attributes	1292
Profiling	1292
VHPI	1292
VPD / VCD	1292
Coverage	1292
Error Messages	1292
Limitations	1292
128-bit Advanced Encryption Standard	1293
Compiler Directives for Source Protection	1293
Using Compiler Directives or Pragmas	1294
-protect128	1294
Example	1295
Automatic Protection Options	1297
Using Automatic Protection Options	1298
-autoprotect128	1299
-auto2protect128	1299
-auto3protect128	1300
+protect option	1301
+putprotect+<Dir-name>	1301
+autoprotect[file_suffix]	1302
+auto2protect[file_suffix]	1302
+auto3protect[file_suffix]	1303
+deleteprotected	1303
+pli_unprotected	1304
Protecting ‘include File Directive	1304
+autoincludeprotect	1304
Enabling Debug Access to Ports and Instance Hierarchy	1305
+autobodyprotect	1305
Debugging Partially Encrypted Source Code	1305
Skipping Encrypted Source Code	1305
gen_vcs_ip	1306

Contents

Syntax	1307
Analysis Options	1307
Exporting The IP	1308
Use Model	1308
IP Vendor	1308
IP Generation	1309
IP User	1309
Licensing	1309
<hr/>	
30. VCS Fine-Grained Parallelism Technology	1310
Introduction	1310
Use Model	1312
Compile Option for -fgp	1314
Runtime Options for -fgp	1314
Profiling to Detect Design Suitability for Parallelism	1318
Limitations	1319
<hr/>	
31. Integrating VCS With Certitude	1320
Introduction to Certitude	1320
VCS and Certitude Integration	1320
Loading Designs Automatically in Verdi with Native Certitude	1321
Use Model	1322
Points to Note	1322
Dumping and Comparing Waveforms in Verdi for SystemC Designs	1323
Use Model	1323
Point to Note	1324
Reducing Compilation Time in Native Certitude With VCS Partition Compile Flow	1324
Use Model	1325
Example	1326
Limitation	1327
<hr/>	
32. Integrating VCS with Vera	1328
Setting Up Vera and VCS	1328
Using Vera with VCS	1329

Contents

Usage Model	1329
Two-Step Flow	1330
Three-Step Flow	1330
33. VCS Mixed-Signal Simulation	1331
Introduction to VCS and CustomSim	1332
Analyzing a Design	1333
Elaborating a Design	1333
Running the Simulation	1333
Setting up the Environment	1333
Licenses	1333
Required UNIX Paths and Variable Settings	1334
Use Model	1334
Example	1335
Scheduling Analog-to-Digital Events in the NBA Region	1336
Use Model	1336
Support of Verilog Force and Release Assignments on Wreal Nets	1336
Usage Example	1337
Limitations	1338
Support for Wreal Nets in Verilog-AMS Flow	1338
Usage Example	1338
Support for SystemC Designs in Verilog-AMS	1339
Use Model	1340
Usage Example	1340
Support for Wildcard Character and the -exclude Option in the Mixed Signal Control Command	1343
Use Model	1343
Usage Example	1344
Limitation	1344
Enhancement to the force Command to Force SPICE Bus Ports	1344
Terminology	1345
Inclusions in the MSV Control File	1345
Examples	1346
UCLI force/release	1346
Verilog force/release Procedural Statements	1348
\$hdl_xmr_force/\$hdl_xmr_release Verilog System Tasks	1349
HDL_XMR_FORCE/HDL_XMR_RELEASE VHDL Sub-Programs	1353

Contents

Bit-Select/Part-Select	1357
Limitations	1357
VCS with AMS Integration	1357
Integrating VCS With AMS	1358
Setting up the Environment	1358
Use Model	1358
Analyzing all VHDL, Verilog, Verilog-AMS Files	1358
Elaborating the Design	1358
Simulating the Design	1359
Limitations in VCS-AMS Flow	1359
Support for Predefined Nettypes in SystemVerilog-SPICE Flow	1359
Predefined Nettypes	1360
Use Model	1362
User-Defined Nettypes	1364
Limitations	1367
Support for SystemVerilog Nettypes in Mixed-Signal Simulation	1367
Type Compatibility Between Nettype and VHDL Type/Subtype	1368
Editing the Setup File	1368
Rules for Type Compatibility	1368
Example 1	1369
Example 2	1370
Limitations	1372
Using SystemVerilog Nettypes in Mixed-Signal Simulation	1372
Supported Topologies	1372
Case 1 - SystemVerilog Testbench With SPICE and VHDL Children	1372
Case 2 - SPICE Testbench With VHDL and SystemVerilog Children	1373
Use Model	1373
Example	1374
Limitations	1377
Support for SystemVerilog Packed Array at Mixed Design Boundary	1377
Editing the Setup File	1378
Key Points to Note	1378
Verilog-on-Top Scenario	1379
VHDL-on-Top Scenario	1379
Limitations	1380
Support for Unmapped User-Defined Nettypes in SystemVerilog-SPICE Flow ..	1380
Use Model	1381
Limitation	1383

Contents

Support for User-Defined Nettypes in Mixed-Signal Simulation	1383
Use Model	1383
Mixed-Signal Control Command	1383
Usage Example	1384
Limitations	1386
Enhancements in Type Coercion and Converter Insertions	1386
Enhancements for Type Coercion	1386
Support for Basic Type-Coercion	1386
Enhancement to Type Coercion Issues for SV Interface	1387
Enhancements in Diagnostics and Report Generation	1387
Enhancements to Converter Insertions	1387
Support for Merge Mode Converter Insertion	1387
Quality Enhancements for Converter Insertion	1388
Options for Coercion Flow	1389
Enhancements to Event Functions in SV-SPICE Flow	1390
Introduction	1390
Enhancements to Event Functions	1390
Examples	1391
Example for the <code>snps_cross</code> Event Function	1391
Example for the <code>snps_above</code> Event Function	1391
Example for the <code>snps_absdelta</code> Event Function	1391
Support for UPF Power-Aware Interface Elements in Mixed Signal Simulation Environment	1391
Association of UPF Supplies to Interface Elements	1392
Boundary SPICE Port With SPA Specified in the UPF	1392
Boundary SPICE Port With SRSN Specified in the UPF	1392
SPICE Cell With Liberty Information	1393
SPICE Cell Without SPA, SRSN, and Liberty Information	1393
Examples	1393
Example-1: Input Case	1393
Example-2: Output Case	1395
Real Number Modeling With VIZ Nettypes	1397
Examples	1398
KCL in Resolution Function for Voltage Calculation	1398
Using KCL for VIZ Models	1399
Library of Predefined VIZ Models and Resolution Function	1400
Connectivity Technologies in VCS	1400
Interconnect Versus Wire in System Verilog LRM	1400
Type-Coercion	1401

Contents

Detailed Coercion	1401
Basic Coercion	1402
Coercion in Wire Versus SV-Interconnect	1403
Command-Line Options for Coercion	1403
VCS Converters	1404
Local Converters	1405
Converter Insertion	1406
Converter Direction	1406
Use Model	1406
Usage Example	1407
Predefined Converter Modules	1409
Conversion Rules	1411
Conversion Rule for wreal*_to_wreal*_type	1411
Conversion Rule for logic_to_current_r/logic_to_real/logic_to_voltage_r	1412
Conversion Rule for logic_to_th_wire	1412
Conversion Rule for logic_to_v_wire*	1412
Conversion Rule for logic_to_wreal*	1413
Conversion Rule for real_to_logic/voltage_r_to_logic/ wreal*_to_logic/ current_r_to_logic	1413
Conversion Rule for th_wire_to_logic	1414
Conversion Rule for v_wire*_to_logic	1414
Requirements	1415
Support for Wreal Nets	1415
Introduction	1416
Supported Features	1416
Use Model	1419
Migrating From wreal Flow to nettype Flow	1419
Support for Converting Wreal to Nettype	1421
Use Model	1422
Limitations	1422
Support for Detail Discipline Resolution of Mixed Signals	1422
Detail Discipline Resolution Algorithm	1423
Use Model	1425
Usage Example	1425
Limitations	1427
34. Integrating VCS with Specman	1428
Type Support	1428
Usage Flow	1429

Contents

Setting Up The Environment	1430
Specman e code accessing VHDL only	1430
Specman e Code Accessing Verilog Only	1432
Specman e code accessing both VHDL and Verilog	1433
Guidelines for Specifying HDL Path or Tick Access with VCS-Specman Interface	1435
Using specrun and specview	1436
Version Checker for Specman	1437
Use Model	1437
Precedence Order	1438
<hr/>	
35. Integrating VCS with Denali	1439
Setting Up Denali Environment for VCS	1439
Integrating Denali with VCS	1439
Use Model	1440
Use Model for VHDL Memory Models	1440
Use Model for Verilog Memory Models	1441
Two-Step Flow	1441
Three-Step Flow	1441
Execute Denali Commands at UCLI Prompt	1442
<hr/>	
36. Integrating VCS With Native Low Power (NLP)	1443
<hr/>	
37. Unified UVM Library for VCS and Verdi	1444
Transaction/Message Recording in Verdi with VCS	1444
Compilation	1444
Enabling FSDB Transaction Recording	1444
Simulation	1445
Dumping Transactions or Messages in Verdi Flow	1445
<hr/>	
38. Debugging with Verdi	1447
Introduction	1447
Generating Verdi KDB	1448
Reading Compiled Design with Verdi	1449
Example	1451

Contents

Compiling Designs	1451
Setting Up Verdi	1451
Import KDB	1451
Invoke Verdi in Interactive Mode	1452
Invoke Verdi in Post-Processing Mode	1452
Key Points to Note	1452
Limitations	1453
Dumping FSDB File for Various Flows	1453
Setting Up Verdi	1453
Use Model for FSDB Dumping	1453
Using VHDL Procedures or Verilog System Tasks	1455
Using UCLI	1455
Examples	1455
VCS Two-step Flow	1456
VCS Three-step Flow	1456
VCS Two-step Flow	1456
VCS Three-step Flow	1457
Interactive and Post-Processing Debug	1457
Prerequisites	1458
Interactive Simulation Debug Flow	1458
Examples	1458
Simulation Control Commands	1460
Key Points to Note	1461
Post-Processing Debug Flow	1461
Reducing Disk Space for Post-Process Only Debug	1462
A. VCS Environment Variables	1463
Simulation Environment Variables	1463
Setup Variables	1464
Analysis Setup Variables	1464
Compilation/Elaboration Setup Variables	1466
Simulation Setup Variables	1468
C Compilation and Linking Setup Variables	1472
Timescale Implementation	1473
Understanding `timescale	1474
Verilog only and Verilog Top Mixed Design	1476
VHDL only and VHDL Top Mixed Designs	1476
Setting up Simulator Resolution From Command Line	1477
Other Useful Timescale Related Options	1478
Non-Compatible Options	1479

Contents

Limitations	1480
Optional Environment Variables	1480
Using Environment Variables in Verilog Source Code	1483
<hr/>	
B. Analysis Utilities	1484
The vhdlan Utility	1484
Using Smart Order	1488
Use Model	1489
Syntax:	1489
Example:	1489
Syntax:	1489
Example:	1489
Limitations	1490
The vlogan Utility	1490
<hr/>	
C. Compilation/Elaboration Options	1503
Option for Code Generation	1506
Options for Accessing Verilog Libraries	1506
Options for Incremental Compilation	1508
Option to Save Disk Space	1509
Options for Help	1509
Option for SystemVerilog	1509
Option to Enable Optimized Debug Capabilities and FSDB Dumping for Testbench Root Cause Analysis (TBRCA)	1509
Options for SystemVerilog Assertions	1509
Options to Enable Compilation of OVA Case Pragmas	1517
Options for Native Testbench	1517
Options for Different Versions of Verilog	1523
Option for Initializing Verilog Variables, Registers and Memories with Random Values	1524
Option for Initializing Verilog Variables, Registers and Memories in Selective Parts of a Design	1526
Options for Selecting Register or Memory Initialization	1529
Options for Using Radiant Technology	1529
Options for Starting Simulation Right After Compilation	1530

Contents

Options for Specifying Delays and SDF Files	1530
Options for Compiling an SDF File	1536
Options for Specify Blocks and Timing Checks	1536
Options for Pulse Filtering	1537
Options for Negative Timing Checks	1538
Options for Profiling Your Design	1539
Options to Specify Source Files and Compilation/Elaboration Options in a File	1539
Options for Compiling Runtime Options Into the Executable	1541
Options for PLI Applications	1541
Options to Enable the VCS DirectC Interface	1542
Options for Flushing Certain Output Text File Buffers	1543
Options for Simulating SWIFT VMC Models and SmartModels	1543
Options for Controlling Messages	1544
Option to Run VCS in Syntax Checking Mode	1548
Limitations	1549
Options for Cell Definition	1549
Options for Licensing	1550
Options for Controlling the Linker	1551
Options for Controlling the C Compiler	1552
Options for Source Protection	1554
Options for Mixed Analog/Digital Simulation	1554
Unified Option to Change Generic and Parameter Values	1555
Options for Changing Parameter Values	1555
Checking for x and z Values in Conditional Expressions	1556
Options for Detecting Race Conditions	1556
Options to Specify the Time Scale	1556
Option to Exclude Environment Variables During Timestamp Checks	1560
Options for Overriding Generics and Parameters	1560
Suppressing Reporting of Generic Override Messages	1564
Global -check_all Option	1564
Use Model	1565
Limitation	1566
Option to Enable Bounds Check at Compile-Time	1566

Contents

Runtime Checks for Out-of-Bounds Condition	1567
Error-[DT-OBAE] Out of Bounds Access for Queues	1567
Example	1567
Error-[DT-OBAE] Out of Bounds Access for Dynamic Arrays	1568
Example	1568
Warning-[AOOBAW] Array Out of Bounds Access for Fixed Size Unpacked Arrays	1568
Example	1568
Warning-[AOOBAW] Array Out of Bounds Access for Fixed Size Packed Arrays	1569
Example	1569
Error-[DT-OBAE] Intermediate Access for Dynamic Arrays	1570
Example	1570
Warning-[AAIIW] Array Access with Intermediate Index	1570
Example	1570
Warning-[AAIIW] Array Access with Intermediate Index for Fixed Size Packed Arrays	1571
Example	1571
Option to Enable Extra Runtime Checks in VHDL	1571
Error-[SIMERR_FDIVZERO_SCOPE] Divide by Zero Error	1572
Example:	1572
Error-[SIMERR_INCONSISTENTIC] Incorrect Binding Range	1572
Example	1572
Error-[SIMERR_INCONSISTENTIS] Subtype Constraints Inconsistencies	1573
Example	1573
Options to Detect Multiple Conflicts on Buses	1574
Use Model	1574
Outputs	1576
Usage Example	1577
Limitations	1583
Gate-Level Simulations	1584
Local Timescale Precision on Timing Check for Modules	1584
Use Model	1584
Using configuration file	1584
Using the <code>-auto_tchk_local_precision</code> option	1585
Usage Example	1585
Limitations	1586
Options for Additional Multiple Driver Checks	1586
Usage Example	1587

Contents

Default behavior:	1587
Behavior with <code>-varindex_drivers</code> option:	1587
Usage Example	1588
Default behavior:	1588
Behavior with the <code>-ntb_opts multi_driver_no_source_info</code> option:	1588
Option for Function Call Evaluation at Time Zero	1589
General Options	1590
Specifying Directories for ‘include’ Searches	1590
Enable the VCS/SystemC Cosimulation Interface	1590
TetraMAX	1591
Suppressing Port Coersion to inout	1591
Allow Inout Port Connection Width Mismatches	1591
Specifying a VCD File	1591
Enabling Dumping	1591
Enabling Identifier Search	1592
Specifying a Log File	1592
Changing Source File Identifiers to Upper Case	1592
Defining a Text Macro	1593
Undefining a Text Macro	1593
Option for Macro Expansion	1594
Specifying the Name of the Executable File	1594
Returning The Platform Directory Name	1595
Maximum Donut Layers for a Mixed HDL Design	1595
Enabling feature beyond VHDL LRM	1595
Enabling Loop Detect	1595
Changing the Time Slot of Sequential UDP Output Evaluation	1596
Gate-Level Performance	1596
Option to Omit Compilation of Code Between Pragmas	1597
Generating a List of Source Files	1598
Passing Options Starting With “-” to VCS in -R Flow	1599
Option for Dumping Environment Variables	1599
D. Simulation Options	1601
Options for Simulating Native Testbenches	1602
Options for SystemVerilog Assertions	1607
Options to Control Termination of Simulation	1614

Contents

Options for Enabling and Disabling Specify Blocks	1615
Options for Specifying When Simulation Stops	1615
Options for Recording Output	1616
Options for Controlling Messages	1616
Options for VPD Files	1617
Options for VCD Files	1618
Options for Specifying Delays	1619
Options for Flushing Certain Output Text File Buffers	1621
Options for Licensing	1621
Option to Specify User-Defined Runtime Options in a File	1622
Option for the Support of Reading Gzipped Files	1622
Example	1622
Output	1623
Supported APIs	1623
Limitations	1624
Option for Initializing Verilog Variables, Registers and Memories at Runtime	1624
Option for Initializing Verilog Variables, Registers and Memories in Selective Parts of a Design at Runtime	1625
Options for Initializing Verilog Registers, Memories, and Sequential UDPs at Non-Zero Simulation Time	1626
Prerequisites for Deferred Initreg	1626
Use Model of Deferred Initreg	1626
System Task	1627
Command Line	1628
Unified Command-Line Interface (UCLI)	1629
Example of Deferred Initreg	1629
Limitations	1631
General Options	1631
Viewing the Compile Time Options	1631
Recording Where ACC Capabilities are Used	1632
Suppressing the \$stop System Task	1632
Debugging Plusarg-Controlled Testbench Code	1632
Enabling User-defined Plusarg Options	1633
Enabling Overriding the Timing of a SWIFT SmartModel	1633
Enabling feature beyond VHDL LRM	1633
Enabling Loop Detect	1634
Specifying acc_handle_simulated_net PLI Routine	1634

Contents

Loading DPI Libraries Dynamically at Runtime	1635
Loading PLI Libraries Dynamically at Runtime	1635
Independent Seeding Across Multiple Instances	1636
<hr/>	
E. Verilog Compiler Directives and System Tasks	1637
Compiler Directives	1637
Compiler Directives for Cell Definition	1637
Compiler Directives for Setting Defaults	1638
Compiler Directives for Macros	1638
Compiler Directives for Delays	1640
Compiler Directives for Back Annotating SDF Delay Values	1640
Compiler Directives for Source Protection	1641
General Compiler Directives	1641
Compiler Directive for Including a Source File	1641
Compiler Directive for Setting the Time Scale	1641
Compiler Directive for Specifying a Library	1641
Compiler Directive for File Names and Line Numbers	1642
Unimplemented Compiler Directives	1642
System Tasks and Functions	1642
System Tasks for SystemVerilog Assertions Severity	1643
System Tasks for SystemVerilog Assertions Control	1643
System Tasks for SystemVerilog Assertions	1644
System Tasks for VCD Files	1644
System Tasks for LSI Certification VCD and EVCD Files	1646
System Tasks for VPD Files	1647
System Tasks for SystemVerilog Assertions	1652
System Tasks for Executing Operating System Commands	1653
System Tasks for Log Files	1653
System Tasks for Data Type Conversions	1653
System Tasks for Displaying Information	1654
System Tasks for File I/O	1654
System Tasks for Loading Memories	1656
System Tasks for Time Scale	1657
System Tasks for Simulation Control	1657
System Tasks for Timing Checks	1657
Timing Checks for Clock and Control Signals	1658
System Tasks for PLA Modeling	1660
System Tasks for Stochastic Analysis	1660

Contents

System Tasks for Simulation Time	1660
System Tasks for Probabilistic Distribution	1661
System Tasks for Resetting VCS	1661
General System Tasks and Functions	1662
Checks for a Plusarg	1662
SDF Files	1662
Counting the Drivers on a Net	1662
Depositing Values	1662
Fast Processing Stimulus Patterns	1662
Saving and Restarting The Simulation State	1663
Checking for X and Z Values in Conditional Expressions	1663
Calculating Bus Widths	1664
Displaying the Method Stack	1664
IEEE Standard System Tasks Not Yet Implemented	1668
F. PLI Access Routines	1669
Access Routines for Reading and Writing to Memories	1669
acc_setmem_int	1670
acc_getmem_int	1671
acc_clearmem_int	1672
Examples	1672
acc_setmem_hexstr	1675
Examples	1676
acc_getmem_hexstr	1678
acc_setmem_bitstr	1679
acc_getmem_bitstr	1680
acc_handle_mem_by_fullname	1680
acc_readmem	1681
Examples	1682
acc_getmem_range	1683
acc_getmem_size	1683
acc_getmem_word_int	1684
acc_getmem_word_range	1684
Access Routines for Multidimensional Arrays	1685
tf_mdanodeinfo and tf_imdanodeinfo	1686
acc_get_mda_range	1687
acc_get_mda_word_range()	1688
acc_getmda_bitstr()	1689
acc_setmda_bitstr()	1690

Contents

Access Routines for Probabilistic Distribution	1691
vcs_random	1692
vcs_random_const_seed	1692
vcs_random_seed	1693
vcs_dist_uniform	1693
vcs_dist_normal	1694
vcs_dist_exponential	1695
vcs_dist_poisson	1695
Access Routines for Returning a Pointer to a Parameter Value	1696
acc_fetch_paramval_str	1696
Access Routines for Line Callbacks	1697
acc_mod_lcb_add	1697
acc_mod_lcb_del	1698
acc_mod_lcb_enabled	1700
acc_mod_lcb_fetch	1700
acc_mod_lcb_fetch2	1701
acc_mod_sfi_fetch	1702
Access Routines for Source Protection	1703
vcsSpClose	1706
vcsSpEncodeOff	1707
vcsSpEncodeOn	1708
vcsSpEncoding	1709
vcsSpGetFilePtr	1709
vcsSplInitialize	1710
vcsSpOvaDecodeLine	1711
vcsSpOvaDisable	1712
vcsSpOvaEnable	1713
vcsSpSetDisplayMsgFlag	1714
vcsSpSetFilePtr	1714
vcsSpSetLibLicenseCode	1715
vcsSpSetPliProtectionFlag	1716
vcsSpWriteChar	1716
vcsSpWriteString	1717
Access Routine for Signal in a Generate Block	1718
acc_object_of_type	1718
VCS API Routines	1719
Vcsinit()	1719

Contents

VcsSimUntil()	1719
---------------	------

Preface

The Verification Methodology Manual (VMM) describes the framework for developing re-usable verification components, subenvironments and environments. The preface discusses the following:

- [Customer Support](#)
- [Synopsys Statement on Inclusivity and Diversity](#)

Customer Support

For any online access to the self-help resources, you can refer to the documentation and searchable knowledge base available in SolvNetPlus.

To obtain support for your VCS product, choose one of the following:

- Open a Case through SolvNetPlus.

Go to and provide the requested information, including:

- **Product - L1 as VCS**
- **Case Type**

Fill in the remaining fields according to your environment and issue.

- Send an e-mail message to vcs_support@synopsys.com

Include product name (L1), sub-product name/technology (L2), and product version in your e-mail, so it can be routed correctly.

Your e-mail will be acknowledged by automatic reply and assigned a Case number along with Case reference ID in the subject (**ref: ...:ref**).

For any further communication on this Case via e-mail, **send an e-mail to vcs_support@synopsys.com and ensure to have the same Case ref ID in the subject header** or else it will open duplicate Cases.

Note:

In general, we need to be able to reproduce the problem in order to fix it, so a simple model demonstrating the error is the most effective way for us to identify the bug. If that is not possible, then provide a detailed explanation of the problem along with complete error and corresponding code, if any/ permissible.

Synopsys Statement on Inclusivity and Diversity

Synopsys is committed to creating an inclusive environment where every employee, customer, and partner feels welcomed. We are reviewing and removing exclusionary language from our products and supporting customer-facing collateral. Our effort also includes internal initiatives to remove biased language from our engineering and working environment, including terms that are embedded in our software and IPs. At the same time, we are working to ensure that our web content and software applications are usable to people of varying abilities. You may still find examples of non-inclusive language in our software or documentation as our IPs implement industry-standard specifications that are currently under review to remove exclusionary language.

1

Getting Started

VCS® is a high-performance, high-capacity simulator that incorporates advanced, high-level abstraction verification technologies into a single open native platform.

VCS is a compiled code simulator. It enables you to analyze, compile, and simulate Verilog, VHDL, mixed-HDL, SystemVerilog, OpenVera and SystemC design descriptions. It also provides you with a set of simulation and debugging features to validate your design. These features provide capabilities for source-level debugging and simulation result viewing.

VCS accelerates complete system verification by delivering the fastest and highest capacity Verilog, VHDL, and mixed HDL simulation for RTL functional verification. The seamless support for mixed-language simulation of VCS provides a high performance solution to your IP integration problems and gate-level simulation. VCS supports VHDL External Names feature introduced in the *1076-2008-IEEE Standard VHDL Language Reference Manual*.

This chapter includes the following sections:

- [Simulator Support with Technologies](#)
- [Simulation Preemption Support](#)
- [Setting Up the Simulator](#)
- [Using the Simulator](#)
- [Default Time Unit and Time Precision](#)
- [Searching Identifiers in the Design Using UNIX Commands](#)

Simulator Support with Technologies

VCS supports the following IEEE standards:

- The Verilog language as defined in the *Standard Verilog Hardware Description Language* (IEEE Std 1364).
- The VHDL Language as defined in the *Standard VHDL Hardware Description Language* (IEEE VHDL 1076-1993).
- The SystemVerilog language (with some exceptions) as defined in the *IEEE Standard for SystemVerilog -- Unified Hardware Design, Specification, and Verification Language* (IEEE Std 1800™ - 2012)

In addition to its standard Verilog, VHDL, and mixed HDL and SystemVerilog compilation and simulation capabilities, VCS includes the following integrated set of features and tools:

- SystemC - VCS / SystemC Co-simulation Interface enables VCS and the SystemC modeling environment to work together when simulating a system described in the Verilog, VHDL, and SystemC languages. For more information, refer to “Using SystemC” .
- Verdi — For more information, refer to “*Using Verdi*”.
- Unified Command-line Interface (UCLI) — For more information, refer to “*Using UCLI*” .
- Built-In Coverage Metrics — a comprehensive built-in coverage analysis functionality that includes condition, toggle, line, finite-state-machine (FSM), path, and branch coverage. You can use coverage metrics to determine the quality of coverage of your verification test and focus on creating additional test cases. You only need to compile once to run both simulation and coverage analysis. For more information, refer to “*Coverage*” .
- DirectC Interface — this interface allows you to directly embed user-created C/C++ functions within your Verilog design description. This results in a significant improvement in ease-of-use and performance over existing PLI-based methods. VCS atomically recognizes C/C++ function calls and integrates them for simulation, thus eliminating the need to manually create PLI files.

VCS supports Synopsys DesignWare IPs, VCS Verification Library, VMC models, Vera, CustomSim, CustomSimHSIM and CustomSim FineSim. For information on integrating VCS with CustomSim, refer to the *Discovery AMS: Mixed-Signal Simulation User Guide*. For more information about CutomSim FineSim, see the FineSim User Guide: Pro and SPICE Reference.

VCS can also be integrated with third-party tools such as Specman, Denali, and other acceleration and emulation systems.

Simulation Preemption Support

VCS supports simulation preemption. If you suspend a VCS simulation, VCS waits for the safe memory point to suspend the job and checks in the license. When VCS simulation is resumed at a later time, it checks out the license and continues the simulation from the point where it was suspended. You can use `ctrl+z` or `kill -TSTP <pid>` to preempt simulation in VCS.

Setting Up the Simulator

This section outlines the basic steps for preparing to run VCS. It includes the following topics:

- [Verifying Your System Configuration](#)
 - [Obtaining a License](#)
 - [Setting Up Your Environment](#)
 - [Creating a synopsys_sim.setup File](#)
 - [Displaying Setup Information](#)
 - [Displaying Design Information Analyzed Into a Library](#)
-

Verifying Your System Configuration

You can use the `syschk.sh` script to check if your system and environment match the QSC requirements for a given release of a Synopsys product. The QSC (Qualified System Configurations) represents all system configurations maintained internally and tested by Synopsys.

To check whether the system you are on meets the QSC requirements, enter:

```
% syschk.sh
```

When you encounter any issue, run the script with tracing enabled to capture the output and contact Synopsys. To enable tracing, you can either uncomment the `set -x` line in the `syschk.sh` file or enter the following command:

```
% sh -x syschk.sh >& syschk.log
```

Use `syschk.sh -v` to generate a more verbose output stream including the exact path for various binaries used by the script, etc. For example:

```
% syschk.sh -v
```

Note:

If you copy the `syschk.sh` script to another location before using it, you must also copy the `syschk.dat` data file to the same directory.

You can also refer to the “Supported Platforms and Products” section of the VCS Release Notes for the list of supported platforms and recommended C compiler and linker versions.

Obtaining a License

You must have a license to run VCS. To obtain a license, contact your local Synopsys Sales Representative. Your Sales Representative will need the hostid for your machine.

To start a new license, do the following:

- ▶ Verify that your license file is functioning correctly:

```
% lmcksum -c license_file_pathname
```

Running this licensing utility ensures that the license file is not corrupt. You should see an “OK” for every INCREMENT statement in the license file.

Note:

The `snpslmd` platform binaries and accompanying FlexLM utilities are shipped separately and are not included with this distribution. You can download these binaries as part of the Synopsys Common Licensing (SCL) kit from the Synopsys Web Site at:

<http://www.synopsys.com/cgi-bin/ASP/sk/smartkeys.cgi>

1. Start the license server:

```
% lmgrd -c license_file_pathname -l logfile_pathname
```

2. Set the `LM_LICENSE_FILE` or `SNPSLMD_LICENSE_FILE` environment variable to point to the license file. For example:

```
% setenv LM_LICENSE_FILE /u/edatools/vcs/license.dat
```

or

```
% setenv SNPSLMD_LICENSE_FILE /u/edatools/vcs/  
license.dat
```

You can use `SNPSLMD_LICENSE_FILE` environment variable to set licenses explicitly for Synopsys tools.

If you set the `SNPSLMD_LICENSE_FILE` environment variable, then VCS ignores the `LM_LICENSE_FILE` environment variable.

Note:

A single VCS license (under Synopsys' Common Licensing Program) enables you to run Verilog-only, VHDL-only, or mixed-HDL simulations.

Setting Up Your Environment

To run VCS, you need to set the following environment variables:

- `$VCS_HOME` environment variable

Set the environment variable `VCS_HOME` to the path where VCS is installed as shown below:

```
% setenv VCS_HOME installation_path
```

- `$PATH` environment variable

Set your UNIX PATH variable to `$VCS_HOME/bin` as shown below:

```
% set path = ($VCS_HOME/bin $path)
```

OR

```
% setenv PATH $VCS_HOME/bin:$PATH
```

- `LM_LICENSE_FILE` or `SNPSLMD_LICENSE_FILE` environment variable:

Set the license variable `LM_LICENSE_FILE` or `SNPSLMD_LICENSE_FILE` to your license file as shown below:

```
% setenv LM_LICENSE_FILE Location_to_the_license_file
```

OR

```
% setenv SNPSLMD_LICENSE_FILE /u/edatools/vcs/  
license.dat
```

You can use `SNPSLMD_LICENSE_FILE` environment variable to set licenses explicitly for Synopsys tools.

If you set the `SNPSLMD_LICENSE_FILE` environment variable, then VCS ignores the `LM_LICENSE_FILE` environment variable.

For additional information on environment variables, see “VCS Environment Variables” .

Creating a `synopsys_sim.setup` File

VCS uses the `synopsys_sim.setup` file to configure its environment for VHDL and mixed-HDL designs. This file maps the VHDL design library names to specific host directories, sets search paths, and assigns values to simulation control variables.

When you invoke VCS, it looks for the `synopsys_sim.setup` files in the following three directories with the same order:

- Master setup directory

The `synopsys_sim.setup` file in the `$VCS_HOME/bin` directory contains default settings for your entire installation. VCS reads this file first.

- Your home directory

VCS reads the setup file in your home directory second, if present. The settings in this file take precedence over the conflicting settings in your `synopsys_sim.setup` file in the master setup directory, and carry over the rest.

- Your run directory

VCS reads the setup file in your design directory last. The settings in this file take precedence over the conflicting settings in your `synopsys_sim.setup` file in the master setup directory, and the `synopsys_sim.setup` file in your home directory, and will carry over the rest. You can use this file to customize the environment for a particular design.

Note:

This is the directory you invoke and run VCS from; it is not the directory where you store or generate your design files.

The key components of the setup file are the name mappings in the design libraries and the variable assignments. See the following sections for additional information.

The following rules pertain to setup files:

- Blank lines are ignored.
- Physical directory names are case-sensitive.
- All commented lines begin with two dashes (--) .
- The backslash character (\) is used for line continuation.

The following is a sample `synopsys_sim.setup` file:

```
--VCS setup file for ASIC
--Mapping default work directory
```

```
WORK > DEFAULT
DEFAULT : ./work
```

```
--Library Mapping

STATS_PKG : ./stat_work
MEM_PKG : ./mem_work

--Simulation variables

TIMEBASE = ps

TIME_RESOLUTION = 1NS
```

The Concept of a Library in VCS

When you analyze a design, VCS stores the intermediate files in a design library, also called as a logical library. This logical library is pointed to a physical library, which is a physical directory in your UNIX file system. *You specify this mapping in the synopsys_sim.setup file as shown in the following lines:*

```
WORK > DEFAULT
DEFAULT : ./worklib
```

In the above example, `WORK` is the default logical library and is mapped to the physical library `worklib`. With the above setting, by default VCS stores all the intermediate files in the library `work`, and it errors out if the library `work` does not exist in the specified path.

Library Name Mapping

For flexibility in library naming, VCS allows you to create multiple logical libraries each one pointing to a different physical library. The syntax to map a logical library to a physical library is shown below:

```
logical_name : physical_name
```

Note:

Logical library names are case insensitive.

The following examples show two logical libraries `ALU8` and `ALU16` mapped to `alu_8bit` and `alu_16bit` physical libraries. During analysis, you can use the `-work` option to analyze the files into the respective libraries.

```
ALU8 : ./alu_8bit
ALU16 : ./alu_16bit
```

The VCS built-in standard libraries have the following default name mappings:

```
IEEE : $VCS_HOME/$ARCH/packages/IEEE/lib
SYNOPSYS : $VCS_HOME/$ARCH/packages/synopsys/lib
```

In these default mappings, `$ARCH` is any one of the following - sparcOS5, sparc64, linux, amd64, rs6000, hp32, suse32, or suse64.

Use these built-in libraries in your design, whenever possible, to get maximum performance from VCS.

Including Other Setup Files

To include any other setup files, specify the following in the `synopsys_sim.setup` file:

```
OTHERS = [filename]
```

Note that you cannot override the environment settings using this file. In addition, files included in this manner can be nested up to 8 levels.

If VCS is unable to open the specified file, it exits with the following error message:

```
Error: analysis preParsing vhdl-314
      snps_setup fatal error: (Severity SNPS SETUP USER
      FATAL) Cannot open included setup file "user_setup.file"
```

Using the `SYNOPSYS_SIM_SETUP` Environment Variable

You can also specify a setup file to define VCS setup variables. To do this, set the `SYNOPSYS_SIM_SETUP` variable to your setup file as shown below:

```
% setenv SYNOPSYS_SIM_SETUP my_setup
```

Note that you can use any name for this setup file; you do not need to use `synopsys_sim.setup`.

The settings in this file take precedence over conflicting settings in any regular setup file in the current directory, home directory, or installation directory, and is also searched during simulation. If the file you specify in the `SYNOPSYS_SIM_SETUP` variable cannot be opened, VCS issues the following message:

```
Warning: analysis preParsing vhdl-315
      snps_setup message: (Severity SNPS SETUP USER WARNING)
      Cannot open setup file "synopsys_sim.setup"
```

Supporting VHDL Non-Locally Static Aggregates

VCS supports non-static aggregates with two choices and with one `others` choice. This supported scenarios must satisfy the following conditions:

- Array aggregate with one non-locally static choice plus `others`. However, if the range of the aggregate is not locally static, the choice can be locally static or non-locally static.
- Choice is a single element.

- The element type is scalar.
- For multi-dimensional array aggregates and record aggregates, only the non-locally-static aggregate must be at the bottom.

Use Model

To enable this feature, use the `COMPLEX_AGGREGATES = TRUE` variable in the `synopsys_sim.setup` file.

Usage Example

Consider the following `test.vhd` file. The new supported value is highlighted in the example.

Example 1 test.vhd file

```
entity top is
  generic (
    BITS: natural := 8
  );
end entity top;

architecture arch of top is
  constant VAL0 : bit_vector(BITS-1 downto 0) := (BITS-1 => '1', others => '0');
  constant VAL1 : bit_vector(8-1 downto 0) := (8-1 => '1', others => '0');
begin
  assert VAL0=VAL1 report "wrong aggregate" severity error;
end architecture arch;
```

Run the example using the following commands:

- `vhdlan test.vhd`
- `vcs top`
- `./simv`

Displaying Setup Information

To list and display all current setup information in your `synopsys_sim.setup` file, enter the following command at the prompt:

```
% show_setup
```

The full syntax of the `show_setup` command is as follows:

```
% show_setup [-v] [-lib]
```

The `show_setup` command options are:

`-v`

Displays the version number and exits.

`-lib`

Displays the library mapping.

The `show_setup` command lists setup information in alphabetical order.

The following example uses `show_setup` to check if optimizations are on for event simulation:

```
% show_setup | grep OPTIMIZE
```

The result of this command is:

```
OPTIMIZE = FALSE
```

Note:

The `show_setup` command shows the cumulative effect of reading each of the three possible `synopsys_sim.setup` files.

Displaying Design Information Analyzed Into a Library

The `lolib` executable displays the following information:

- Entity name, module name, architecture name, configuration name, location of the source file, VCS version, and the timestamp information as and when the file is analyzed.
- All design unit names analyzed in the specified library.
- Architecture name of each entity and package body name of each package.

By default, `lolib` lists all design units analyzed into the default logical library.

The syntax of `lolib` is as follows:

```
% lolib [-l] [-r] [-lib path] design_unit_name
```

The `lolib` command options are:

`-l`

Displays entity name, architecture name, configuration name, location of the source file, VCS version and the timestamp for when the design file was analyzed.

`-r`

Displays architecture name of each entity, and package body name of each package.

`-lib path`

Displays the list of design units, package name, and the configuration name in the specified logical library.

`design_unit_name`

`design_unit_name` can be a module, entity, architecture, package body, or a configuration.

Example

```
% llib -l ZERO

Library: worklibs
ENTITY      ZERO
Source file   : /u/snps/vhdl/zero.vhd
VCS Version  : Y-2006.06-SP1-5
Timestamp     : Mon Aug 13 22:31:34 2007
Library (four state only): worklibs
```

As illustrated in the example, the design unit ZERO is analyzed into the `worklibs` logical library. The `llib` executable also provides the location of the source file, VCS version used to analyze the design unit, and the timestamp information.

Using the Simulator

You can use the following flows to simulate a design using VCS:

- [Two-step Flow](#)
- [Three-step Flow](#)
- [Basic Usage Model](#)

Two-step Flow

The two-step flow is supported only for Verilog HDL and SystemVerilog designs. Simulating a design using two-step flow involves the following two basic steps:

- [Compiling the Design](#)
- [Simulating the Design](#)

Compiling the Design

Compiling is the first step to simulate your design. In this phase, VCS builds the instance hierarchy and generates a binary executable `simv`. This binary executable is later used for simulation. For more information, see “Two-step Flow” .

Simulating the Design

During compilation, VCS generates a binary executable, `simv`. You can use `simv` to run the simulation. For more information, see “Two-step Flow” .

Three-step Flow

The three-step flow is supported for Verilog, VHDL, and mixed HDL designs. VCS uses the following three basic steps to compile, elaborate and simulate any Verilog, VHDL, and mixed HDL designs:

- [Analyzing the Design](#)
- [Elaborating the Design](#)
- [Simulating the Design](#)

Analyzing the Design

VCS provides you with the `vhdlan` and `vlogan` executables to analyze your VHDL and Verilog design code. `vhdlan/vlogan` analyzes your design and stores the intermediate files in the design or a work library.

By default, the `vhdlan` option is VHDL-93 compliant, and `vlogan` is Verilog-2000 compliant. You can switch to VHDL-87 using the `-vhdl87` option with `vhdlan`. Similarly, you can switch to VHDL 2002 or VHDL 2008 by using the `-vhdl02` or `-vhdl08` option respectively. For more information, see “Three-step Flow” . You can switch to the SystemVerilog mode by using the option `-sverilog` with `vlogan`.

Elaborating the Design

VCS provides you with the `vcs` executable to compile and elaborate the design. This executable compiles/elaborates your design using the intermediate files in the design or work library, generates the object code, and statically links them to generate a binary simulation executable, `simv`. For more information, see “Three-step Flow” .

Simulating the Design

Simulate your design by executing the binary simulation executable, `simv`. For more information, see “Three-step Flow” .

Basic Usage Model

The following sections describe the basic use model for two-step flow and three-step flow:

- [Two-step Flow](#)
- [Three-step Flow](#)

Two-step Flow

Compilation

```
% vcs [compile_options] Verilog_files
```

Simulation

```
% simv [run_options]
```

Three-step Flow

Analysis

Always analyze Verilog before VHDL.

```
% vlogan [vlogan_options] file1.v file2.v
% vhdlan [vhdlan_options] file3.vhd file4.vhd
```

Note:

Specify the VHDL bottommost entity first, then move up in order.

Elaboration

```
% vcs [elaboration_options] design_unit
```

The `design_unit` can be one of the following:

`module`

Verilog top module name.

`entity`

VHDL top entity name.

`entity_archname`

Name of the top entity and architecture to be simulated. By default, `archname` is the most recently analyzed architecture.

`cfgname`

Name of the top-level event configuration to be simulated.

Simulation

```
% simv [run_options]
```

Default Time Unit and Time Precision

The default time unit for Verilog and SystemVerilog simulation is 1 s.

The default time precision for Verilog and SystemVerilog simulation is 1 s.

For VHDL simulation there is no concept of a default time unit and delay values, for example, must have a unit name or unit of measurement, for example:

```
wait for 10.123123 ns;
```

The default time precision for an entire VHDL design is specified with the `TIME_RESOLUTION 1 ns` entry in the `synopsys_sim.setup` file in the VCS installation (see [Creating a synopsys_sim.setup File](#)).

The default time precision for the VHDL part of a mixed HDL design is the smallest or finest of these two:

- What is specified with the `TIME_RESOLUTION` entry in the `synopsys_sim.setup` file.
- The smallest time precision from the Verilog or SystemVerilog part of the design.

You can override the default time precision with the `-time_res` elaboration option.

Note:

The `-time_res` option has no effect on the Verilog code.

Searching Identifiers in the Design Using UNIX Commands

You can use the following `vcsfind` UNIX command to search for identifiers in your design. The `vcsfind` script is located in `$VCS_HOME/bin`. You must specify the location of the `fsearch.db` file.

```
vcsfind [<options> --] [<identifier>] [(+/-)<search group>]+
```

Where,

options

Search options in the following table. These options must be separated by a “--” from the search query. Any change to the Verdi GUI settings has no effect on the `vcsfind` command.

Table 1 Supported Search Options

Search Option	Description
--version	Displays program's version number and exits
-h, --help	Displays help message and exits
-b, --bw(Black and White)	Highlights with bold and underline only, no colors.
-d N, --dir_levels=N	Prints n directory levels for every matching line. Default is 0.
-f DB-FILE, --file=DB-FILE	Specifies the database file. Default is vcsfind.db
-H, --gui-help	Prints help for GUI use.
-l N, --limit=N	Limits search to the first n matches. 0 means no limit. Default is 1000.
-m, --match_only	Matches the query pattern only. Does not display scope information.
-o OUTPUT-FILE, --output=OUTPUT-FILE	Outputs into a file. Default is stdout/stderr. This option bundles stdout and stderr, so -o - will redirect errors to stdout.
-p, --plain	Does not highlight matches in bold.
-r, --regexp	Regular expression search pattern. The pattern is interpreted as ^<pattern>\$, so .* may be desired at the beginning and end of the pattern.
-t, --translate	Translation mode. Prints only the translation of the query pattern into the internal SQL query string.
-u, --uclimode	Enables UCLI mode. This option is used for interaction with UCLI.
-v, --verbose	Enables verbose mode.

identifier

Identifier string to be searched.

search group

The name of the group to be included to search or excluded from search. The following search groups are supported:

Packages, Modules, Ports, Parameters, Vars, Functions, Assertions, Types, Members, Instances

You can also use Verdi to search for the identifiers in your design.

Examples

```
% vcsfind -f simv.daidir/debug_dump/fsearch/fsearch.db -- Top
```

Following is the sample output:

```
Matching modules:  
top.v:11 module Top  
    scope: Top
```

```
Matching instances:  
top.v:11 inst Top of module Top  
    scope: Top
```

```
Total: 4 results found in 0.053 seconds
```

UTF-8 Unicode File Format

VCS supports UTF-8 Unicode file format during compilation. The UTF-8 Unicode (with BOM) file format is not supported and results in an error message.

2

VCS Flow

This chapter describes the simulation flows supported by VCS. You can use the following flows to simulate your design using VCS:

- [Three-step Flow](#)
 - [Two-step Flow](#)
 - [Partition Compile Flow](#)
-

Three-step Flow

Simulating a design using three-step flow involves three basic steps:

- [Analysis](#)
- [Elaboration](#)
- [Simulation](#)

VCS uses these three steps to compile any design irrespective of the HDL, HVL, and other supported technologies used. For information on supported technologies, see [Simulator Support with Technologies](#).

For information on the data types supported across the language boundary for both ports and generics or parameters, see the [VCS Simulation Coding and Modeling Style Guide](#).

For information on licensing options, see [Options for Licensing](#).

Analysis

Analysis is the first step to simulate your design. In this phase, you analyze your VHDL, Verilog, SystemVerilog, and OpenVera files using `vhdlan` or `vlogan` accordingly. The following section includes a few example command lines to analyze your design files:

Analyzing your VHDL files:

```
% vhdlan [vhdlan_options] file1.vhd file2.vhd
```

Analyzing your Verilog files:

```
% vlogan [vlogan_options] file1.v file2.v
```

Analyzing your SystemVerilog files:

```
% vlogan -sverilog [vlogan_options] file1.sv file2.sv file3.sv
```

For the complete usage model, see [Using SystemVerilog](#).

Analyzing your OpenVera files:

```
% vlogan -ntb [vlogan_options] file1.vr file2.vr file3.v
```

For the complete use model, see [Using OpenVera Native Testbench](#).

Analyzing your SystemVerilog and OpenVera files:

```
% vlogan -sverilog -ntb [vlogan_options] file1.sv file2.vr file3.v
```

Note that you can analyze SystemVerilog files or OpenVera files along with other Verilog files in the same `vlogan` command line as shown in the examples. Unless it is required, you do not need to separately analyze these files.

In the analysis phase, VCS checks the design for the syntax errors. In this phase, VCS generates the intermediate files required for elaboration and saves these files in the design or work library pointed to by your default logical library. For information on library mapping, see “The Concept of a Library in VCS”. You can instruct VCS to save these intermediate files in a different library using the `-work` option with the `vhdlan` or `vlogan` executables.

Before you analyze your design using `vhdlan` or `vlogan`, ensure that the library mappings are defined in the `synopsys_sim.setup` file, and that the specified physical library for the logical library exists. If the physical directory does not exist, VCS exits with an error message.

VCS provides `vhdlan` and `vlogan` executable to analyze VHDL and Verilog design files, respectively. The following sections describe the usage of these two executables and some of the commonly used options.

Using vhdlan

The `vhdlan` executable analyzes your VHDL design files and stores the generated intermediate files in the design or work library. The syntax for the `vhdlan` executable is as follows:

```
% vhdlan [vhdlan_options] VHDL_filename_list
```

Commonly Used Analysis Options

This section lists some of the commonly used `vhdlan` options. For a complete list of options, see section “The `vhdlan` Utility” .

Command Options

`-help`

Displays usage information for `vhdlan`.

`-nc`

Suppresses the Synopsys copyright message.

`-q`

Suppresses all `vhdlan` messages.

`-version`

Displays the version number of `vhdlan` and exits without running analysis.

`-full64`

Analyzes the design for 64-bit simulation.

`-work library`

Maps a design library name to the logical library name `WORK` that receives the output of `vhdlan`. Mapping with this command line option overrides any assignment of `WORK` to another library name in the setup file.

`library` can also be a physical path that corresponds to a logical library name defined in the setup file.

`-vhdl87`

Enables to analyze non-portable VHDL code that contains object names that are now VHDL-93 reserved words by default. VCS is VHDL-93 compliant.

`-vhdl02`

Enables to analyze the VHDL 2002 protected type. For more information, see “Support for VHDL 2002, 2008 and 2019” .

`-vhdl08`

Enables to analyze the VHDL 2008 constructs provided in the chapter “Support for VHDL 2002, 2008 and 2019” .

`-output outfile`

Redirects standard output from VCS analysis (that usually goes to the screen) to the file that you specify as *outfile*.

`-x1rm`

Enables VHDL features beyond those described in LRM.

`-f filename`

Specifies a file that contains a list of source files. You should specify the bottom most VHDL entity first, and then move up in order.

`-functional_vital`

Specifies generating code for functional VITAL simulation mode.

`-l filename`

Specifies a log file where VCS records the analyzer messages.

`-no_functional_vital`

Specifies generating code for full-timing VITAL simulation mode.

`VHDL_filename_list`

Specifies the VHDL source file names to be analyzed. If you do not provide an extension, `.vhd` is assumed.

The maximum identifier name length is 250 for package, package body and configuration names. The combined length of an entity name plus architecture name must not exceed 250 characters as well. All other VHDL identifier names and string literals do not have a limitation.

`-init_std_logic`

You can initialize all uninitialized VHDL signals, ports and variables of the data type `STD_LOGIC/STD_ULOGIC` (scalar/vector) with a given 9-value. A VHDL signal or variable of this type can take on the following values – ‘U’, ‘X’, ‘0’, ‘1’, ‘Z’, ‘W’, ‘L’, ‘H’, ‘-’.

You can supply the value at `vhdlan` command line option as shown in the following command line:

`% vhdlan hello.vhd -init_std_logic 0`

You can also initialize the value in the `synopsys_sim_setup` file.

In the `synopsys_sim_setup` file, you can set the value to any one of the nine values to the variable `INIT_STD_LOGIC`. For example, `INIT_STD_LOGIC=0`. To create a `synopsys_sim_setup` file, see “Creating a `synopsys_sim.setup` File” .

Using vlogan

Similar to `vhdlan`, the `vlogan` executable analyzes your Verilog design files and stores the generated intermediate files in the design or work library. The syntax for the `vlogan` executable is as follows:

```
% vlogan [vlogan_options] Verilog_filename_list
```

Commonly Used Analysis Options

This section lists some of the commonly used `vlogan` options. For a complete list of options, see “The `vlogan` Utility”.

Command Options

`-help`

Displays usage information for `vlogan`.

`-nc`

Suppresses the Synopsys copyright message.

`-q`

Suppresses all `vlogan` messages.

`-f filename`

Specifies a file that contains a list of source files.

The maximum line length in the specified file `filename` must be less than 1024 characters. VCS truncates the line exceeding this limit and issues a warning message.

`-full64`

Analyzes the design for 64-bit simulation.

`-ignore keyword_argument`

Suppresses warning messages depending on which keyword argument is specified. The keyword arguments are as follows:

`unique_checks`

Suppresses warning messages about `unique if` and `unique case` statements.

`priority_checks`

Suppresses warning messages about `priority if` and `priority case` statements.

`all`

Suppresses warning messages about unique if, unique case, priority if and priority case statements.

`-l filename`

Specifies a log file where VCS records the analyzer messages.

`-liblist logical_lib1+logical_lib2+...logical_libn`

It specifies the library search order for resolving imported package definitions. The `vlogan -liblist` option restricts the libraries in which `vlogan` should search for resolving package references found while analyzing.

If the `-liblist` option is not included, `vlogan` searches all the logical libraries listed in the `synopsys_sim.setup` file.

When using multiple `vlogan` commands with the same `-work` logical library, run the commands sequentially, and if one command uses `-liblist`, then ensure that all the remaining `vlogan` commands are using the same `-liblist` argument list, as shown in the following command lines:

```
%vlogan a.v -work shared_lib -liblist shared_lib+ovm_lib+common_lib
%vlogan b.v -work shared_lib -liblist shared_lib+ovm_lib+common_lib
%vlogan c.v -work shared_lib -liblist shared_lib+ovm_lib+common_lib
-ntb
```

Enables the use of the OpenVera testbench language constructs described in the *OpenVera Language Reference Manual: Native Testbench*.

`-ntb_define macro`

Specifies any OpenVera macro name on the command line. You can specify multiple macro names using the plus (+) character.

`-ntb_fileext .ext`

Specifies an OpenVera file name extension. You can specify multiple file name extensions using the plus (+) character.

`-ntb_incdir directory_path`

Specifies the include directory path for OpenVera files. You can specify multiple include directories using the plus (+) character.

`-ova_file filename`

Identifies `filename` as an assertion file. It is not required if the file name ends with `.ova`. For multiple assertion files, repeat this option with each file.

`-sverilog`

Enables the analysis of SystemVerilog source code.

`-sv_pragma`

Instructs VCS to compile the SystemVerilog Assertions code that follows the `sv_pragma` keyword in a single line or multi-line comment.

`-timescale=time_unit/time_precision`

This option enables you to specify the timescale for the source files that do not contain `'timescale` compiler directive and precede the source files that contain.

Do not include spaces when specifying the arguments to this option.

`-v library_file`

Specifies a Verilog library file to search for module definitions.

`-y library_directory`

Enables you to specify a Verilog library directory to search for module definitions. VCS looks in the source files in this directory for definitions of the module and UDP instances that VCS found in your source code. However, for which it did not find the corresponding module or UDP definitions in your source code.

VCS looks in this directory for a file with the same name as the module or UDP identifier in the instance (not the instance name). If it finds this file, VCS looks in the file for the module or UDP definition to resolve the instance.

If you have multiple modules with the same name in different libraries, VCS selects the module defined in the library that is specified with the first `-y` option.

For example:

If `rev1/cell.v` and `rev2/cell.v` and `rev3/cell.v` all exist and define the module `cell()`, and you issue the following command:

```
% vlogan -y rev1 -y rev2 -y rev3 +libext+.v top.v
```

VCS selects `cell.v` from `rev1`.

However, if the `top.v` file has a ``uselib` compiler directive as shown below, then ``uselib` takes priority.

```
//top.v
`uselib directory = /proj/libraries/rev3
//rest of top module code
//end top.v
```

In this case, VCS uses `rev3/cell.v` when you issue the following command:

```
% vlogan -y rev1 -y rev2 +libext+.v top.v
```

Include the `+libext` compile time option to specify the file name extension of the files you want VCS to search for in these directories.

`-work library`

Maps a design library name to the logical library name `WORK` that receives the output of `vlogan`. Mapping with the command-line option overrides any assignment of `WORK` to another library name in the setup file.

`+define+macro`

Defines a text macro. Test this definition in your Verilog source code using the `'ifdef` compiler directive.

`+libext+extension+`

Specifies that VCS searches only for files with the specified file name extensions in a library directory. You can specify more than one extension, separating the extensions with the plus (+) character. For example, `+libext+.v+.V+` specifies searching files with either the `.v` or `.V` extension in a library. The order in which you add file name extensions to this option does not specify an order in which VCS searches files in the library with these file name extensions.

`+lint=[no] ID|none|all`

Enables messages that provides information about when your Verilog code contains something that is bad style, but is often used in designs.

`+liborder`

Specifies searching for module definitions for unresolved module instances through the remainder of the library where `vlogan` finds the instance, then searching the subsequent library on the `vlogan` command line before searching in the first library in the command line.

`+librescan`

Always specifies searching libraries for module definitions of unresolved module instances beginning with the first library in the `vlogan` command line.

`+incdir+directory`

Specifies the directories that contain the files specified with the `'include` compiler directive. You can specify more than one directory, separating each path name with the “+” character.

`+nowarnTFMPC`

Suppress the Too few module port connections warning messages during Verilog Compilation.

+systemverilogext+ext

Specifies a file name extension for SystemVerilog source files. If you use a different file name extension for the SystemVerilog part of your source code and you use this option, the `-sverilog` option has to be omitted.

+verilog2001ext+ext

Specifies a file name extension for Verilog 2001 source files.

+verilog1995ext+ext

Specifies a file name extension for Verilog 1995 files. Using this option allows you to write Verilog 1995 code that would be invalid in Verilog 2001 or SystemVerilog code, such as using Verilog 2001 or SystemVerilog keywords, like `localparam` and `logic`, as names.

+warn

Enables or disables warning messages.

Verilog_source_filename

Specifies the name of the Verilog source file.

-incr_vlogan

Enables you to abort a `vlogan` command when there are no changes in the following:

- source files analyzed by that same command in previous parsing iteration.
- imported SV packages referenced at source files analyzed by that command.
- command line user options (`+define+`) or `vlogan` command options.
- shell environments. See [Disabling Environment Variables Checks on Different Machines](#).

Note:

If a change is detected, the `vlogan` command runs from scratch.

For example:

```
% vlogan -incr_vlogan
```

You can configure incremental `vlogan` to ignore certain changes in shell environments using the following commands:

```
% vlogan -incr_vlogan -vts_ignore_env=Env1, Env2,...
```

For information about the `-vts_ignore_env` option, see the “Option to Exclude Environment Variables During Timestamp Checks” section.

Note:

The following options are `vlogan` parse options:

```
-ignore unique_checks|priority_checks|all
-ntb_define macro
-ntb_fileext .ext
-sv_pragma
-v library_file
-y library_directory
-sverilog
-incr_vlogan
+define+macro
+inccdir+[directory]
+libext+extension+
+systemverilogext+ext
+verilog1995ext+ext
+verilog2001ext+ext
+lint=[no] ID|none|all
+nowarnTFMPC
+warn
```

VCS issues a message with appropriate severity if any of these options are not allowed and used at elaboration.

Disabling Environment Variables Checks on Different Machines

You can use the `-vts_ignore_vars` option to disable checking of the following environment values that help not to reparse the design again.

- Environment variables that are controlled by `bsub` and all the variables starting with `LSB_`: `HOME`, `LS_JOBPID`, `LSB_ACCT_MAP`, `LSB_EXIT_PRE_ABORT`, `LSB_EXIT_QUEUE`, `LSB_EVENT_ATTRIB`, `LSB_HOSTS`, `LSB_INTERACTIVE`, `LSB_INTERACTIVE_SSH`, `LSB_INTERACTIVE_TTY`, `LSB_JOBFILENAME`, `LSB_JOBGROUP`, `LSB_JOBID`, `LSB_JOBNAME`, `LSB_JOB_STARTER`, `LSB_QUEUE`, `LSB_RESTART`, `LSB_TRAPSIGS`, `LSB_XJOB_SSH`, `LSF_VERSION`, `PWD`, `USER`, `VIRTUAL_HOSTNAME`.
- Environment variables related to non-interactive jobs: `TERM`, `TERMCAP`.
- Windows-specific environment variables: `COMPUTERNAME`, `COMSPEC`, `NTRESKIT`, `OS2LIBPATH`, `PROCESSOR_ARCHITECTURE`, `PROCESSOR_IDENTIFIER`, `PROCESSOR_LEVEL`, `PROCESSOR_REVISION`, `SYSTEMDRIVE`, `SYSTEMROOT`, `TEMP`, `TMP`.
- The following environment variables do not take effect on the execution hosts: `LSB_DEFAULTPROJECT`, `LSB_DEFAULT_JOBGROUP`, `LSB_TSJOB_ENVNAME`,

```
LSB_TSJOB_PASSWD, LSF_DISPLAY_ALL_TSC, LSF_JOB_SECURITY_LABEL,
LSB_DEFAULT_USERGROUP, LSB_DEFAULT_RESREQ, LSB_DEFAULTQUEUE,
BSUB_CHK_RESREQ, LSB_UNIXGROUP, LSB_JOB_CWD.
```

For other machine environment change, the design is re-analyzed.

Analyzing the Design to Different Libraries

You can analyze your design to different libraries using the `-work` option with either the `vhdlan` or `vlogan` executable. However, to use this feature, map the required logical libraries to physical libraries. For information about mapping libraries, see, “Library Name Mapping” .

With the `-work` option, you can specify either the logical library name or the physical library name, specified in your `synopsys_sim.setup` file as follows:

```
% vhdlan -work libname1 VHDL_filename_list
% vlogan -work libname1 Verilog_filename_list
```

These command lines analyze your VHDL files and Verilog files, and saves the intermediate files in the `libname1` library. VCS resolves all VHDL files having:

```
library libname1;
use libname1.all;
```

Elaboration

Elaborating is the second step to simulate your design. In this phase, using the intermediate files generated during analysis, VCS builds the instance hierarchy and generates a binary executable `simv`. This binary executable is later used for simulation.

In this phase, you can choose to elaborate the design either in the optimized mode or in the debug mode. The runtime performance of VCS is based on the mode you choose and the level of flexibility required during simulation. Synopsys recommends you to use full debug or partial debug mode until the design correctness is achieved, and then switch to optimized mode.

In optimized mode, also called as batch mode, VCS delivers the best compile time and runtime performance for a design. You typically choose optimized mode to run regressions or when you do not require extensive debug capabilities. For more information, see “Optimizing Simulation Performance for Desired Debug Visibility With the `-debug_access` Option” .

You compile the design in debug mode, also called interactive mode, when you are in the initial phase of your development cycle, or when you need more debug capabilities or tools to debug the design issues. In this mode, the performance is not the best that VCS can deliver. However, using some of the compile time options, you can compile your design in

full debug or partial debug mode to get maximum performance in debug mode. For more information, see “Compiling/Elaborating the Design in the Debug Mode”.

Using VCS

The syntax to use `vcs` is as follows:

```
% vcs [elab_options] [libname.]design_unit
```

Where,

`libname`

The library name where you analyzed your top module, entity, or the configuration. If not specified, VCS looks for the specified `design_unit` in the list of libraries specified in the `synopsys_sim.setup` file as per the order specified. For more information, see [Creating a synopsys_sim.setup File](#).

Here, the `design_unit` can be one of the following:

`module`

Verilog top module name.

`entity`

VHDL top entity name.

`entity_archname`

Name of the top entity and architecture to be simulated. By default, `archname` is the most recently analyzed architecture.

`cfgname`

Name of the top-level configuration.

Commonly Used Options

This section lists some of the commonly used `vcs` options. For a complete list of options, see [Compilation/Elaboration Options](#).

Options for Help

`-h` or `-help`

Lists descriptions of the most commonly used VCS compile time and runtime options.

`-ID`

Returns useful information, such as VCS version and build date, VCS compiler version (same as VCS), and your work station name, platform, and host ID (used in licensing).

Options for 64-bit Elaboration

`-full64`

Enables elaboration and simulation in 64-bit mode.

Alternatively, you can use the `VCS_TARGET_ARCH` environment variable to enable elaboration and simulation in 64-bit mode architecture. For more information, see `VCS_TARGET_ARCH`.

Option to Specify Elaboration Options in a File

`-file filename`

Specifies a file containing elaboration options.

Options for Starting Simulation After Elaboration

`-R`

Runs the executable file immediately after VCS links it together.

Options for Changing Generics and Parameter Values

`-gfile cmdfile`

Overrides the default values for design generics or parameters using values from the file `cmdfile`. The `cmdfile` file is an include file that contains `assign` commands targeting design generics.

Options for Controlling Messages

`-notice`

Enables verbose diagnostic messages.

`-q`

Quiet mode; suppresses messages, such as those about the C compiler VCS is using, the source files VCS is parsing, the top-level modules, or the specified timescale.

`-V`

Verbose mode; compiles verbosely. The compiler driver program prints the commands it executes as it runs the C compiler, assembler, and linker.

Specifying a Log File

`-l filename`

Specifies a file where VCS records elaboration messages. If you also enter the `-R` option, VCS records messages from both elaboration and simulation in the same file.

Simulation

During elaboration, using the intermediate files generated, VCS creates a binary executable, `simv`. You can use `simv` to run the simulation. Based on how you elaborate the design, you can run the simulation using the following two modes:

- [Interactive Mode](#)
- [Batch Mode](#)
- [Commonly Used Runtime Options](#)

For information about elaborating the design, see [Elaboration](#).

Interactive Mode

You can elaborate your design in the interactive mode, also called debug mode, in the initial phase of your design cycle. In this phase, you require abilities to debug the design issues using a GUI or through the command line. To debug using a GUI, you can use Verdi, and to debug through the command-line interface, you can use Unified Command-line Interface (UCLI).

Note:

To simulate a design in the interactive mode, elaborate the design using `-debug_access+r`, `-debug_access+all` or refer to section [Optimizing Simulation Performance for Desired Debug Visibility With the -debug_access Option](#) for more debug compile time options. For information on elaborating the design, see [Elaboration](#)

Batch Mode

You can elaborate your design in batch mode, also called as optimized mode, when most of your design issues are resolved. In this phase, you can achieve better performance to run regressions with minimum debug abilities.

Note:

The runtime performance reduces if you use `-debug_access`. Use this option only when you require runtime debug abilities.

The following command line simulates the design in batch mode:

```
% simv
```

Commonly Used Runtime Options

Use the following command line to simulate the design:

```
% simv_executable [runtime_options]
```

By default, VCS generates the binary executable `simv`. However, you can use the compile time option, `-o` with the `vcs` command line to generate the binary executable with the specified name.

`-gui`

This option starts Verdi when `VERDI_HOME` is set.

`-ucli`

This option starts `simv` in UCLI mode.

For information on Verdi, see [Using Verdi](#).

For a complete list of runtime options, see [Simulation Options](#).

Two-step Flow

The two-step flow is supported only for Verilog HDL and SystemVerilog designs.

Simulating a design using two-step flow involves two basic steps:

- [Compilation](#)
- [Simulation](#)

Compilation

Compiling is the first step to simulate your design. In this phase, VCS builds the instance hierarchy and generates a binary executable `simv`. This binary executable is later used for simulation.

In this phase, you can choose to compile the design either in optimized mode or in debug mode. Runtime performance of VCS is based on the mode you choose and the level of flexibility required during simulation. Synopsys recommends to use full debug or partial debug mode until the design correctness is achieved and then switch to optimized mode.

In the optimized mode, also called as batch mode, VCS delivers the best compile time and runtime performance for a design. You typically choose optimized mode to run regressions, or when you do not require extensive debug capabilities. For more information, see “Optimizing Simulation Performance for Desired Debug Visibility With the `-debug_access` Option”.

You can compile the design in debug mode, also called interactive mode, when you are in the initial phase of your development cycle, or when you need more debug capabilities or tools to debug the design issues. In this mode, the performance is not the best that VCS can deliver. However, using some of the compile time options, you can compile your

design in full debug or partial debug mode to get maximum performance in debug mode. For more information, see “Compiling/Elaborating the Design in the Debug Mode”.

For information on licensing options, refer to “Options for Licensing”.

Using vcs

The syntax to use VCS is as follows:

```
% vcs [compile options] Verilog_files
```

Commonly Used Options

This section lists some of the commonly used `vcs` options.

Options for Help

`-h` or `-help`

Lists descriptions of the most commonly used VCS compile and runtime options.

`-ID`

Returns useful information, such as VCS version and build date, VCS compiler version (same as VCS), and your work station name, platform, and host ID (used in licensing).

Options for Accessing Verilog Libraries

`-v filename`

Enables you to specify a Verilog library file. VCS looks in this file for definitions of the module and UDP instances that VCS found in your source code, however, for which it did not find the corresponding module or UDP definitions in your source code.

`-y directory`

Enables you to specify a Verilog library directory. VCS searches in the source files in this directory for definitions of the module and UDP instances that VCS found in your source code but for which it did not find the corresponding module or UDP definitions in your source code. VCS searches in this directory for a file with the same name as the module or UDP identifier in the instance (not the instance name). If it finds this file, VCS searches in the file for the module or UDP definition to resolve the instance.

If you have multiple modules with the same name in different libraries, VCS selects the module defined in the library that is specified with the first `-y` option.

For example:

If `rev1/cell.v` and `rev2/cell.v` and `rev3/cell.v` all exist and define the module `cell()`, and you issue the following command:

```
% vcs -y rev1 -y rev2 -y rev3 +libext+.v top.v
```

VCS selects `cell.v` from `rev1`.

However, if the `top.v` file has a ``uselib` compiler directive as follows:

```
//top.v
`uselib directory = /proj/libraries/rev3
//rest of top module code
//end top.v
```

then, ``uselib` takes priority. In this case, VCS uses `rev3/cell.v` when you issue the following command:

```
% vcs -y rev1 -y rev2 +libext+.v top.v
```

Include the `+libext` compile time option to specify the file name extension of the files you want VCS to look for in these directories.

`+incdir+directory+`

Specifies the directory or directories that VCS searches for include files used in the ``include` compiler directive. You can specify multiple directories using the plus (+) character.

`+libext+extension+`

Specifies that VCS searches only for files with the specified file name extensions in a library directory. You can specify more than one extension, separating the extensions with the plus (+) character. For example, `+libext+.v+.V+` specifies searching for files with either the `.v` or `.V` extension in a library. The order in which you add file name extensions to this option does not specify an order in which VCS searches files in the library with these file name extensions.

`+liborder`

Specifies searching for module definitions for unresolved module instances through the remainder of the library where VCS finds the instance, then searching the next and then the next library on the `vcs` command line before searching in the first library on the command line.

`+liborder` and `+librescan` switches on elaboration command line will have impact only when the user specifies `-y/-v` on elaboration command line.

Options for 64-bit Compilation

`-full64`

Enables compilation and simulation in 64-bit mode.

Option to Specify Files and Compile Time Options in a File

`-file filename`

Specifies a file containing a list of files and compile-time options.

Options for Verdi

`-verdi`

This option starts Verdi.

Options for Starting Simulation After Compilation

`-R`

Runs the executable file immediately after VCS links it together.

Options for Changing Parameter Values

`-pvalue+parameter_hierarchical_name=value`

Changes the specified parameter to the specified value.

`-parameters filename`

Changes parameters specified in the file to values specified in the file. The syntax for a line in the file is as follows:

`assign value path_to_parameter`

The path to the parameter is similar to a hierarchical name except that you use the forward slash character (/) instead of a period as the delimiter.

Options for Controlling Messages

`-notice`

Enables verbose diagnostic messages.

`-q`

Quiet mode; suppresses messages such as those about the C compiler VCS is using, the source files VCS is parsing, the top-level modules, or the specified timescale.

`-V`

Verbose mode; compiles verbosely. The compiler driver program prints the commands it executes as it runs the C compiler, assembler, and linker.

Specifying a Log File

`-l filename`

Specifies a file where VCS records compilation messages. If you also enter the `-R` option, VCS records messages from both compilation and simulation in the same file.

Defining a Text Macro

```
+define+macro=value+
```

Defines a text macro in your source code to a value or character string. You can test this definition in your Verilog source code using the `'ifdef` compiler directive.

The `=value` argument is optional.

For example:

```
% vcs design.v +define+USETHIS
```

The macro is used inside the source file using the `'ifdef` compiler directive. If this macro is not defined using the `+define` option, then the `else` portion in the code takes priority.

```
`ifdef USETHIS
    package p1;
    endpackage
`else
    package p2;
    Endpackage
`endif
```

Simulation

During compilation, VCS generates a binary executable, `simv`. You can use `simv` to run the simulation. Based on how you compile the design, you can run your simulation using the following modes:

- [Interactive Mode](#)
- [Batch Mode](#)
- [Commonly Used Runtime Options](#)

For information on compiling the design, see [Compilation](#).

Interactive Mode

You can compile your design in interactive mode, also called debug mode, in the initial phase of your design cycle. In this phase, you require abilities to debug the design issues using a GUI or through the command line. To debug using a GUI, you can use Verdi, and to debug through the command-line interface, you can use the Unified Command-line Interface (UCLI).

Note:

To simulate the design in the interactive mode, compile the design using the `-debug_access (+<option>)` compile time option.

For information on compiling the design, see [Compilation](#).

Batch Mode

You can compile your design in batch mode, also called as optimized mode, when most of your design issues are resolved. In this phase, you can achieve better performance to run regressions and with minimum debug abilities.

Note:

The runtime performance reduces if you use the `-debug_access (+<option>)` option. Use this option only when you require runtime debug abilities.

The following command line simulates the design in batch mode:

```
% simv
```

Commonly Used Runtime Options

Use the following command line to simulate the design:

```
% executable [runtime_options]
```

By default, VCS generates the binary executable `simv`. However, you can use the compile time option, `-o` with the `vcs` command line to generate the binary executable with the specified name.

```
-gui
```

This option starts Verdi when `VERDI_HOME` is set.

```
-ucli
```

This option starts `simv` in UCLI mode.

For information on Verdi, see [Using Verdi](#).

For a complete list of runtime options, see [Simulation Options](#).

Partition Compile Flow

Compile your design using the Partition Compile flow to enhance compile productivity for the iterative process of compilation and recompilation phases. It works with both VCS three-step and two-step flows.

For more information, refer to the Partition Compile section in [Compile Time Productivity](#).

3

Modeling Your Design

Verilog coding style is the most important factor that affects the simulation performance of a design. How you write your design can make the difference between a fast error-free simulation, and one that suffers from race conditions and poor performance. This chapter describes some Verilog modeling techniques that helps you to simulate your designs most efficiently with VCS.

This chapter includes the following topics:

- [Avoiding Race Conditions](#)
- [Race Detection in Verilog Code](#)
- [Race Detection Tool to Identify Race between Clock and Data](#)
- [Optimizing Testbenches for Debugging](#)
- [Creating Models That Simulate Faster](#)
- [Creating Models That Simulate Faster](#)
- [Case Statement Behavior](#)
- [Precedence in Text Macro Definitions](#)
- [Memory Size Limits in the Simulator](#)
- [Using Sparse Memory Models](#)
- [Obtaining Scope Information](#)
- [Avoiding Circular Dependency](#)
- [Translating VHDL Package to SystemVerilog Package](#)
- [VITAL2000 Negative Constraint Calculation](#)

Avoiding Race Conditions

A race condition is defined as a coding style for which there is more than one correct result. Since the output of the race condition is unpredictable, it can cause unexpected problems during simulation. It is easy to accidentally code race conditions in Verilog. For

example, in *Digital Design with Verilog HDL* by Sternheim, Singh, and Trivedi, at least two of the examples provided with the book (adder and cachemem) have race conditions. VCS provides some tools for race detection.

Some common race conditions and ways of avoiding them are described in the following sections.

Using and Setting a Value at the Same Time

In this example, the two parallel blocks have no guaranteed ordering, so it is ambiguous whether the \$display statement will be executed.

```
module race;
  reg a;
  initial begin
    a = 0;
    #10 a = 1;
  end
  initial begin
    #10 if (a) $display("may not print");
  end
endmodule
```

The solution is to delay the \$display statement with a #0 delay:

```
initial begin
  #10 if (a)
    #0 $display("may not print");
end
```

You can also move it to the next time step with a non-zero delay.

Setting a Value Twice at the Same Time

In this example, the race condition occurs at time 10 because no ordering is guaranteed between the two parallel initial blocks.

```
module race;
  reg r1;
  initial #10 r1 = 0;
  initial #10 r1 = 1;
  initial
    #20 if (r1) $display("may not print");
endmodule
```

The solution is to stagger the assignments to register r1 by finite time, so that the ordering of the assignments is guaranteed. Note that using the non-blocking assignment (<=) in both assignments to r1 would not remove the race condition in this example.

Flip-Flop Race Condition

It is very common to have race conditions near latches or flip-flops. Here is one variant in which an intermediate node `a` between two flip-flops is set and sampled at the same time:

```
module test(out,in,clk);
    input in,clk;
    output out;
    wire a;
    dff dff0(a,in,clk);
    dff dff1(out,a,clk);
endmodule
module dff(q,d,clk);
    output q;
    input d,clk;
    reg q;
    always @ (posedge clk)
        q = d;      // race!
endmodule
```

The solution for this case is straightforward. Use the non-blocking assignment in the flip-flop to guarantee the order of assignments to the output of the instances of the flip-flop and sampling of that output. The change looks like this:

```
always @ (posedge clk)
    q <= d;      // ok
```

Or add a nonzero delay on the output of the flip-flop:

```
always @ (posedge clk)
    q = #1 d;      // ok
```

Or use a non-zero delay in addition to the non-blocking form:

```
always @ (posedge clk)
    q <= #1 d;      // ok
```

Note that the following change does not resolve the race condition:

```
always @ (posedge clk)
    #1 q = d;      // race!
```

The `#1` delay simply shifts the original race by one time unit, so that the intermediate node is set and sampled one time unit *after* the `posedge` of clock, rather than *on* the `posedge` of clock. Avoid this coding style.

If you are modeling flip-flops using sequential UDPs (User-Defined Primitives), note that VCS evaluates the output terminals of sequential UDP (User-Defined Primitive) in the NBA region. This can cause a race condition. The default behavior is required by the SystemVerilog LRM, IEEE Std 1800-2009.

Continuous Assignment Evaluation

Continuous assignments with no delay are sometimes propagated earlier in VCS than in Verilog-XL. This is fully correct behavior, but exposes race conditions such as the one in the following code fragment:

```
assign x = y;
initial begin
    y = 1;
    #1
    y = 0;
    $display(x);
end
```

In VCS, this displays 0, while in Verilog-XL, it displays 1, because the assignment of the value to `x` races with the usage of that value by the `$display`.

Another example of this type of race condition is the following:

```
assign state0 = (state == 3'h0);
always @(posedge clk)
begin
    state = 0;
    if (state0)
        // do something
end
```

The modification of `state` may propagate to `state0` before the `if` statement, causing unexpected behavior. You can avoid this by using the non-blocking assignment to `state` in the procedural code as follows:

```
state <= 0;
if (state0)
    // do something
```

This guarantees that `state` is not updated until the end of the time step, that is, after the `if` statement is executed.

Counting Events

A different type of race condition occurs when code depends on the number of times events are triggered in the same time step. For instance, in the following example, if `A` and `B` change at the same time, it is unpredictable whether `count` is incremented once or twice:

```
always @ (A or B)
count = count + 1;
```

Another form of this race condition is to toggle a register within the `always` block. If toggled once or twice, the result may be unexpected behavior.

The solution to this race condition is to make the code inside the `always` block insensitive to the number of times it is called.

Time Zero Race Conditions

The following race condition is subtle, but very common:

```
always @(posedge clock)
  $display("May or may not display");
initial begin
  clock = 1;
  forever #50 clock = ~clock;
end
```

This is a race condition because the transition of `clock` to 1 (`posedge`) may happen before or after the event trigger (`always @(posedge clock)`) is established. Often the race is not evident in the simulation result because reset occurs at time zero.

The solution to this race condition is to guarantee that no transitions take place at time zero of any signals inside event triggers. Rewrite the clock driver in the above example as follows:

```
initial begin
  clock = 1'bX;
  #50 clock = 1'b0;
  forever #50 clock = ~clock;
end
```

Race Detection in Verilog Code

VCS provides the following race detection tools:

- **Dynamic Race Detection Tool** - Finds the race conditions during simulation.
- **Static Race Detection Tool** - Finds the race conditions by analyzing source code during compilation.

The above two tools are described in the following sections:

- [The Dynamic Race Detection Tool](#)
- [The Static Race Detection Tool](#)

The Dynamic Race Detection Tool

This section consists of following topics:

- [Introduction to the Dynamic Race Detection Tool](#)
- [Enabling Race Detection](#)
- [The Race Detection Report](#)
- [Post-Processing the Report](#)
- [Debugging Simulation Mismatches](#)

Introduction to the Dynamic Race Detection Tool

The dynamic race detection tool finds two basic types of race conditions during simulation:

- [Read - Write Race Condition](#)
- [Write - Write Race Condition](#)

Read - Write Race Condition

The Read - Write race condition occurs when both Read and Write on a signal take place at the same simulation time.

Example:

```
initial
#5 var1 = 0; // write operation on signal var1

initial
#5 var2 = var1; // read operation on signal var2
```

Read

Procedural assignment in any one of the always or initial block, or a continuous assignment samples the value of signal var1 to drive signal var2.

Write

Procedural assignment in another always or initial block, or another continuous assignment assigns a new value to signal var1.

In the above example, at the simulation time 5, there is both read and write operation on signal var1. When simulation time 5 is over, you do not know if signal var2 will have the value 0 or the previous value of signal var1.

Write - Write Race Condition

The Write - Write race condition occurs when multiple writes on a signal take place at the same simulation time.

Example:

```
initial
#5 var1 = 0; // write operation on signal var1

initial
#5 var1 = 1; // write operation on signal var1
```

Write-Write

Value of the signal *var1* is non-deterministic when there are multiple concurrent procedural assignments on the same variable at the same simulation time.

In the above example, at simulation time 5, different initial blocks assign 0 and 1 to signal *var1*. When simulation time 5 is over, you do not know if *var1* signal value is 0 or 1.

Finding these race conditions is important because in Verilog simulation you cannot control the order of execution of statements in different always or initial blocks, or continuous assignments that execute at the same simulation time. This means that a race condition can produce different simulation results when you simulate a design with different, but both properly functioning Verilog simulators.

Even worse, a race condition can result in different simulation results with different versions of a particular simulator, or with different optimizations or performance features of the same version of a simulator.

Note:

`$dumpvars` can also expose races.

Also, sometimes modifications in one part of a design can cause hidden race conditions to surface even in unmodified parts of a design, thereby causing different simulation results from the unmodified part of the design.

The indications of a race condition are the following:

- Simulation results do not match when comparing simulators
- Design modifications cause inexplicable results
- Simulation results do not match between different simulation runs of the same simulator, when different versions or different optimization features of that simulator are used

Therefore, even when a Verilog design appears to be simulating correctly, and you see the results you want, you should look for race conditions and remove them so that you will continue to see the same simulation results from an unrevised design well into the future. Also, you should look for race conditions while a design is in development.

VCS can help you find these race conditions by writing report files about the race conditions in your design.

VCS writes the reports at runtime, but you should enable race detection at compile-time with a compile-time option.

The reports can be lengthy for large designs. You can post-process the report to generate another shorter report that is limited, for example, to only part of the design or to only between certain simulation times.

Enabling Race Detection

When you compile your design, you can enable race detection during simulation for your entire design or part of your design.

The `-race` compile-time option enables race detection for your entire design.

The `-racecd` compile-time option enables race detection for the part of your design that is enclosed between the `'race` and `'endrace` compiler directives.

Note:

The `-race` and `-racecd` compile-time options support dynamic race detection for both pure Verilog and SystemVerilog data types.

The Race Detection Report

While VCS simulates your design, it writes race detection reports to the `race.out` and `race.unique.out` files.

The `race.out` file contains a line for all race conditions that it finds at all times throughout the simulation. If VCS executes two different statements in the same time step for several times, the `race.out` file contains a line for each of these times.

The `race.unique.out` file contains only the lines for race conditions that are unique, and which have not been reported in a previous line.

Note:

The `race.unique.out` is automatically created by the `PostRace.pl` Perl script after the simulation. This script needs a perl5 interpreter. The first line of the script points to Perl at a specific location, see [Modifying the PostRace.pl Script](#). If that location at your site is not a perl5 interpreter, the script fails with syntax errors.

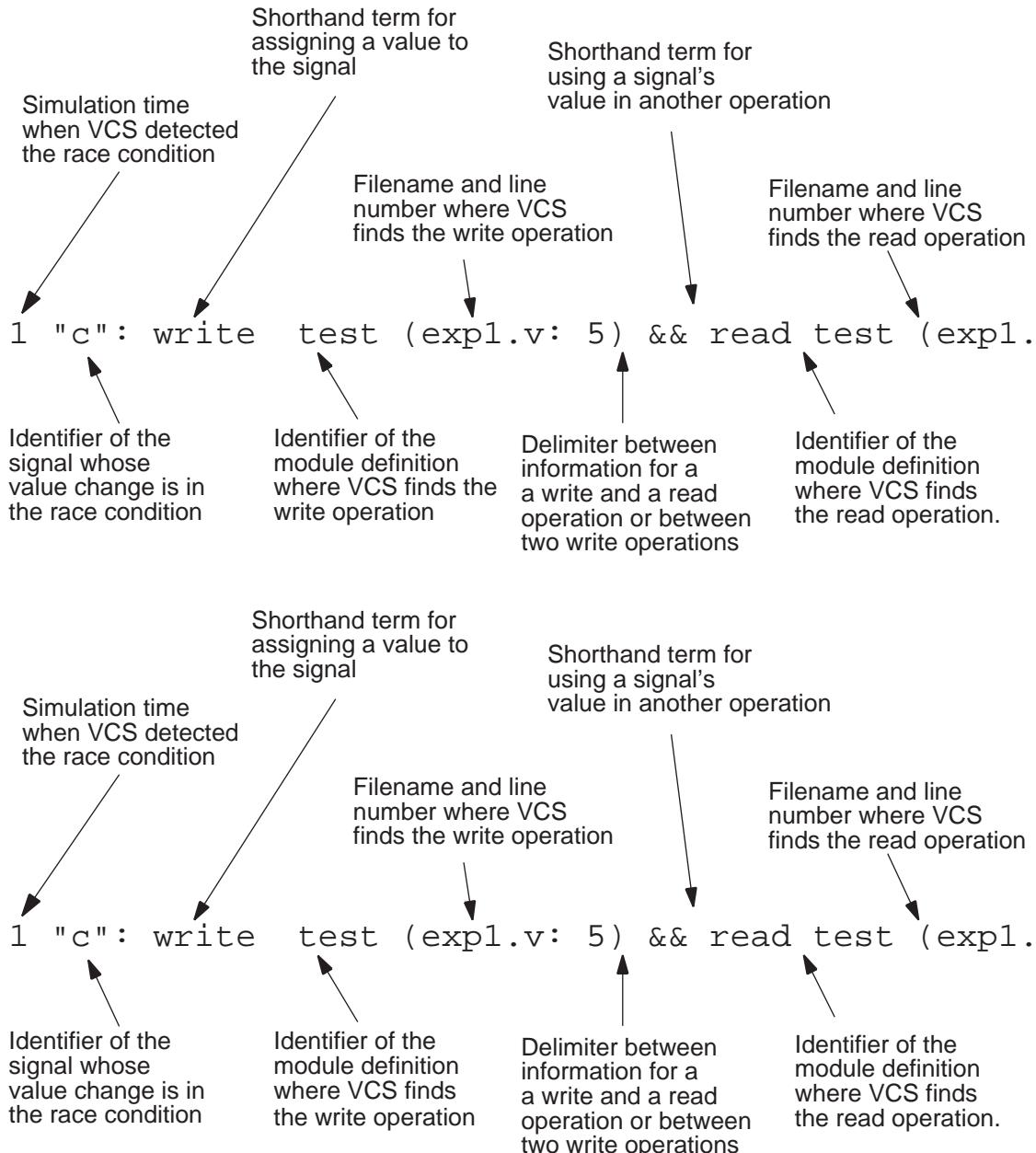
The report describes read-write and write-write race conditions. The following is an example of the contents of a small `race.out` file:

```
Synopsys Simulation VCS RACE REPORT

0 "c": write test (exp1.v: 5) && read test (exp1.v:23)
1 "a": write test (exp1.v: 16) && write test (exp1.v:10)
1 "c": write test (exp1.v: 5) && read test (exp1.v:17)
```

END RACE REPORT

The following explains a line in the `race.out` file:



The following is the source file, with line numbers added for this race condition report:

```
1. module test;
2.   reg a,b,c,d;
```

```

3.
4.   always @(a or b)
5.     c = a & b;
6.
7.   always
8.     begin
9.       a = 1;
10.      #1 a = 0;
11.      #2;
12.    end
13.
14.   always
15.     begin
16.       #1 a = 1;
17.       d = b | c;
18.       #2;
19.     end
20.
21. initial
22. begin
23.   $display("%m c = %b",c);
24.   #2 $finish;
25. end
26. endmodule

```

As stipulated in `race.out`:

- At simulation time 0, there is a procedural assignment to reg `c` on line 5, and also `$display` system task displays the value of reg `c` on line 23.
- At simulation time 1, there is a procedural assignment to reg `a` on line 10 and another procedural assignment to reg `a` on line 16.
- Also, at simulation time 1, there is a procedural assignment to reg `c` on line 5, and the value of reg `c` is in an expression that is evaluated in a procedural assignment to another register on line 17.

Races of No Consequence

Sometimes race conditions exist, such as write-write race to a signal at the same simulation time, but the two statements that are assigning to the signal are assigning the same value. This is a race of no consequence, and the race tool indicates this with `**NC` at the end of the line for the race in the `race.out` file.

```

0 "r4": write test (nc1.v: 40) && write test (nc1.v:44)**NC
20 "r4": write test (nc1.v: 40) && write test (nc1.v:44)**NC
40 "r4": write test (nc1.v: 40) && write test (nc1.v:44)**NC
60 "r4": write test (nc1.v: 40) && write test (nc1.v:44)
80 "r4": write test (nc1.v: 40) && write test (nc1.v:44)**NC

```

Post-Processing the Report

VCS comes with the `PostRace.pl` Perl script that you can use to post-process the `race.out` report to generate another report that contains a subset of the race conditions in the `race.out` file. You should include options on the command line for the `PostRace.pl` script to specify this subset. These options are as follows:

`-hier module_instance` Specifies the hierarchical name of a module instance. The new report lists only the race conditions found in this instance and all module instances hierarchically under this instance.

`-sig signal` Specifies the signal that you want to examine for race conditions. You can only specify one signal, and must not include a hierarchical name for the signal. If two signals in different module instances have the same identifier, the report lists race conditions for both signals.

`-minmax min max` Specifies the minimum (or earliest) and the maximum (or latest) simulation time in the report.

`-nozero` Omits race conditions that occur at simulation time 0.

`-uniq` Omits race conditions that also occurred earlier in the simulation. The output is the same as the contents of the `race.unique.out` file.

`-f filename` Specifies the name of the input file. Use this option if you have changed the name of the `race.out` file.

`-o filename` The default name of the output file is `race.out.post`. If you want a different name, specify it with this option.

You can enter more than one of these options on the `PostRace.pl` command line.

If you enter an option more than once, the script uses the last of these multiple entries.

Unless you specify a different name with the `-o` option, the report generated by the `PostRace.pl` script is in the `race.out.post` file.

The following is an example of the command line:

```
PostRace.pl -minmax 80 250 -f mydesign.race.out -o
mydesign.race.out.post
```

In this example, the output file is named `mydesign.race.out.post`, and reports the race conditions between 80 and 250 time units. The post-process file is named `mydesign.race.out`.

Modifying the PostRace.pl Script

The first line of the `PostRace.pl` Perl script is as follows:

```
#! /usr/local/bin/perl
```

If Perl is installed at a different location at your site, you must modify the first line of this script. This script needs a perl5 interpreter. You can find this script at the following location:

```
vcs_install_dir/bin/PostRace.pl
```

Debugging Simulation Mismatches

A design can contain several race conditions where many of them behave the same in different simulations, so they are not the cause of a simulation mismatch. For a simulation mismatch, you must find critical races. Critical races are the race conditions that cause the simulation mismatch. This section describes how to do this.

Add system tasks to generate VCD files to the source code of the simulations that mismatch. Recompile them with the `-race` or `-racedc` options and run the simulations again.

When you have two VCD files, find their differences with the `vcdiff` utility. This utility is located in the `vcs_install_dir/bin` directory. The command line for `vcdiff` is as follows:

```
vcdiff vcdfile1.dmp vcdfile2.dmp -options > output_filename
```

If you enter the `vcdiff` command without arguments, you see the usage information including the options.

Method 1: If the Number of Unique Race Conditions is Small

A unique race condition is a race condition that can occur several times during simulation, but only the first occurrence is reported in the `race.unique.out` file. If the number of lines in the `race.unique.out` file is smaller than the number of unique race conditions, then for each signal in the `race.unique.out` file:

1. Look in the output file from the `vcdiff` utility. If the signal values are different, you have found a critical write-write race condition.
2. If the signal values are not different, look for the signals that are assigned the value of this signal, or assigned expressions that include this signal (read operations).
3. If the values of these other signals are different at any point in the two simulations, note the simulation times of these differences on the other signals, and post-process the `race.out` file looking for race conditions in the first signal at around the simulation times of the value differences on the other signals. Specify simulation times before and after the time of these differences with the `-minmax` option. Enter:

```
PostRace.pl -sig first_signal -minmax time time2
```

If the `race.out.post` file contains the first signal, then it is a critical race condition, and must be corrected.

Method 2: If the Number of Unique Races is Large

If there are many lines in the `race.unique.out` file and a large number of unique race conditions, then the method of finding the critical race conditions is to do the following:

1. Look in the output file from the `vcdiff` utility for the simulation time of the first difference in simulation values.
2. Post-process the `race.out` file looking for races at the time of the first simulation value difference. Specify simulation times before and after the time of these differences with the `-minmax` option. Enter:

```
PostRace.pl -minmax time time2
```

3. For each signal in the resulting `race.out.post` file:
 - If the simulation values differ in the two simulations, then the race condition in the `race.out.post` file is a critical race condition.
 - If the simulation values are not different, check the signals that are assigned the value of this signal or assigned expressions that include this signal. If the values of these other signals are different, then the race condition in the `race.out.post` file is a critical race condition.

Method 3: An Alternative When the Number of Unique Race Conditions is Large

1. Look in the output file from the `vcdiff` utility for the simulation time of the first difference in simulation values.
2. For each signal that has a difference at this simulation time:
 - a. Traverse the signal dependency backwards in the design until you find a signal whose values are same in both simulations.
 - b. Look for a race condition on that signal at that time. Enter: `PostRace.pl -sig signal -minmax time time2`. If there is a race condition at that time on that signal, then it is a critical race condition.

The Static Race Detection Tool

It is possible for a group of statements to combine and form a loop, so that the loop is executed once by VCS and more than once by other Verilog simulators. This is a race condition.

These situations arise when level-sensitive sensitivity lists (event controls which immediately follow the `always` keyword in an `always` block, and which do not contain the `posedge` or `negedge` keywords) and procedural assignment statements in the `always` blocks combine with other statements such as continuous assignment or module instantiation statements to form a potential loop. It is observed that these situations do

not occur if the always blocks contain delays or other timing information, non-blocking assignment statements, or PLI calls through user-defined system tasks.

You can use the `+race=all` compile-time option to start the static race detection tool.

Note:

The `+race=all` compile-time option supports only pure Verilog constructs.

After compilation, the static race detection tool writes the file named `race.out.static` which reports the race conditions.

The following example shows an `always` block that combines with other statements to form a loop:

```

35      always @( A or C ) begin
36          D = C;
37          B = A;
38      end
39
40      assign C = B;

```

The `race.out.static` file from the compilation of this source code follows:

```

Race-[CLF] Combinational loop found
    "source.v", 35: The trigger 'C' of the always block can cause
        the following sequence of event(s) which can again trigger
        the always block.
    "source.v", 37: B = A;
        which triggers 'B'.
    "source.v", 40: assign C = B;
        which triggers 'C'.

```

Race Detection Tool to Identify Race between Clock and Data

VCS supports a race detection tool that finds the race condition between clock and data and generates the diagnostic output.

This race detect tool detects race for the following conditions:

- Whenever the clock and data arrives at the same time, VCS detects the race condition and provides the RTL information.
- Race is reported when the data is updated before the clock for the same timestamp.

This race detection tool helps you to know the following:

- Glitches in the design that are encountered during simulation.
 - Flops that are affected by the glitch during simulation.
 - The frequency and timing of glitches in the design.
-

Use Model

When you compile your design, you can enable the race detection tool during simulation for your entire design.

The `-hsopt=racedetect` option enables the race detection tool for your entire design.

If no clock-data race is detected during simulation, VCS reports the following information:

`Clock Data Race: No Race detected.`

VCS stores the RTL information in the `hsRaceInfo.db` file.

The format of the output is as follows:

```
Clock Data Race detected @ <n> Module: <Module_name> Instance:<br/>
<Instance_name> Line: <Line_no> File: <File_name> RTL_MOD_NAME:<br/>
<Module_name> Object: <Data_signal>
```

Here,

`n`

Time at which the race occurred.

`Module_name`

Module in which the race was detected.

`Instance_name`

Hierarchy pointing to the specific instance of the module.

`Line_no`

Line number pointing to the source file.

`File_name`

Name of the source file.

`Module_name`

Name of the module.

Data_signal

Data signal in the detected clock data race.

Examples

Consider the following example. The lines where the race is detected is highlighted in the example:

Example 2 The content of the test.v file.

```

`noinline
module top;
reg clk; reg clk1;
wire [31:0] o1, o2, o3, o4, o5, o6;
integer d; reg r;
bot1 b1(o1,o2,o3, clk,clk1, d, (d+1), (d+2));
bot2 b2(o4,o5,o6, clk,clk1, (d + 10), (d+11), (d+12));

always begin #10; clk <= ~clk; end
always begin #7; clk1 <= ~clk1; end
always begin #50; d = d + 1; end
always begin #21; d = d + 1; end

initial
begin
    r = 1'b1; clk = 1'b0; clk1 = 1'b0; d = 20;
    #200;
    $display(o1, o2, o3, o4, o5, o6); $finish;
end
endmodule

`noinline
module bot1(q,q1, q2, clk,clk1, d, d1,d2);
integer q,q1,q2;
output q,q1,q2;
input clk,clk1;
input [31:0] d,d1,d2;

always @ (posedge clk) q <= d;
always @ (posedge clk1) q1 <= d1;
always @ (posedge clk) q2 <= d2;
endmodule

`noinline
module bot2(q,q1, q2, clk, clk1, d, d1,d2);
integer q,q1,q2;
output q,q1,q2;

```

```

input clk,clk1;
input [31:0] d,d1,d2;
integer dtemp;

always @(posedge clk1) q <= d;
always @(posedge clk) q1 <= d1;
always @(posedge clk1) q2 <= d2;
endmodule

```

Compile the test cases as follows:

```

% vcs test.v -hsopt=racedetect
% simv
% cat hsRaceInfo.db

```

It generates the following output:

```

Clock Data Race detected @ 21 Module: bot1 Instance: top.b1          Object: d1
    Line: 33 File: data_2clk_race.v RTL_MOD_NAME: bot1
Clock Data Race detected @ 21 Module: bot2 Instance: top.b2          Object: d
    Line: 45 File: data_2clk_race.v RTL_MOD_NAME: bot2
    Line: 47 File: data_2clk_race.v RTL_MOD_NAME: bot2
Object: d2
Clock Data Race detected @ 50 Module: bot1 Instance: top.b1          Object: d
    Line: 32 File: data_2clk_race.v RTL_MOD_NAME: bot1
    Line: 34 File: data_2clk_race.v RTL_MOD_NAME: bot1
Object: d2
Clock Data Race detected @ 50 Module: bot2 Instance: top.b2          Object: d1
    Line: 46 File: data_2clk_race.v RTL_MOD_NAME: bot2
Clock Data Race detected @ 63 Module: bot1 Instance: top.b1          Object: d1
    Line: 33 File: data_2clk_race.v RTL_MOD_NAME: bot1
Clock Data Race detected @ 63 Module: bot2 Instance: top.b2          Object: d1
    Line: 45 File: data_2clk_race.v RTL_MOD_NAME: bot2
    Line: 47 File: data_2clk_race.v RTL_MOD_NAME: bot2
Object: d2
Clock Data Race detected @ 105 Module: bot1 Instance: top.b1         Object: d1
    Line: 33 File: data_2clk_race.v RTL_MOD_NAME: bot1
Clock Data Race detected @ 105 Module: bot2 Instance: top.b2         Object: d
    Line: 45 File: data_2clk_race.v RTL_MOD_NAME: bot2
    Line: 47 File: data_2clk_race.v RTL_MOD_NAME: bot2
Object: d2
Clock Data Race detected @ 147 Module: bot1 Instance: top.b1         Object: d1
    Line: 33 File: data_2clk_race.v RTL_MOD_NAME: bot1
Clock Data Race detected @ 147 Module: bot2 Instance: top.b2         Object: d
    Line: 45 File: data_2clk_race.v RTL_MOD_NAME: bot2
    Line: 47 File: data_2clk_race.v RTL_MOD_NAME: bot2
Object: d2
Clock Data Race detected @ 150 Module: bot1 Instance: top.b1         Object: d
    Line: 32 File: data_2clk_race.v RTL_MOD_NAME: bot1
    Line: 34 File: data_2clk_race.v RTL_MOD_NAME: bot1
Object: d2
Clock Data Race detected @ 150 Module: bot2 Instance: top.b2         Object: d1
    Line: 46 File: data_2clk_race.v RTL_MOD_NAME: bot2
Clock Data Race detected @ 189 Module: bot1 Instance: top.b1         Object: d1
    Line: 33 File: data_2clk_race.v RTL_MOD_NAME: bot1
Clock Data Race detected @ 189 Module: bot2 Instance: top.b2         Object: d
    Line: 45 File: data_2clk_race.v RTL_MOD_NAME: bot2
    Line: 47 File: data_2clk_race.v RTL_MOD_NAME: bot2
Object: d2

```

Limitations

The feature has the following limitations:

- RTL information is not provided for clock-only glitches.
- No race detect output file is generated if you are using any of the following VCS technologies:
 - X-propagation
 - Congruency
 - Partition Compile flow
 - Mixed-Signal design
 - FGP flow
 - `-dbsflagsbytearray` option

Optimizing Testbenches for Debugging

Testbenches typically execute debugging features, for example, displaying text in certain situations as specified with the `$monitor` or `$display` system tasks. Another debugging feature, which is typically enabled in testbenches, is writing simulation history files during simulation so that you can view the results after simulation.

Among other things, these simulation history files record the simulation times at which the signals in your design change value. These simulation history files can be either ASCII Value-Change-Dump (VCD) files that you can input into a number of third-party viewers, or binary FSDB files that you can input into Verdi.

The `$dumpvars` system task specifies writing a VCD file, and the `$fsdbDumpvars` system task specifies writing a FSDB file. You can use the `vcd2fsdb` utility to also convert VCD file into FSDB, which translates the VCD file to a FSDB file and then displays the results in Verdi.

Debugging features significantly slow down the simulation performance of any logic simulator including VCS. This is particularly true for operations that make VCS display text on the screen and even more so for operations that make VCS write information to a file. For this reason, you will want to be selective about where in your design and where in the development cycle of your design you enable debugging features. The following sections describe a number of techniques that you can use to choose when debugging features are enabled.

Conditional Compilation

Use `'ifdef`, `'else`, and `'endif` compiler directives in your testbench to specify which system tasks you want to compile for debugging features. Then, when you compile the design with the `+define` compile-time option on the command line (or when the `'define` compiler directive appears in the source code), VCS compiles these tasks for debugging features. For example:

```
initial
begin
`ifdef postprocess
$vcplusplus(0,design_1);
$vcplusplusdeltacycleon;
$vcplusplusglitchon;
`endif
end
```

In this case, the `vcs` command is as follows:

```
% vcs testbench.v design.v +define+postprocess
```

The system tasks in this initial block record several types of information in a FSDB file. In this particular case, the information is for all the signals in the design, so the performance cost is extensive. You would only want to do this early in the development cycle of the design when finding bugs is more important than simulation speed.

The command line includes the `+define+postprocess` compile-time option, which tells VCS to compile the design with these system tasks compiled into the testbench.

Later in the development cycle of the design, you can compile the design without the `+define+postprocess` compile-time option, and VCS does not compile these system tasks into the testbench. Doing so enables VCS to simulate your design much faster.

Advantages and Disadvantages

The advantage of this technique is that simulation can run faster than if you enable debugging features at runtime. When you use conditional compilation, VCS has all the information it needs at compile-time.

The disadvantage of this technique is that you have to recompile the testbench to include these system tasks in the testbench, thus increasing the overall compilation time in the development cycle of your design.

Synopsys recommends that you consider this technique as a way to prevent these system tasks from inadvertently remaining compiled into the testbench, later in the development cycle, when you want faster performance.

Enabling Debugging Features at Runtime

Use the `$test$plusargs` system function in place of the '`ifdef`' compiler directives. The `$test$plusargs` system function checks for a plusarg runtime option on the `simv` command line.

Note:

A plusarg option is an option that has a plus (+) symbol as a prefix.

An example of the `$test$plusargs` system function is as follows:

```
initial
if ($test$plusargs("postprocess"))
begin
$vcpluspluson(0,design_1);
$vcplusplusdeltacycleon;
$vcplusplusglitchon;
end
```

In this technique you do not include the `+define` compile-time argument on the `vcs` command line. Instead you compile the system tasks into the testbench and then enable the execution of the system tasks with the runtime argument to the `$test$plusargs` system function. Therefore, in this example, the `simv` command line is as follows:

```
% simv +postprocess
```

During simulation, VCS writes the VPD file with all the information specified by these system tasks. Later, you can execute another `simv` command line without the `+postprocess` runtime option. As a result, VCS does not write the VPD file, and therefore runs faster.

There is a pitfall to this technique. This system function matches any plusarg that has the function's argument as a prefix. For example:

```
module top;
initial
begin
if ( $test$plusargs("a") )
    $display("\n<< Now a >>\n");
else if ( $test$plusargs("ab") )
    $display("\n<< Now ab >>\n");
else if ( $test$plusargs("abc") )
    $display("\n<< Now abc >>\n");
end
endmodule
```

No matter whether you enter the `+a`, `+ab`, or `+abc` plusarg, when you simulate the executable, VCS always displays the following:

```
<<< Now a >>>
```

To avoid this pitfall, enter the longest plusarg first. For example, you would revise the previous example as follows:

```
module top;
initial
begin
if ( $test$plusargs("abc") )
    $display("\n<< Now abc >>\n");
else if ( $test$plusargs("ab") )
    $display("\n<< Now ab >>\n");
else if ( $test$plusargs("a") )
    $display("\n<< Now a >>\n");
end
endmodule
```

Advantages and Disadvantages

The advantage to using this technique is that you do not have to recompile the testbench to stop VCS from writing the VPD file. This technique is something to consider using, particularly early in the development cycle of your design, when you are fixing a lot of bugs and already doing a lot of recompilation.

The disadvantages to this technique are considerable. Compiling these system tasks, or any system tasks that write to a file, into the testbench requires VCS to compile the `simv` executable so that it is possible for it to write the VPD file when the runtime option is included on the command line. This means that the simulation runs significantly slower than if you don't compile these system tasks into the testbench. This impact on performance remains even when you don't include the runtime option on the `simv` command line.

Using the `$test$plusargs` system function forces VCS to consider the worst case scenario — plusargs are used at runtime — and VCS generates the `simv` executable with the corresponding overhead to prepare for these plusargs. The more fixed information VCS has at compile-time, the more VCS can optimize `simv` for efficient simulation. Alternatively, the more user control at runtime, the more overhead VCS has to add to `simv` to accept runtime options, and the less efficient the simulation.

For this reason, Synopsys recommends that if you use this technique, you should plan to abandon it fairly early in the development cycle and switch to either the conditional compilation technique for writing simulation history files, or a combination of the two techniques.

Combining the Techniques

Some users find that they have the greatest amount of control over the advantages and disadvantages of these techniques when they combine them. Consider the following example:

```
`ifdef comppostprocess
initial
  if ($test$plusargs ("runpostprocess"))
    begin
      $vcpluspluson(0,design_1);
      $vcplusplusdeltacycleon;
      $vcplusplusglitchon;
    end
`endif
```

In this instance, both the `+define+comppostprocess` compile-time option and the `+runpostprocess` runtime option are required for VCS to write the VPD file. This technique allows you to avoid recompiling just to prevent VCS from writing the file during the next simulation and also provides you with a way to recompile the testbench, later in the development cycle, to exclude these system tasks without first editing the source code for the testbench.

Creating Models That Simulate Faster

When modeling your design, for faster simulation, use higher levels of abstraction. Behavioral and RTL models simulate much faster than gate and switch level models. This rule of thumb is not unique to VCS; it applies to all Verilog simulators and even all logic simulators in general.

What is unique to VCS are the acceleration algorithms that make behavioral and RTL models simulate even faster. In fact, VCS is particularly optimized for RTL models for which simulation performance is critical.

These acceleration algorithms work better for some designs than for others. Certain types of designs prevent VCS from applying some of these algorithms. This section describes the design styles that simulate faster or slower.

The acceleration algorithms apply to most data types and primitives and most types of statements, but not all of them. This section also describes the data types, primitives, and types of statements that you should try to avoid.

VCS is optimized for simulating sequential devices. Under certain circumstances, VCS infers that an `always` block is a sequential device and simulates the `always` block much faster. This section describes the coding guidelines you should follow to make VCS infer an `always` block as a sequential device.

When writing an `always` block, if you cannot follow the inferencing rules for a sequential device, there are still things that you should keep in mind so that VCS simulates the `always` block faster. This section also describes the guidelines for coding faster simulating `always` blocks that VCS infers to be combinatorial instead of sequential devices.

Unaccelerated Data Types, Primitives, and Statements

VCS cannot accelerate certain data types and primitives. VCS also cannot accelerate certain types of statements. This section describes the data types, primitives, and types of statements that you should try to avoid.

Avoid Unaccelerated Data Types

VCS cannot accelerate certain data types. The following table lists these data types:

Data Type	Description in IEEE Std 1364-2001
time and realtime	Page 22
real	Page 22
named event	Page 138
trireg net	Page 26
integer array	Page 22

Avoid Unaccelerated Primitives

VCS cannot accelerate `tranif1`, `tranif0`, `rtranif1`, `rtranif0`, `tran`, and `rtran` switches. They are defined in the IEEE Std 1364-2001.

Avoid Calls to User-defined Tasks or Functions Declared in Another Module

VCS cannot accelerate user-defined tasks or functions declared in another module. For example:

```
module bottom (x,y);
.
.
.
always @ y
top.task_identifier(y,rb);
endmodule
```

Avoid Strength Specifications in Continuous Assignment Statements

Omit strength specifications in continuous assignment statements. For example:

```
assign net1 = flag1;
```

Simulates faster than:

```
assign (strong1, pullo) net1= flag1;
```

Continuous assignment statements are described in the IEEE Std 1364-2001.

Inferring Faster Simulating Sequential Devices

VCS is optimized to simulate sequential devices. If VCS can infer that an `always` block behaves like a sequential device, VCS can simulate the `always` block much faster.

The IEEE Std 1364-2001 defines `always` constructs on page 149. Verilog users commonly use the term `always` block when referring to an `always` construct.

VCS can infer whether an `always` block is a combinatorial or sequential device. This section describes the basis on which VCS makes this inference.

Avoid Unaccelerated Statements

VCS does not infer an `always` block to be a sequential device if it contains any of the following statements:

Statement	Description in IEEE Std 1364-2001
<code>force</code> and <code>release</code> procedural statements	Page 126-127
<code>repeat</code> statements	Page 134-135, see the other looping statements on these pages and consider them as an alternative.
<code>wait</code> statements, also called as level-sensitive event controls	Page 141
<code>disable</code> statements	Page 162-164
<code>fork-join</code> block statements, also called as parallel blocks	Page 146-147

Using either blocking or non-blocking procedural assignment statements in the `always` block does not prevent VCS from inferring a sequential device, but in VCS blocking procedural assignment statements are more efficient.

Synopsys recommends zero delay non-blocking assignment statements to avoid race conditions.

IEEE Std 1364-2001 describes blocking and non-blocking procedural assignment statements on pages 119-124.

Place Task Enabling Statements in Their Own always Block and Use No Delays

IEEE Std 1364-2001 defines tasks and task enabling statements on pages 151-156.

VCS infers that an `always` block that contains a task enabling statement is a sequential device only when there are no delays in the task declaration.

All Sequential Controls Must Be in the Sensitivity List

To borrow a concept from VHDL, the sensitivity list for an `always` block is the event control that immediately follows the `always` keyword.

IEEE Std 1364-2001 defines event controls on page 138 and mentions sensitivity lists on page 139.

For correct inference, all sequential controls must be in the sensitivity list. The following code examples illustrate this rule:

- VCS does not infer the following DFF to be a sequential device:

```
always @ (d)
  @ (posedge clk) q <= d;
```

Even though `clk` is in an event control, it is not in the sensitivity list event control.

- VCS does not infer the following latch to be a sequential device:

```
always begin
  wait clk; q <= d; @ d;
end
```

There is no sensitivity list event control.

- VCS infers the following latch to be a sequential device:

```
always @ (clk or d)
  if (clk) q <= d;
```

The sequential controls, `clk` and `d`, are in the sensitivity list event control.

Avoid Level-Sensitive Sensitivity Lists Whose Signals are Used “Completely”

VCS infers a combinational device instead of a sequential device if the following conditions are both met:

- The sensitivity list event control is level sensitive.
 A level sensitive event control does not contain the `posedge` or `negedge` keywords.
- The signals in the sensitivity list event control are used “completely” in the `always` block.

Used “completely” means that there is a possible simulation event if the signal has a true or a false (1 or 0) value.

The following code examples illustrate this rule:

Example 1

VCS infers that the following `always` block is combinatorial, not sequential:

```
always @ (a or b)
  y = a or b
```

Here, the sensitivity list event control is level sensitive and VCS assigns a value to `y` whether `a` or `b` are true or false.

Example 2

VCS also infers that the following `always` block is combinatorial, not sequential:

```
always @ (sel or a or b)
  if (sel)
    y=a;
  else
    y=b;
```

Here, the sensitivity list event control is also level sensitive and VCS assigns a value to `y` whether `a`, `b`, or `sel` are true or false. Note that the `if-else` conditional statement uses signal `sel` completely, VCS executes an assignment statement whether `sel` is true or false.

Example 3

VCS infers that the following `always` block is sequential:

```
always @ (sel or a or b)
  if (sel)
    y=a;
```

In this instance, there is no simulation event when signal `sel` is false (0).

Modeling Faster `always` Blocks

Whether VCS infers an `always` block to be a sequential device or not, there are modeling techniques you should use for faster simulation.

Place All Signals Being Read in the Sensitivity List

The sensitivity list for an `always` block is the event control that immediately follows the `always` keyword. Place all nets and registers, whose values you are assigning to other registers, in the `always` block, and place all nets and registers, whose value changes trigger simulation events, in the sensitivity list control.

Use Blocking Procedural Assignment Statements

In VCS, blocking procedural assignment statements are more efficient.

Synopsys recommends zero delay non-blocking procedural assignment statements to avoid race conditions.

IEEE Std 1364-2001 describes blocking and non-blocking procedural assignment statements on pages 119-124.

Avoid force and release Procedural Statements

IEEE Std 1364-2001 defines these statements on pages 126-127. A few occurrences of these statements in combinatorial `always` blocks does not noticeably slow down simulation, but their frequent use does lead to a performance cost.

Using Verilog 2001 Constructs

In G-2012.09 and newer releases, Verilog 2001 or V2K source code conforms to the Verilog IEEE Std 1364-2001 instead of the Verilog IEEE Std 1364-1995.

If your Verilog code contains a V2K keyword as an identifier, you can tell VCS not to recognize V2K keywords with the `-v95` compile time option, for example:

```
module cell (....,....);
```

The module identifier `cell` is a keyword in Verilog 2001, so to use it as an identifier, include the `-v95` compile-time option.

The following table lists the implemented constructs in IEEE Std 1364-2001 and whether you need a compile-time option to use them.

IEEE Std 1364-2001 Construct	Default
comma separated event control expressions: <code>always @ (r1,r2,r3)</code>	yes

IEEE Std 1364-2001 Construct	Default
name-based parameter passing: <code>modname #(param_name(value)) inst_name(sig1,...);</code>	yes
ANSI-style port and argument lists: <code>module dev(output reg [7:0] out1, input wire [7:0] w1);</code>	yes
initialize a reg in its declaration: <code>reg [15:0] r2 = 0;</code>	yes
conditional compiler directives: <code>ifndef</code> and <code>'elseif</code>	yes
disabling the default net data type: <code>'default nettype</code>	yes
signed arithmetic extensions: <code>reg signed [7:0] r1;</code>	yes
file I/O system tasks: <code>\$open \$fsanf \$scanf</code> and more	yes
passing values from the runtime command line: <code>\$value\$plusarg</code> system function	yes
indexed part-selects: <code>reg1[8+:5]=5'b11111;</code>	yes
multi-dimensional arrays: <code>reg [7:0] r1 [3:0] [3:0];</code>	yes
maintaining file name and line number: <code>'line</code>	yes
implicit event control expression lists: <code>always @*</code>	yes
the power operator: <code>r1=r2**r3;</code>	yes
attributes: <code>(* optimize_power=1 *)module dev (res,out,clk,data1,data2);</code>	yes
generate statements	yes
localparam declarations	yes
Automatic tasks and functions <code>task automatic t1();</code>	requires the -svverilog compile-time option
constant functions <code>localparam lp1 = const_func(p1);</code>	yes
parameters with a bit range <code>parameter bit [7:0][31:0] P = {32'd1,32'd2,32'd3,32'd4,32'd5,32'd6,32'd7,32'd8};</code>	requires the -svverilog compile-time option

Case Statement Behavior

The IEEE Std 1364-2001 standards for the Verilog language state that you can enter the question mark character (?) in place of the z character in the `casex` and `casez` statements. The standard does not specify that you can also make this substitution in the `case` statements, and you might infer that this substitution is not allowed in the `case` statements.

VCS, like other Verilog simulators, does not make this inference, and allows you to also substitute ? for z in the `case` statements. If you do, remember that z does not stand for “don’t care” in a `case` statement, like it does in a `casez` or `casex` statement. In a `case` statement, z stands for the usual high impedance, and therefore so does ?.

Precedence in Text Macro Definitions

In text macros, the line continuation character (\) has a higher precedence than the one line comment characters (//). This means that VCS can merge a subsequent line with the text in a one-line comment, for example:

```
`define print_me_1 \
$display( "Hello 1" ); // just a comment \
$display( "I'm OK" );
```

VCS merges the second `$display` system task with the comment on the previous line and does not display the text string I'm OK.

The following are the precedence rules for text macro definitions:

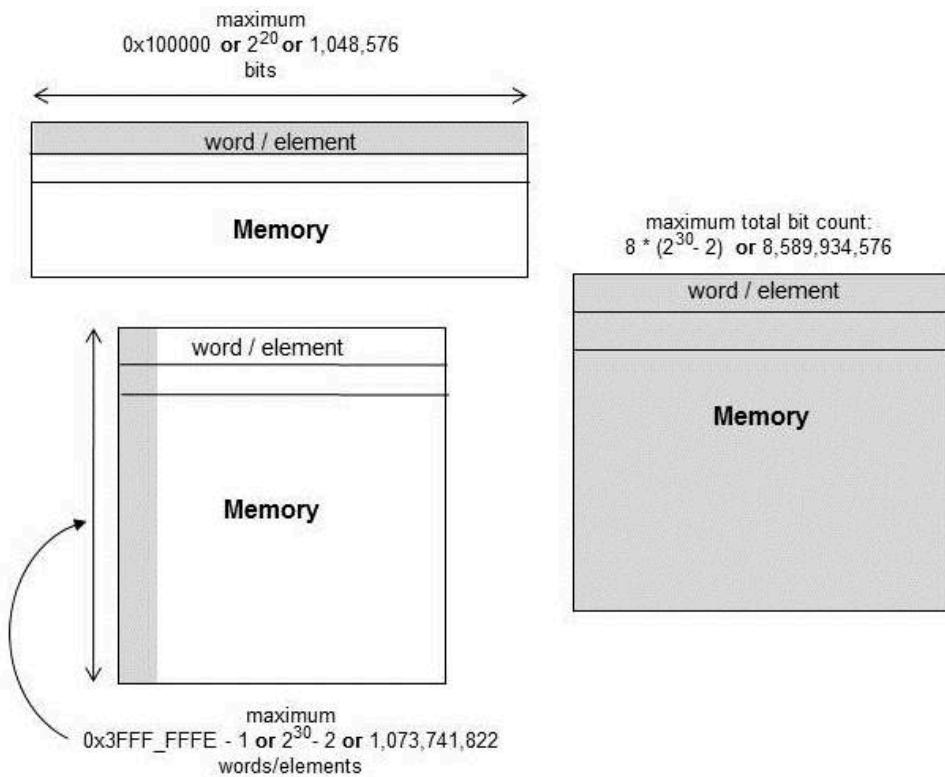
1. The ``undef` compiler directive has a higher precedence than the `+define` compile-time option.
2. The `+define` compile-time option has a higher precedence than the ``define` and ``undefineall` compiler directives.

Memory Size Limits in the Simulator

The bit width for a word or an element in a memory in VCS must be less than 0x100000 (or 220 or 1,048,576) bits.

The number of elements or words (sometimes also called rows) in a memory in VCS must be less than 0x3FFF_FFFE-1 (or 230 - 2 or 1,073,741,822) elements or words.

The total bit count of a memory (total number of elements * word size) must be less than 8 * (1024 * 1024 * 1024 - 2) or 8589934576.



The memory consumption of `reg` and `bit` variables is not the same. The `reg` data type has 4 states and the `bit` data type has 2 states.

Consider the following testcase:

```
module top;
    bit [127:0] mem_bit [0:1<<26];
    reg [127:0] mem_reg [0:1<<26];
    initial begin mem_bit = '{default: 'b0}; end
    initial begin mem_reg = '{default: 'b1}; end
endmodule
```

Memory Size Calculation of Bit Data Type

The bit data type is a 2-state variable and requires 1 bit to store. In the above testcase, the memory consumed by the bit variable `mem_bit` is:

$$1 * (2^{**} 26 * 128) = 8,589,934,592 \text{ bits (1GB)}$$

The size of the memory is within the (2GB -1) limit.

Memory Size Calculation of Reg Data Type

The reg data type is a 4-state variable and requires 2 bits to store. In the above testcase, the memory consumed by the reg variable `mem_reg` is:

$$2 * (2^{26} * 128) = 17,179,869,184 \text{ bits (2GB)}$$

The size of memory is outside range ($> 2\text{GB } -1$), so VCS issues the `Error-[MTL] Memory Too Large` error message. To resolve this error message, you should reduce the size of the memory as follows:

```
reg [127:0] mem_reg [0:1<<25];
```

Note:

- Starting with the VCS S-2020.12 version, sparse memory optimizations are enabled by default. VCS treats huge Verilog memories ($> 2\text{GB } -1$) as sparse memories (allocates memory dynamically) and does not issue the `Error-[MTL] Memory Too Large` error message for such memories.
- By default, an information message is generated when Address Space Layout Randomization (ASLR) is enabled on your machine. You can use the `-suppress=ASLR_DETECTED_INFO` runtime option to suppress printing of this message in standard output (stdout) and the runtime log file.

Using Sparse Memory Models

If your design contains a large memory, the `simv` executable needs large amounts of machine memory to simulate it. However, if `/*sparse*/` is specified, the large memory does not occupy the IP space, so the above 2G-1 size limit (See [Memory Size Limits in the Simulator](#)) does not exist. The maximum memory size depends on address space size. If `/*sparse*/` is not specified, both full 64-bit and 32-bit VCS have the same limitation (2G-1 size limit), because even with full 64-bit, VCS still uses 32-bit IP index in back-end and runtime. So, if the memory size exceeds 2G, simulation will have errors.

You can use the `/*sparse*/` pragma or meta comment in the memory declaration to specify a sparse memory model. For example:

```
reg /*sparse*/ [31:0] pattern [0:10_000_000];
integer i, j;
initial
begin
    for (j=1; j<10_000; j=j+1)
        for (i=0; i<10_000_000; i=i+1_000)
            pattern[i] = i+j;
end
endmodule
```

In simulations, this memory model uses 4 MB of machine memory with the `/*sparse*/` pragma, 81 MB without it.

The larger the memory, and the fewer elements in the memory that your design reads or writes to, the more machine memory you will save by using this feature. It is intended for memories that contain at least a few MBs. If your design accesses 1% of its elements you could save 97% of machine memory. If your design accesses 50% of its elements, you save 25% of machine memory. Do not use this feature if your design accesses more than 50% of its elements because using the feature in these cases may lead to more memory consumption than not using it.

Note:

- Sparse memory models cannot be manipulated by PLI applications through `tf` calls (the `tf_nodeinfo` routine issues a warning for sparse memory and returns NULL for the memory handle).
- Sparse memory models cannot be used as a personality matrix in PLA system tasks.

Obtaining Scope Information

VCS has custom format specifications (IEEE Std 1364-2001 does not define these) for displaying scope information. It also has system functions for returning information about the current scope.

Scope Format Specifications

The IEEE Std 1364-2001 describes the `%m` format specification for system tasks for displaying information such as `$write` and `$display`. The `%m` specification tells VCS to display the hierarchical name of the module instance that contains the system task. If the system task is in a scope lower than a module instance, it tells VCS to do the following:

- In named begin-end or fork-join blocks, it adds the block name to the hierarchical name.
- In user-defined tasks or functions, it considers the hierarchical name of the task declaration or function definition as the hierarchical name of the module instance.

VCS has the following additional format specifications for displaying scope information:

`%i`

Specifies the same as `%m` with the following difference: when in a user-defined task or function, the hierarchical name is the name of an instance or named block containing the task enabling statement or function call, not the hierarchical name of the task or function declaration.

If the task enabling statement is in another user-defined task, the hierarchical name is the name of an instance or named block containing the task enabling statement for this other user-defined task.

If the function call is in another user-defined function, the hierarchical name is the name of an instance or named block containing the function call for this other user-defined function.

If the function call is in a user-defined task, the hierarchical name is the name of an instance or named block containing the task enabling statement for this user-defined task.

`%-i`

Specifies that the hierarchical name is always of a module instance, not a named block or user-defined task or function. If the system task (such as `$write` and `$display`) is in:

- A named block — the hierarchical name is that of the module instance that contains the named block
- A user-defined task or function — the hierarchical name is that of the module instance containing the task enabling statement or function call

Note:

The `%i` and `%-i` format specifications are not supported with the `$monitor` system task.

The following commented code example shows what these format specifications do:

```
module top;
reg r1;

task my_task;
input taskin;
begin
$display("%m");           // displays "top.my_task"
$display("%i");           // displays "top.d1.named"
$display("%-i");          // displays "top.d1"
end
endtask

function my_func;
input taskin;
begin
$display("%m");           // displays "top.my_func"
$display("%i");           // displays "top.d1.named"
$display("%-i");          // displays "top.d1"
end
endfunction

dev1 d1 (r1);
endmodule
```

```
module dev1(inport);
  input inport;

  initial
  begin:named
    reg namedreg;
    $display("%m"); // displays "top.d1.named"
    $display("%i"); // displays "top.d1.named"
    $display("%-i"); // displays "top.d1"
    namedreg=1;
    top.my_task(namedreg);
    namedreg = top.my_func(namedreg);
  end

endmodule
```

Note:

- Scope hierarchical names using the %m format specifier always starts with a black slash(/) and has a white space added to the Verilog instance name, when Verilog is the last instance in the hierarchy. The following example illustrates a case where scope hierarchy name uses %m format specifier.

Table 2 Content of the test.v and test.vhd file

test.v	test.vhd
<i>module leaf(input clk); initial \$display ("%m"); bot bt_inst(); sub_leaf inst();endmodule: leaf module sub_leaf(); initial \$display("%m"); bot2 inst();endmodulemodule bot2; initial \$display("%m");endmodule</i>	<i>entity top is end top; architecture arch of top isbegin leaf_inst : entity work.leaf;end;entity bot isend bot;architecture arch of bot isbegin sub_leaf_inst : entity work.sub_leaf;end;</i>

Run the files using the following commands:

- vlogan -sverilog test.v
- vhdlan test.vhd
- vcs top
- ./simv

The output generated is as follows:

```
\TOP.LEAF_INST .inst
\TOP.LEAF_INST .inst.inst
\TOP.LEAF_INST
\TOP.LEAF_INST.bot_inst.SUB_LEAF_INST
\TOP.LEAF_INST.bot_inst.SUB_LEAF_INST .inst
```

In the output from [Table 2](#), you see that the scope hierarchical names starts with a black slash(\) and a white space is added to the Verilog instance name

`\TOP.LEAF_INST inst`

Returning Information About the Scope

The `$activeinst` system function returns information about the module instance that contains this system function. The `$activescope` system function returns information about the scope that contains the system function. This scope can be a module instance, a named block, a user-defined task, or a function in a module instance.

When VCS executes these system functions, it performs the following:

1. Stores the current scope in a temporary location.
2. If there are no arguments, it returns a pointer to the temporary location. Pointers are not used in Verilog, but they are in DirectC applications.

The possible arguments are hierarchical names. If there are arguments, it compares them from left to right with the current scope. If an argument matches, the system function returns a 32-bit non-zero value. If none of the arguments match the current scope, the system function returns a 32-bit zero value.

The following example contains these system functions:

```
module top;
reg r1;
initial
r1=1;
dev1 d1(r1);
endmodule

module dev1(in);
input in;
always @ (posedge in)
begin:named
if ($activeinst("top.d0","top.d1"))
$display("%i");
if ($activescope("top.d0.block","top.d1.named"))
$display("%-i");
end
endmodule
```

The following is an example of a DirectC application that uses the `$activeinst` system function:

```

extern void showInst(input bit[31:0]);           declaration of C function named showInst

module discriminator;
task t;
reg[31:0] r;           $activeinst system function without arguments
begin                 passed to the C function
    showInst($activeinst);
    if($activeinst("top.cl", "top.c3"))
    begin
        r = $activeinst;
        $display("for instance %i the pointer is %s", r ? "non-zero" : "zero");
    end
end
endtask
endmodule

```



```

module child;
initial discriminator.t;
endmodule

```



```

module top;
child c1();
child c2();
child c3();
child c4();
endmodule

```

In task *t*, the following occurs:

1. The `$activeinst` system function returns a pointer to the current scope, which is passed to the C function `showInst`. It is a pointer to a volatile or temporary char buffer containing the name of the instance.
2. A nested begin block executes only if the current scope is either `top.cl` or `top.c3`.
3. VCS displays whether `$activeinst` points to a zero or non-zero value.

The C code is as follows:

```

#include <stdio.h>

void showInst(unsigned str_arg)
{
    const char *str = (const char *)str_arg;
    printf("DirectC: [%s]\n", str);
}

```

Function `showInst` declares the `char` pointer `str` and assigns to it the value of its parameter, which is the pointer in `$activeinst` in the Verilog code. Then with a `printf` statement, it displays the hierarchical name that `str` is pointing to. Notice that the function begins the information it displays with `DirectC:` so that you can differentiate it from what VCS displays.

During simulation, VCS and the C function display the following:

```
DirectC: [top.c1]
for instance top.c1 the pointer is non-zero
DirectC: [top.c2]
DirectC: [top.c3]
for instance top.c3 the pointer is non-zero
DirectC: [top.c4]
```

Avoiding Circular Dependency

The `$random` system function has an optional seed argument. You can use this argument to make the return value of this system function the assigned value in a continuous assignment, procedural continuous assignment, or `force` statement. For example:

```
assign out = $random(in);

initial
begin
  assign dr1 = $random(in);
  force dr2 = $random(in);
```

When you do this, you might set up a circular dependency between the seed value and the statement, resulting in an infinite loop and a simulation failure.

This circular dependency doesn't usually occur, but it can occur, so VCS displays a warning message when you use a seeded argument with these kinds of statements. This warning message is as follows:

```
Warning-[RWSI] $random() with a 'seed' input
$random in the following statement was called with a 'seed' input
This may cause an infinite loop and an eventual crash at runtime.
"expl.v", 24: assign dr1 = $random(in);
```

The warning message ends with the source file name and line number of the statement, followed by the statement itself.

This possible circular dependency does not occur either when you use a seed argument and the return value is the assigned value in a procedural assignment statement, or when you do not use the seed argument in a continuous, procedural continuous, or `force` statement.

For example:

```
assign out = $random();

initial
begin
  assign dr1 = $random();
  force dr2 = $random();
  dr3 = $random(in);
```

These statements do not generate the warning message.

You can tell VCS not to display the warning message by using the `+warn=noRWSI` compile-time argument and option.

Translating VHDL Package to SystemVerilog Package

VCS mixed-HDL simulation supports the translation of VHDL types and constants in a package to its equivalent SystemVerilog types and constants. The VHDL package is translated automatically when the `-gen_sv_pkg` option is included during the analysis phase.

Use Model

To translate the VHDL types and constants in a package to its equivalent SystemVerilog package, include the `-gen_sv_pkg` option during the analysis phase. All packages must be self-contained. That is, the types, constants, and so on declared within a package might not use types, constants, and so on declared in another package unless the dependent entities are declared in STANDARD or IEEE packages. Such types, constants, and so on are not translated and a warning message is displayed.

See the following table [Table 3](#) for the supported VHDL types and constants and its equivalent mapping in the SystemVerilog package.

By default, the translated package is saved in the invocation directory of `vhdlan` (the target directory). You can override the default directory using the `-gen_sv_pkg_dir <target directory>` option.

For example, consider this VHDL package:

```
bit_conv.vhd

library IEEE;
use IEEE.STD_LOGIC_1164.all;

package bit_pkg is
    type bit_null_t is array (0 to 0) of bit;
    type s_t is array (1024 downto 0) of std_logic;
    type s_vec_t is array (0 to 1024) of std_ulogic_vector(1 downto 0);
    type bit_mda is array (integer range 0 to 1024, integer range 0 to 1024)
        of bit_vector(1 downto 0);
    TYPE bit_range is array (-10 to 25) of bit;
    TYPE pet IS (dog,cat,bird,horse,kid);
    TYPE pet_it IS ARRAY (pet RANGE dog TO cat) OF bit;
    TYPE Byte IS ARRAY (7 DOWNTO 0) OF bit;
    TYPE Memory IS ARRAY (0 TO 2**16-1) OF Byte;
    subtype s1_t is bit;
    subtype s1_vec_t is bit_vector(1024 downto 0);
```

```

subtype s1_vec_t_2d is s1_vec_t;
subtype s1_vec_t_3d is s1_vec_t_2d;
type int_t is array (7 downto 0) of integer;
type nat_t is array (7 downto 0) of natural;
type pos_t is array (7 downto 0) of positive;

end bit_pkg;

package body bit_pkg is
end bit_pkg;

```

Use these command lines to convert the `bit_conv.vhd` package to its equivalent SystemVerilog package, `bit_pkg.sv`. Analyze the resultant file using the `-sverilog` switch.

```
% vhdlan -nc bit_conv.vhd -gen_sv_pkg
% vlog -sverilog bit_pkg.sv
```

This SystemVerilog package is generated:

`bit_pkg.sv`

```

package bit_pkg;
    typedef bit [0:0] bit_null_t;
    typedef logic [1024:0] s_t;
    typedef logic [1:0] s_vec_t [0:1024];
    typedef bit [1:0] bit_mda [0:1024][0:1024];
    typedef bit [-10:25] bit_range;
    typedef enum {
        dog=0,
        cat=1,
        bird=2,
        horse=3,
        kid=4
    } pet;
    typedef bit [0:1] pet_it;
    typedef bit [7:0] Byte;
    typedef Byte Memory [0:65535];
    typedef bit s1_t;
    typedef bit [1024:0] s1_vec_t;
    typedef bit [1024:0] s1_vec_t_2d;
    typedef bit [1024:0] s1_vec_t_3d;
    typedef int int_t [7:0];
    typedef int nat_t [7:0];
    typedef int pos_t [7:0];
endpackage

```

VHDL Type Mapping

This table lists the mapping of VHDL types with its equivalent SystemVerilog types:

Table 3 VHDL Type Mapping With Its SystemVerilog Equivalent

VHDL Type	SystemVerilog
User defined Enumerated type	2-state enumeration type
character	byte
string	byte array
bit	bit
boolean	bit
std_logic, std_ulogic	logic
integer, discrete ranged types, and subtypes	int
real	real
Constrained array of real	Unpacked array of real
Constrained std_logic_vector, std_ulogic_vector	1 dimensional packed array of logic (signed if there is a 'use ieee.std_logic_signed.all')
Constrained unsigned	1 dimensional packed array of logic (unsigned)
Constrained signed	1 dimensional packed array of logic (signed)
1 dimensional constrained array of std_[u]logic	1 dimensional packed array of logic (unsigned)
1 dimensional constrained array of bit, Boolean	1 dimensional packed array of bit (unsigned)
1 dimensional constrained array of integer or enumerated type	1 dimensional unpacked array of int or 2-state enum type (Verilog does not allow packed int/enum array)
Record with elements of packable field types: std_[u]logic, array of std_[u]logic/, integer, bit, boolean, enumeration, array of bit, array of boolean, and other records with packable fields.	Packed struct
Records containing types that do not map to packable field types	Unpacked struct

Table 3 VHDL Type Mapping With Its SystemVerilog Equivalent (Continued)

VHDL Type	SystemVerilog
1- or N-dimensional: array of std_[u]logic_vector, bit_vector array of std_[u]logic, bit	Respectively: unpacked array (1-dimensional/N- dimensional) of packed logic array same as above
Multidimensional array (M1:M2,N1:N2) of std_[u]logic_vector, bit_vector	Unpacked array ([M1:M2] [N1:N2]) of packed logic array
1-dimensional array of packable record	Unpacked array of packed struct
Arrays of records containing types that do not map to packable field types	Not supported
Multidimensional array of scalar types other than std_[u]logic and bit, record types, and array types other than std_[u]logic_vector and bit_vector	Not supported

Packable field types are types that are allowed in packed arrays and structs in Verilog. These are single bit data types (bit, logic, reg), integral types (byte, shortint, int, longint, integer), and two-state enumerated types along with arrays and structs thereof.

Note:

The translation does not happen for the unsupported types and a warning message is displayed. However, the translation proceeds to the subsequent types. You can see these details as a comment in the translated SystemVerilog package, which states the reason for non-translation.

Limitations

Following are the VHDL types that are not supported for translation:

- Character enumeration types.
- Physical types.
- Unconstrained array types.
- File types.
- Access types.
- Non-type package constructs, such as functions, procedures, shared variables, and package signals.

- Declarations containing extended identifiers.
- VHDL resolved types. These cannot have their resolution function translated. However, the data type without the resolution function is translated.

In addition, these conditions disable translation:

- Types or constants that are named using a SystemVerilog keyword.
- Arrays containing null ranges.
- Types dependent upon other types that have not been translated. This excludes the types dependent upon constants that have not been translated; the evaluated value of constant is used rather than a reference to the constant itself. The named constant declarations in the package are also translated into the named constant declarations in the resultant SystemVerilog package. The value of the constant (the RHS of the declaration) is also translated by value rather than by reference similar to types.
- Enumeration types containing character literals (except built-in enumeration types, such as std_ulogic, bit, and character)

VITAL2000 Negative Constraint Calculation

VCS supports the VITAL2000 Negative Constraint Calculation (NCC) algorithm.

Following are the key changes in the VITAL2000 NCC algorithm:

- Vector support for VITAL generics has been added.
- Additional errors and warnings are reported by the VITAL2000 NCC routines:
 - After the NCC stage, if the resulting value of any timing generic sub-element is negative, the value is set to 0 ns, and a warning is issued.
 - A vector timing generic must be CrossArc or ParallelArc. Any other size of this timing generic is not allowed.

CrossArc is a vector timing arc whose size is equal to the product of the sizes of the two ports corresponding to the timing generic. ParallelArc is a vector timing arc whose size is equal to the individual sizes of the two ports corresponding to the timing generic.

Using VITAL2000 NCC

VCS, by default, applies the VITAL95 NCC algorithm to VITAL models. To use VITAL2000 NCC, set the `USE_VITAL_2000` variable to `TRUE` in the `synopsys_sim.setup` file.

The VCS four-step approach for negative constraint calculation remains unchanged for VITAL2000 NCC. For more information about this approach, see the “Negative Constraint Calculation (NCC)” section.

Disabling VITAL2000 Conformance Checks

To disable conformance checks while using VITAL2000 NCC, set `RELAX_CONFORMANCE` and `RELAX_CONFORMANCE_2K` variables to `TRUE` in the `synopsys_sim.setup` file as shown below:

```
RELAX_CONFORMANCE_2K = TRUE  
RELAX_CONFORMANCE = TRUE
```

By default, both these variables are set to `FALSE`.

4

Compiling/Elaborating the Design

This chapter describes the following sections:

- [Compiling/Elaborating the Design in the Debug Mode](#)
 - [Optimizing Simulation Performance for Desired Debug Visibility With the -debug_access Option](#)
 - [Dynamic Loading of DPI Libraries at Runtime](#)
 - [Dynamic Loading of PLI Libraries at Runtime](#)
 - [Key Compilation or Elaboration Features](#)
-

Compiling/Elaborating the Design in the Debug Mode

Debug mode, also called interactive mode, is typically used (but not limited to):

- During your initial phase of the design, when you need to debug the design using debug tools, such as Verdi or UCLI.
- When you are using PLIs.
- When you use UCLI commands to force a signal to write into registers/nets.

VCS provides the following compile-time options for the debug mode:

```
-debug_access (+<option>),  
-debug_region=(<option>) (+<option>)
```

The following examples show how to compile the design in full, partial, and minimum debug modes:

Compiling/Elaborating the Design in the Partial Debug Read Mode

```
% vcs -debug_access+r [compile_options] TOP.v
```

Compiling/Elaborating the Design in the Full Debug Mode

```
% vcs -debug_access+all [compile_options] TOP.v
```

Compiling/Elaborating the Design With the Desired Debug Capability

```
% vcs -debug_access<+options> [compile_options] TOP.v
```

For more information on `-debug_access` and `-debug_region` options, see the [Optimizing Simulation Performance for Desired Debug Visibility With the `-debug_access` Option](#) section.

Optimizing Simulation Performance for Desired Debug Visibility With the `-debug_access` Option

You can use the `-debug_access` option at compile time to selectively enable the required debug capabilities in a simulation. You can optimize simulation performance by enabling only the required debug capabilities.

This following topics describe this feature in detail:

- [Use Model](#)
- [Specifying Design Regions for `-debug_access` Capabilities](#)
- [Automatic Debug Capability Addition Using `\$fsdbDumpvars\(level,path\)`](#)
- [Enabling Additional Debug Capabilities](#)
- [Reduction in the Objects Being Dumped](#)
- [Using `-debug_access` With Tab Files \(With `-P` Option\)](#)
- [Unused Tab File Calls](#)
- [Including Tab Files](#)
- [Interaction With Other Debug Options](#)

Use Model

Following is the use model of the `-debug_access` option:

```
-debug_access (+<option_name>) *
```

[Table 4](#) describes the supported options of `-debug_access`.

The `-debug_access` option without any additional `option_name` enables VPD and FSDB dumping capability. This option allows you to enter the UCLI prompt and use only the `run`, `quit`, and `dump` commands.

Table 4 Supported Options of `-debug_access`

Option Name	Description
r	This option enables the read capability for the entire design. This enables PLI access to <code>get value</code> , and enables the UCLI <code>get</code> command. This is the minimum debug option to invoke the Verdi interactive mode.
w	This option applies write (deposit) capability to the registers and variables for the entire design.
wn	This option applies write (deposit) capability to the nets for the entire design.
fn	This option applies force capability to the nets for the entire design. This option is equivalent to <code>-debug_access+r+fn</code> .
fwn	This option applies write (deposit) and force capability to all nets in the design. This option is equivalent to <code>-debug_access+r+wn+fn</code> .
f	This option enables the following: Read capability on registers, variables, and nets Write (deposit) capability on registers and variables Force capability on registers, variables, and nets. This option is equivalent to <code>-debug_access+r+w+fn+f</code> .
drivers	This option enables driver debugging capability. This option is equivalent to <code>-debug_access+r+drivers</code> .
line	This option enables line debugging. It allows you to use the commands for step/next and line breakpoints. This option is equivalent to <code>-debug_access+r+line</code> .
cbk	This option enables dumping of SystemVerilog string type signals and PLI-based callbacks on nets, registers, and variables. This option is equivalent to <code>-debug_access+r+cbk</code> .
cbkd	This option enables both dumping and PLI-based callbacks on dynamic nets, registers, and variables defined in classes. Class object debugging is also enabled. This option is equivalent to <code>-debug_access+r+line+cbkd</code> .
thread	This option enables the debugging of the SystemVerilog threads. This option is equivalent to <code>-debug_access+r+thread</code> .
class	This option enables debugging of the SystemVerilog classes and class objects, but the capability is also applied to the remaining portion of the design as specified by the <code>-debug_region</code> option. This option is equivalent to: <code>-debug_access+r+w+thread+line+cbk+cbkd</code>
pp	This option enables write capability on registers and variables, callbacks for the entire design, driver capability, and assertion debug capability. This option is equivalent to: <code>-debug_access+w+cbk+drivers</code>
dmptf	This option enables dumping of ports and internal nodes/memories of tasks/functions.

Table 4 *Supported Options of `-debug_access` (Continued)*

Option Name	Description
all	This option enables all the above options. This option is equivalent to: <code>-debug_access+r+w+wn+f+fn+fwn+drivers+line+cbk+cbkd+thread+class+pp+dmptf</code>
<code>-memcbk</code>	The <code>-debug_access-memcbk</code> option disables callbacks for memories and multidimensional arrays (MDAs). By default, <code>-debug_access</code> enables callbacks for memories and MDAs. <i>For more information, see the Incrementally Removing Debug Capabilities section.</i>
reverse	This option enables the reverse debugging feature.
designer	<p>This option enables the following debug capabilities:</p> <ul style="list-style-type: none"> • Read and write for both variables and nets • PLI and value change callbacks and line callback • FSDB/VPD dumping capability • Debugging or dumping of cell ports • Class and thread debugging (no SystemVerilog Testbench debugging) • Driver debugging <p>This option is equivalent to: <code>-debug_access+line+r+w+wn+cbk+debug_region+cellports</code></p> <p>You can use the <code>-debug_access+designer</code> option for debugging design and simulation mismatches.</p> <p>This option does not support the following:</p> <ul style="list-style-type: none"> • Class and thread debugging (no SystemVerilog Testbench debugging) • Driver debugging
simctrl	<p>This option does not support the following: Class and thread debugging (no SystemVerilog Testbench debugging)</p> <p>Driver debugging</p> <p>Line stepping</p> <p>This option enables the following debug capabilities:</p> <ul style="list-style-type: none"> • PLI and value change callbacks • Assertion control (not assertion dumping) • FSDB/VPD dumping capability <p>You can use the <code>-debug_access+simctrl</code> option for simulation control. This option is equivalent to: <code>-debug_access+r+cbk+assert_c</code></p> <p>For more information on <code>+assert_c</code>, see the Assertion Debug Support section.</p> <ul style="list-style-type: none"> • Class and thread debugging (no SystemVerilog Testbench debugging) • Driver debugging • Line stepping
verbose	Reports the summary of all the <code>-debug_access</code> and <code>-debug_region</code> options passed to the vcs command line.

Example: `-debug_access+r+line+class+drivers`

Key Points to Note

- The `-debug_access` options are case-insensitive.
- Read capability is disabled by default with the `-debug_access` option.
- The following abbreviation of `-debug_access` is supported:
`-debug_acc`
- The dynamic value change callbacks are enabled as part of the `all` and `class` options.

Incrementally Removing Debug Capabilities

VCS allows you to remove the debug capabilities specified with the `-debug_access` option. The following is the syntax to remove the debug capabilities:

`-debug_access (+) * (-) *`

VCS removes the debug capabilities of the options specified with the “-” sign. For example, you can use the `-debug_acc+all-f` option to remove force capability.

Key Points to Note

- VCS issues a warning message when you specify the same option with `-debug_acc` for both addition and removal of debug capability. For example, VCS issues a warning message when both `-debug_acc+all-f` and `-debug_acc+f` options are specified.
- If the `-debug_acc+all` option is specified initially and later if you specify `-debug_acc-`, the final option is as follows: `-debug_acc+all-`. For example, if both `-debug_acc+all` and `-debug_acc-f` options are specified, the final option `-debug_acc+all-f` removes the force capability.
- VCS issues a warning message if you use `-all`, `-class`, `-designer`, or `-simctrl` options with `-debug_access`.
- If you try to remove the capability that has not been added, VCS issues a warning message saying that the specified `-<option>` is ignored.
- Multiple additions/subtractions of the same capability are treated as a single addition/subtraction. No warning message is issued in this case.
- It is recommended to use this feature with the `all`, `class`, `designer`, and `simctrl` options of `-debug_access`.

Assertion Debug Support

The following assertion debug options are supported with `-debug_access`:

Table 5 Assertion Debug Options of `-debug_access`

Option	Description
<code>-assert_d</code>	This option allows you to remove the assertion dumping capabilities from the <code>-debug_access+designer</code> and <code>-debug_access+simctrl</code> options.
<code>+assert_c</code>	This option enables assertion control capabilities.
<code>+assert_f</code>	This option enables the following: <ul style="list-style-type: none">Dumping failures and successes for both concurrent and immediate assertionsAssertions control

Verdi One Search Support

The Verdi One Search feature allows you to search for design information in a database.

By default, One Search feature searches for symbols inside the `simv` executable. Shared libraries that are dynamically linked to `simv` are not searched. The `-debug_access+idents_so` option enables searching inside the shared libraries.

The Verdi One Search feature uses an internal database to quickly perform identifier search in the design. The `-debug_access+idents_db` option allows you to build the database at compile time for Verdi One Search.

Reporting Global Debug Capability Diagnostics

You can use the `-debug_report` option at compile time to enable the reporting of debug capability added to the design.

The diagnostics are reported in an ASCII text file named `debug.report`. This file is generated in the current working directory and cannot be renamed during compilation. You can rename the file after compilation. There is no VCS option to rename the file.

The debug capability diagnostics report allows you to identify the source of debug capability and to reduce debug capability in an informed manner.

The following information is recorded in the debug diagnostics report file:

- Debug capability derived from compile options. You can view this diagnostic report after compilation.
- Debug capability declared within a tab file. This diagnostic report contains File IDs associated with the capability.
- Global Verilog functionality that enables debug capability.

Example

Consider the following files for the global debug capability diagnostic report:

Example 3 `top.sv`

```
module dutA(input clk, output reg a,b);
    always@ (negedge clk)
        begin
            a=clk;
            b=a;
        end
    endmodule

`celldefine
module dutB(input clk, output reg a);
    wire fgh;
    always@ (posedge clk)
        a=clk;
    endmodule
`endcelldefine

module mytop;
    reg clk=0;
    wire a [0:2];
    dutA d1(clk,a[0],a[2]);
    dutB d2(clk,a[1]);
    initial begin
        clk=0;
        forever #1 clk = ~clk;
    end
    initial #100 $finish;
endmodule
```

Example 4 `pli.tab`

```
acc+=frc:*
```

To compile the `top.sv` file, execute the following command:

```
% vcs -P pli.tab -debug_access+cbk -debug_report top.sv
```

To view the global debug capability diagnostic report, execute the following command:

```
% cat debug.report
```

Following is the debug diagnostic report file:

Figure 1 *Debug Diagnostic Report*

```
1 Here is the union of all individual -debug_access options: -debug_access+r+cbk+cbks
2
3 Global           Source          FileID
4 -----
5 acc+=frc:*
6     tab file      1
7 debug_access+cbk vcs compile command
8 +vcsd            -debug_access
9 acc+=s:*
10 acc+=s:*
11 acc+=r,cbk:*
12 +vcsd            addSaifTasksToPliTab
13 acc+=frcr,frcn:*
14 acc+=s:*
15 acc+=r,cbk:*
16
17
18 FileID    FileName
19 -----
20 2        /global/snps_apps/vcsmx_2018.09-Beta/linux/lib/vcsdp_lite.tab
21 3        /global/snps_apps/verdi_2018.09-Beta/share/PLI/VCS/LINUX/verdi.tab
22 1        pli.tab
23
24 Modules with -debug_access caps disabled due to cell module debug caps turned off by default:
25 dutB
```

The tab file diagnostic report contains the file ID associated with the capability.

This diagnostic is reported as dutB is the celldefine module and the design is not compiled with the -debug_region+cell option.

Limitations

This feature does not support the following compile-time option:

- The `+optconfigfile` option as it is replaced by the instance-based tab file.

Specifying Design Regions for `-debug_access` Capabilities

You can use the `-debug_region` option to have better control over the performance of `-debug_access`. The `-debug_region` option enables you to apply debugging capabilities to the desired portion of a design.

You must use the `-debug_region` option along with the `-debug_access` option at compile time.

Following is the use model of `-debug_region`:

`-debug_region(option_name) (option_name)*`

[Table 6](#) describes the options supported by `-debug_region`.

Table 6 Test

Option Name	Description	Default Functionality if <code>-debug_region</code> is not specified
<code>+cell</code>	Applies debug capabilities to both real cell modules and the ports of real cell modules. Cell modules are Verilog modules that are bound with <code>`celldefine</code> and <code>`endcelldefine</code> compiler directives, as described in <i>Verilog 1800-2012 LRM, section 22.10</i> .	Debug capability is not applied to both real cell modules and the ports of real cell modules.
<code>+cellports</code>	Applies debug capabilities only to the ports of real cell modules and lib cell modules.	Debug capability is not added to the ports of real cell modules.
<code>-lib</code>	The <code>-debug_region-lib</code> option removes debug capabilities from libraries (files passed to VCS with the preceding <code>-v/-y</code> compiler options).	Debug capabilities are applied to the libraries.
<code>+encrypt</code>	Applies debug capabilities to the fully-encrypted instances (modules, programs, packages, and interfaces).	Debug capability is not applied to the fully-encrypted instances.
<code>=sv</code>	The <code>-debug_region=sv</code> option applies debug capabilities only to the program, package, interface or module containing SystemVerilog constructs, and to SystemC. Debug capabilities are applied to SystemVerilog code inside cells and fully encrypted blocks only when the <code>+cell</code> option is also used. Debug capabilities are applied to the Verilog code inside fully encrypted modules only when the <code>+encrypt</code> option is also used.	Debug capability is applied to Verilog, VHDL, and SystemC.
<code>=verilog</code>	The <code>-debug_region=verilog</code> option applies debug capabilities to all Verilog code and to SystemC. Debug capabilities are applied to the Verilog code inside cells only when the <code>+cell</code> option is also used. Debug capabilities are applied to the Verilog code inside fully encrypted modules only when the <code>+encrypt</code> option is also used.	Debug capability is applied to Verilog, VHDL, and SystemC.

Table 6 Test (Continued)

Option Name	Description	Default Functionality if <code>-debug_region</code> is not specified
<code>=vhdl</code>	The <code>-debug_region=vhdl</code> option applies debug capabilities to the VHDL code and to SystemC.	Debug capability is applied to Verilog, VHDL, and SystemC.

Note:

- VCS applies debug capability to the standard packages by default.
- The `-debug_region` options are case-insensitive.

Example

- `-debug_access+class -debug_region+cell`
 Applies class debug capabilities to the design and cell modules.

Key Points to Note

- The `-debug_region` option works only for the capabilities specified by the `-debug_access` option. It has no effect on the capabilities specified in tab files or configuration files.
- An error message is issued if you use `-debug_region` without an option.
- An error message is issued if you use `-debug_region` without `-debug_access`.

Region Debug Enhancements

The `-debug_region` functionality is enhanced to include the following features:

- Apply debug capabilities to a specific instance hierarchy
- Apply debug capabilities to the partially encrypted modules

This enhancement gives you more flexibility in applying debug capabilities to a specific instance hierarchy of the design, thereby reducing the number of objects having debug capability.

Debugging Desired Instance of a Design Using `-debug_region`

Following is the use model for debugging desired instance of a design using `-debug_region`:

`-debug_region=level, path(<option>) *`

or,

```
-debug_region+level, path(<option>) *
```

Where,

- `path` identifies the hierarchical path of the specified instance. An error message is issued if the specified `path` does not exist in the design. If `path` is a top module, and only one top module exists in the design, a warning message is issued.

`path` can also include escaped names, but for this case you must enclose the command in double quotes. For example,

```
% vcs -debug_access "-debug_region=0, \my%top .dut" ...
```

- `level` specifies the hierarchical depth of the instance. If `level=0`, then debug capability is applied to the specified instance and all instances under it.

The `-debug_region=level, path` option applies debug capabilities only to the specified instances of a module. The application of debug capability stops when a leaf instance, cell, fully encrypted instance, or an mixed HDL boundary is detected.

You must specify the `-debug_region+cell` option to consider cells (modules that are wrapped with ``celldefine`` - ``endcelldefine`` compiler directives) for debugging. If `-debug_region+cell` is not specified, the application of debug capability stops if a cell is detected when traversing down the `level` of hierarchy.

For allowing debug of cells only in a specific design branch, you must use the `-debug_region=level, path+cell` or `-debug_region=level, path -debug_region +cell` option.

You can specify multiple `-debug_region=level, path` commands as follows:

```
-debug_region=level1, path1 -debug_region=level2, path2
```

or,

```
-debug_region=level, path1, path2
```

For example,

```
-debug_region=0, path1 -debug_region=0, path2
```

In this case `-debug_access` capability is applied to both module instances starting with `path1` and `path2`.

Examples:

- `-debug_region=0, my_top.dut`

Applies `-debug_access` capabilities only to the module instances starting at instance `my_top.dut` and all module instances under `my_top.dut`. The application of debug

capability stops when a leaf instance, cell, fully encrypted instance, or an mixed HDL boundary is reached.

- `-debug_region=0,my_top.dut+cell`

Applies `-debug_access` capabilities to the following:

- Module instances starting at instance `my_top.dut` and all module instances under `my_top.dut`.
- Cells under `my_top.dut`.

- `-debug_region=2,my_top.dut`

Applies `-debug_access` capabilities only to the module instances starting at instance `my_top.dut` and the child instances of `my_top.dut`. Where, `my_top.dut` is the first level and the child instances are second level.

- `-debug_region=0,my_top.dut+dut`

This option combination is not allowed. An error message is issued in this case.

- `-debug_region=0,my_top.dut+tb`

This option combination is not allowed. An error message is issued in this case.

Key Points to Note

- You cannot use the `-debug_region=level,path` option in combination with the `-debug_region=sv|verilog|vhdl` option.
- The `-debug_region` option applies only to the Verilog portion of a design. If you have a mixed design and the `-debug_region=level,path` option is specified, then the debug capabilities are applied only to the Verilog portion of the specified instance.
- For a mixed design, `-debug_access` is applied to the entire VHDL portion of the design.

Automatic Debug Capability Addition Using `$fsdbDumpvars(level,path)`

If the design is not compiled with any `-debug_access*` option and `VERDI_HOME` is set, you can use the `$fsdbDumpvars(level,path)` system task to automatically add debug capability to the applicable portion of the design being dumped. For example,

- `$fsdbDumpvars()` automatically adds `-debug_access` option to the compile command.
- `$fsdbDumpvars(level,path)` automatically adds `-debug_access -debug_region=level,path` options to the compile command.

Enabling Additional Debug Capabilities

This section consists of the following subsections:

- [Driver/Load Debug Capability](#)
- [Statement Debug Capability](#)
- [Force/Deposit Debug Capability](#)
- [Class Debug Capability](#)

Driver/Load Debug Capability

By default, the driver/load debug support is disabled. You must use the `-debug_access+drivers` option at compile time to enable the driver/load debug support. This option enables the following capabilities:

- Active drivers
- UCLI show driver/load

Note:

An error message is issued when you use driver/load debug functionality without specifying `-debug_access+drivers`.

Statement Debug Capability

By default, statement debugging is disabled. In UCLI, the `step`, `next`, and `file/line` breakpoints are disabled. To enable the statement debug capability, use the `-debug_access+line` option at compile time.

Force/Deposit Debug Capability

By default, changing the value of a signal, variable, or net is disabled. In UCLI, the `force` command is disabled, and in VPI, the `vpi_put_value()` function is disabled. To enable value change debug capability, use the following options at compile time:

- `-debug_access+w`: For writing (depositing) values to the registers or variables.
- `-debug_access+wn`: For writing (depositing) values to the nets.
- `-debug_access+f`: For writing (depositing) values to the registers and variables, and for forcing values onto the registers, variables, and nets.
- `-debug_access+fn`: For forcing values onto the nets.

Class Debug Capability

Class debugging capability enables line stepping, object IDs, thread debugging, and write capability.

This allows for object-browser debugging and the usage/display of object IDs in Verdi. It also allows for constraint debugging and thread debugging in Verdi/UCLI.

By default, class debugging is disabled. To enable class debugging, use the `-debug_access+class` option at compile time.

Reduction in the Objects Being Dumped

The `-debug_access` option does not dump the ports of tasks and functions by default. You can use the `-debug_access+dmpf` option to dump the ports of tasks and functions.

Using `-debug_access` With Tab Files (With `-P` Option)

If you use `-debug_access` with tab files, the capabilities of `-debug_access` and the tab files are combined. If you have a tab file with force capability applied using the `-P` option on a module (for example, `testmod`), and `-debug_access+r` is specified, then the debug access capability is applied to all instances, but the force capability is applied only to the instances of the `testmod` module.

The `-debug_access` capabilities are ignored if the design is also compiled with `+applylearn`. In this case, interactive debug mode is enabled.

Unused Tab File Calls

VCS does not apply the debug capabilities of unused tab file calls to the design, except when the design contains SystemC or Spice usage. If a tab file call is marked “persistent”, then the associated debug capabilities are applied to the design.

Including Tab Files

The `-debug_access` option automatically includes all the compile options required for VPD and FSDB dumping. There is no need to specify additional options to enable dumping, adding tab files, or adding PLI objects to link with `simv`.

Dumping FSDB

If the `VERDI_HOME` environment variable is set, you can use `-debug_access` to dump FSDB. There is no need to specify Verdi PLI and tab files on the VCS command line to dump FSDB.

Interaction With Other Debug Options

If you specify multiple `-debug_access` options on the same command line, then the functionality is combined. For example, specifying `-debug_acc+w -debug_acc+drivers` is equivalent to `-debug_acc+w+drivers`.

Dynamic Loading of DPI Libraries at Runtime

This feature is the implementation of the SV LRM appendix on including non-Verilog or non-SystemVerilog code, through the DPI, in a design or testbench. For details, see, Annex J, “Inclusion of foreign language code” in *SystemVerilog LRM IEEE Std. 1800-2012*.

For partition compile, if you declare an import DPI function, and you do not provide the C source code on the VCS command line, VCS displays the following error message:

```
ibvcspc_test_IAwm9b.so: undefined reference to `my_export_dpi'
collect2: ld returned 1 exit status
```

With dynamic loading, this error condition with partition compile and C/C++ source code does not occur.

Use Model

Dynamic loading of a DPI shared library at runtime requires a number of steps before the `simv` command line. These steps are as follows:

- ▶ For three-step flow, analyze and elaborate the Verilog or SystemVerilog code. For example:

```
%vlogan -sverilog other_options test.v
% vcs top_level
```

For two-step flow, compile the Verilog or SystemVerilog code. For example:

```
%> vcs -sverilog other_options test.v
```

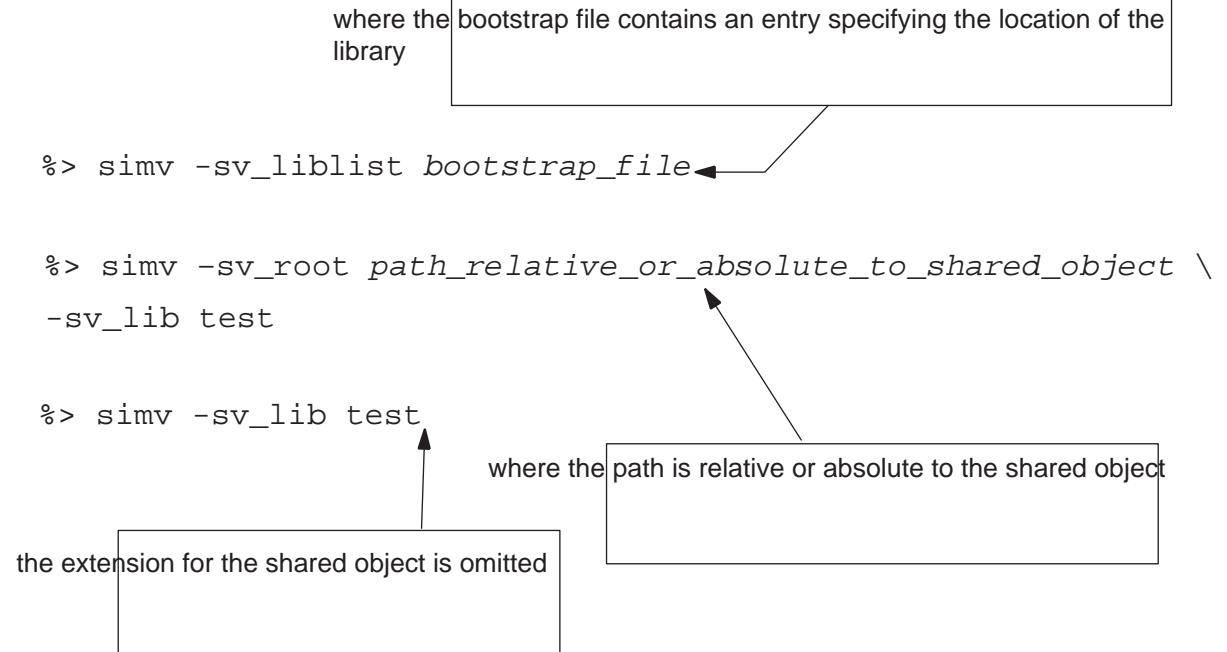
1. Compile the C code and create a shared object, for example:

```
%> gcc -fPIC -Wall ${CFLAGS} -I${VCS_HOME}/include \
other_libraries -c test.c
%> gcc -fPIC -shared ${CFLAGS} -o test.so test.o
```

2. Load the shared object at runtime using one of the following runtime options:

```
-sv_lib -sv_root -sv_liblist
```

For example, `simv` command lines for loading the shared object are as follows:



The following is an example of a bootstrap file:

```
#!SV_LIBRARIES
myclibs/lib1
myclibs/lib3
proj1/clibs/lib4
proj3/clibs/lib2
```

Where, *lib1*, *lib2*, *lib3*, and *lib4* are shared object file names that need to be specified without extension.

Dynamic Loading of PLI Libraries at Runtime

You can dynamically load a PLI library at runtime instead of linking the PLI library at compile time. For this, perform the following steps:

1. Compile the design including the PLI table file for PLI libraries with the `-P` compile-time option:

```
% vcs -P pli.tab design_source_files
```

2. Load the libraries dynamically at runtime, specify the libraries with the `-load` runtime option, and enter `-load` for each library:

```
% simv -load ./pli1.so -load ./pli2.so
```

In this example, there are two `-load` options for the libraries named `pli1.so` and `pli2.so`.

Note:

If the PLI library is linked at compile time, the library has precedence over a PLI library loaded at runtime.

Key Compilation or Elaboration Features

This section describes the following features in detail with a usage model and an example:

- [Initializing Verilog Variables, Registers, and Memories](#)
- [Overriding Generics and Parameters](#)
- [Checking for x and z Values In Conditional Expressions](#)
- [Cross Module References \(XMRs\)](#)
- [Verilog Configurations and Libmaps](#)
- [Lint Warning Message for Missing 'endcelldefine'](#)
- [Error/Warning/Lint Message Control](#)
- [Extracting the Files Used in Elaboration/Compilation](#)
- [Reducing Compile Time for Post-Process Only Debug](#)

Initializing Verilog Variables, Registers, and Memories

You can use one of the following options to initialize Verilog variables, registers, and memories in a design:

- `+vcs+initreg+random`
Enables initialization for an entire design.
- `+vcs+initreg+config+config_file`
Enables initialization for selective parts of a design.

This section discusses the following topics:

- [Initializing Verilog Variables, Registers, and Memories for an Entire Design](#)
- [Initializing Verilog Variables, Registers, and Memories in Selective Parts of a Design](#)
- [Selections for Initialization of Registers or Memories](#)
- [Reporting the Initialized Values of Variables, Registers, and Memories](#)

Initializing Verilog Variables, Registers, and Memories for an Entire Design

You can use the `+vcs+initreg+random` option to initialize all bits of Verilog variables and registers defined in sequential UDPs and memories, including multi-dimensional arrays (MDAs) in your design to a random value of 0 or 1, at time zero. The default random seed is used.

The supported data types are:

- `reg`
- `bit`
- `integer`
- `int`
- `logic`

To enable initialization for an entire design, the `+vcs+initreg+random` option must be specified at compile time, and one of the following options must be specified at runtime:

- `+vcs+initreg+0`
- `+vcs+initreg+1`

- +vcs+initreg+random
- +vcs+initreg+seed_value

Example-1:

```
% vlogan [vlogan_options] file4.v file5.v file6.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

Note:

In the `vhdlan` command line, specify the bottom-most entity first, then move up in order.

```
% vcs +vcs+initreg+random [other_vcs_options] file1.v
                           file2.v file3.v

% simv +vcs+initreg+random [simv_options]
```

All Verilog variables, registers, and memories are assigned random initial values.

Example-2:

```
% vlogan [vlogan_options] file4.v file5.v file6.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
% vcs +vcs+initreg+random [other_vcs_options] file1.v   file2.v file3.v

% simv +vcs+initreg+0 [simv_options]
```

All Verilog variables, registers, and memories are assigned initial value of 0.

For more information on the `+vcs+initreg+random` compile-time option, see “Option for Initializing Verilog Variables, Registers and Memories with Random Values”.

For more information on the runtime initialization options, see “Option for Initializing Verilog Variables, Registers and Memories at Runtime”.

The initialization options may cause potential race conditions due to the initialized values specified. For more information on race condition prevention, see “Option for Initializing Verilog Variables, Registers and Memories with Random Values”.

Initializing Verilog Variables, Registers, and Memories in Selective Parts of a Design

You can use the `+vcs+initreg+config+config_file` option to specify a configuration file for initializing Verilog variables, registers defined in sequential UDPs and memories, including multi-dimensional arrays (MDAs) in your design, at time zero. In the configuration file, you can define the parts of a design to apply the initialization and the initialization values of the variables.

The supported data types are:

- reg
- bit
- integer
- int
- logic

To enable the initialization in selective parts of a design, you can specify the `+vcs +initreg+config+config_file` option at compile time and/or at runtime. The `config_file` option is the configuration file used for the initialization.

If you specify the `+vcs+initreg+config+config_file` option at both compile time and runtime, then the configuration file specified at runtime overrides the configuration file specified at compile time. Specifying the configuration file at runtime allows you to rerun the existing `simv` with the new configuration file without the need to recompile. For more information, see “Option for Initializing Verilog Variables, Registers and Memories in Selective Parts of a Design at Runtime”.

Example: Using the `+vcs+initreg+config+config_file` Option at Compile Time

```
% vlogan [vlogan_options] file4.v file5.v file6.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

Note:

In the `vhdlan` command line, specify the bottom-most entity first, then move up in order.

```
% vcs +vcs+initreg+config+test_config
[other_vcs_options] file1.v file2.v file3.v

% simv [simv_options]
```

The configuration file, `test_config`, is used for initialization.

For details on the configuration file entries, see “Option for Initializing Verilog Variables, Registers and Memories in Selective Parts of a Design”.

Selections for Initialization of Registers or Memories

When the `+vcs+initreg+random` or `+vcs+initreg+config+config_file` option is specified at compile time, you can include one of the following initialization options:

- `+vcs+initreg+random+nomem`
- `+vcs+initreg+random+noreg`

The `+vcs+initreg+random+nomem` option disables initialization of memories or multi-dimensional arrays (MDAs). This option allows initialization of variables that do not have a dimension.

Conversely, the `+vcs+initreg+random+noreg` option disables initialization of variables that do not have a dimension. This option allows initialization of memories or MDAs.

Reporting the Initialized Values of Variables, Registers, and Memories

The `VCS_PRINT_INITREG_INITIALIZATION` environment variable enables printing of all initialized variables, registers, memories, and their initialized values to the `vcs_initreg_random_value.txt` file at runtime. The following is the use model:

```
% setenv VCS_PRINT_INITREG_INITIALIZATION 1
```

The following is the format of `vcs_initreg_random_value.txt` file:

`[Hierarchical_path] = [Decimal_Value] // [Number_Of_Bits] [REG|MEM]`

Where, the `[Decimal_Value]` of 0, 1, 2 represents values of 0, 1, and z respectively. Packed arrays can be represented with decimal values greater than 2.

The following is the sample report of the `vcs_initreg_random_value.txt` file:

```
tb.d1.v1[0] = 2 // 1 MEM
tb.d1.v1[1] = 2 // 1 MEM
tb.d1.v1[2] = 2 // 1 MEM
tb.d1.v1[3] = 2 // 1 MEM
tb.d1.c = 2 // 1 REG
tb.t1.a = 0 // 1 REG
tb.t1.b = 1 // 1 REG
tb.d1.p1 = 65535 // 16 REG
```

Where 4-bit unpacked array `v1` is initialized to `z` (each element of an unpacked array is reported as `MEM`), register `c` is initialized to `z`, register `a` is initialized to 0, register `b` is initialized to 1, and 16-bit packed array `p1` is initialized to decimal value 65535.

Overriding Generics and Parameters

This section discusses the following topics:

- [Overriding Verilog Parameters](#)
- [Overriding VHDL Generics](#)
- [Usage Model](#)

Overriding Verilog Parameters

There are two compile-time options for changing parameter values from the `vcs` command line:

- `-pvalue`
- `-parameters`

You specify a parameter with the `-pvalue` option. It has the following syntax:

```
vcs -pvalue+hierarchical_name_of_parameter=value
```

For example:

```
vcs source.v -pvalue+test.d1.param1=33
```

You specify a file with the `-parameters` option. The file contains command lines for changing values. A line in the file has the following syntax:

```
assign value path_to_the_parameter
```

Here:

`assign`

Keyword that starts a line in the file.

`value`

New value of the parameter.

`path_to_the_parameter`

Hierarchical path to the parameter. This entry is similar to a Verilog hierarchical name except that you use forward slash characters (/), instead of periods, as the delimiters.

The following is an example of the contents of this file:

```
assign 33 test/d1/param1
assign 27 test/d1/param2
```

Note:

The `-parameters` and `-pvalue` options do not work with a `localparam` or a `specparam`.

Overriding VHDL Generics

There are two compile-time options for changing generic values from the `vcs` command line:

- `-gvalue`
- `-generics`

The `-gvalue` option overrides the generic value defined in the source code with the value specified in the command line.

The `-generics` option overrides the default values for the design generics by using values from the file `cmdfile`. The `cmdfile` file is an include file that contains `assign` commands targeting design generics.

VCS allows you to override both generic or parameter values in the design using the compile-time option, `-gfile cmd.txt`.

For more information on generic values, refer to the “*Options for Overriding Generics and Parameters*” section of the Appendix C, “Compilation/Elaboration Options” chapter.

Here, `cmd.txt` is an include file containing `assign` commands to override generic or parameter values. The syntax of the `assign` command is as follows:

```
assign value generics/parameters
```

Note:

You can also override generics at runtime. See, [Using Verdi](#).

Using this option, you can override any generic or parameter of the following datatypes:

- `integer`
- `real`
- `string`

You can also specify more than one generic or parameter in the same line as shown below:

```
assign 1 g1 g2
```

For example:

The usage model to override the default value of a generic "WIDTH" in your top-level VHDL file to "16", is as follows:

```
% vhdlan top.vhd mem.vhd
% vcs top -gfile gen.txt
% simv
```

The include `gen.txt` file contains the following commands:

```
% cat gen.txt
assign 16 WIDTH
```

Similarly, you can use the same assign commands to override the parameters in the Verilog modules as shown in the following example:

```
module top();
parameter filename="mem.txt"
initial
  $display("The filename is %s", filename);
endmodule
```

You can override the default value of the `filename` parameter in the above example, to `mem.txt`, as shown below:

```
% vlogan top.v
% vcs top -gfile param.txt
% simv
```

The include `param.txt` file contains the following commands:

```
% cat param.txt
assign "mem2.txt" filename
```

Usage Model

Analysis

```
% vlogan [vlogan_options] file4.v file5.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

Note:

Specify the VHDL bottommost entity first, then move up in order.

Elaboration

```
% vcs [vcs_options] top_cfg/entity/module -gfile cmd.txt
```

Simulation

```
% simv [simv_options]
```

Note:

Verilog module can have parameters where the type is not specified or an initial value is not defined. When such a module is instantiated in VHDL, these parameters are considered of type `integer`.

When VHDL does not override a parameter, either by declaring the generic as “open,” or by not mentioning generic, then the default values on the Verilog side are 0 (for integer), 0.0 (for real), and an empty string.

Any other parameter or generic assignment than an integer value, leads to an elaboration error.

For example,

```
module dut #(parameter int_int_par1 = 42,
            parameter int_int_par2,
            parameter string str_int_par1,
            parameter str_int_par2) (input clk);
  ...
endmodule

entity top; end entity top;

architecture top_arch of top is
  component dut is
    generic ( int_int_par1 : natural;
              int_int_par2 : natural;
              str_int_par1 : string;
              str_int_par2 : string)
    port (clk : in std_logic);
  end component dut
  signal clk : std_logic;
begin
  V : dut generic map (
    int_int_par1 => 1,
    int_int_par2 => 2,           - this is allowed
    str_int_par1 => "another string", - this is allowed
    str_int_par2 => "hi there")      - this leads to an
  elaboration error: integer expected
  port map ( clk => clk);
end architecture top_arch
```

Note:

For more details, refer to the *Instantiating Verilog or VHDL in Your Design* chapter of *VCS Simulation Coding and Modeling Style Guide*.

Checking for x and z Values In Conditional Expressions

The `-xzcheck` compile-time option tells VCS to display a warning message when it evaluates a conditional expression and finds it to have an x or Z value.

A conditional expression is of the following types or statements:

- A conditional or `if` statement:

```
if(conditional_exp)
    $display("conditional_exp is true");
```

- A `case` statement:

```
case(conditional_exp)
  1'b1: sig2=1;
  1'b0: sig3=1;
  1'bx: sig4=1;
  1'bz: sig5=1;
endcase
```

- A statement using the conditional operator:

```
reg1 = conditional_exp ? 1'b1 : 1'b0;
```

The following is an example of the warning message that VCS displays when it evaluates the conditional expression and finds it to have an `x` or `z` value:

```
warning 'signal_name' within scope hier_name in file_name.v: line_number
to x/z at time simulation_time
```

VCS displays this warning every time it evaluates the conditional expression to have an `x` or `z` value, not just when the signal or signals in the expression transition to an `x` or `z` value.

VCS does not display a warning message when a sub-expression has the value `x` or `z`, but the conditional expression evaluates to `1` or `0` value. For example:

```
r1 = 1'bz;
r2 = 1'b1;
if ( (r1 && r2) || 1'b1)
    r3 = 1;
```

In this example, the conditional expression always evaluates to a value of `1`. Therefore, VCS does not display a warning message.

This section discusses the following topics:

- [Enabling the Checking](#)
- [Filtering Out False Negatives](#)

Enabling the Checking

The `-xzcheck` compile-time option globally checks all the conditional expressions in the design and displays a warning message every time it evaluates a conditional expression to have an `x` or `z` value. You can suppress or enable these warning messages

on selected modules using `$xzcheckoff` and `$xzcheckon` system tasks. For more details on `$xzcheckoff` and `$xzcheckon` system tasks, see “[Checking for X and Z Values in Conditional Expressions](#)”.

The `-xzcheck` compile-time option has an optional argument to suppress the warning for glitches evaluating to `x` or `z` value. Synopsys calls these glitches as false negatives. See [Filtering Out False Negatives](#).

Filtering Out False Negatives

By default, if a signal in a conditional expression transitions to an `x` or `z` value and then to `0` or `1` in the same simulation time step, VCS displays the warning.

Example 1

In this example, VCS displays the warning message when `reg r1` transitions from `0` to `x` to `1` during simulation time `1`.

Example 5 False Negative Example

```
module test;
  reg r1;

  initial
    begin
      r1=1'b0;
      #1 r1=1'bx;

      #0 r1=1'b1;
    end

  always @ (r1)
  begin
    if (r1)
      $display("\n r1 true at %0t\n",$time);
    else
      $display("\n r1 false at %0t\n",$time);
  end
endmodule
```

Example 2

In this example, VCS displays the warning message when `reg r1` transitions from `1` to `x` during simulation time `1`.

Example 6 False Negative Example

```
module test;
  reg r1;

  initial
    begin
      r1=1'b0;
```

```
#1 r1<=1'b1;
r1=1'bx;
end
always @ (r1)
begin
if (r1)
  $display("\n r1 true at %0t\n",$time);
else
  $display("\n r1 false at %0t\n",$time);
end

endmodule
```

If you consider these warning messages to be false negatives, use the `nofalseneg` argument to the `-xzcheck` option to suppress the messages.

For example:

```
% vcs -xzcheck nofalseneg example.v
```

If you compile and simulate Example1 or Example2 with the `-xzcheck` elaboration/compilation option, but without the `nofalseneg` argument, VCS displays the following warning about signal `r1` transitioning to `x` or `z` value:

```
r1 false at 0
Warning: 'r1' within scope test in source.v: 13 goes to x/z at time 1

r1 false at 1

r1 true at 1
```

If you compile and simulate the examples shown earlier in this chapter, Example 1 or Example 2, with the `-xzcheck` elaboration/compilation option and the `nofalseneg` argument, VCS does not display the warning message.

Cross Module References (XMRs)

Verilog enables you to access any internal signal from any other hierarchical block without having to route it through a user interface.

VHDL does not have the language support to allow you to access internal signals from any other hierarchical block. Therefore, it is not possible to either assign or test the value of a signal deep in the design hierarchy without defining it in a global package, and then referencing it in a hierarchical block where it is used.

The `hdl_xmr` procedure (in VHDL code) and the `$hdl_xmr` system task enables you to access internal signals in a mixed HDL design and Verilog only. Therefore, you can handle the signals in the VHDL database. In a mixed HDL or Verilog only environment, you can access VHDL or Verilog signals across language boundaries using this feature.

The `hdl_xmr` procedure and the `$hdl_xmr` system task work only when the source and destination objects match in both type and size.

This section discusses the following topics:

- [The `hdl_xmr` Procedure and the `\$hdl_xmr` System Task](#)
- [Data Types Supported](#)
- [Using the `hdl_xmr` Procedure](#)
- [Using the `\$hdl_xmr` Task](#)
- [Use Model](#)
- [Examples](#)
- [\\$hdl_xmr Support for VHDL Variables](#)
- [Data Type Support and Usage Examples](#)
- [Support for Native XMR force and release](#)

The `hdl_xmr` Procedure and the `$hdl_xmr` System Task

The `hdl_xmr` procedure and the `$hdl_xmr` system task creates a permanent bond between the two objects, called source and destination. Each time an event occurs on the source object, the destination object is assigned a new value of the source object. Note that if the destination object has other sources, like an assignment statement, the last event value (from `hdl_xmr/$hdl_xmr` or the `assignment` statement) is assigned to the destination object, thus, overwriting the previous value.

When the `hdl_xmr` procedure or the `$hdl_xmr` system task is executed, source and destination objects are bound together until the end of the simulation. Therefore, it is important that `hdl_xmr/$hdl_xmr` calls are specified in the code only once.

Note:

- All these following delimiters are supported. "/", ".", ":" except for a pure VHDL design where you cannot use "." as a delimiter.
- For mixed HDL designs, you must use the `-debug_access` option for the `$hdl_xmr` system task to work.

Data Types Supported

`hdl_xmr` and `$hdl_xmr` supports the following data types:

- Scalars, vectors, bit-selects, and part-selects (slices) are supported for both source and destination objects. Global VHDL signals are also supported.
- The following types of VHDL signals are supported with their corresponding Verilog types:
 - `integer`
 - `bit` and `bit vector`
 - `string`
 - `std_logic/std_ulogic/std_logic_vector/std_ulogic_vector`
 - Enumerated datatypes

In case of an integer type, a Verilog type of size 32, for example, `reg[31:0]`, is allowed as a matching type. Similarly, for a packed struct `std_logic_vector/std_ulogic_vector` is allowed as a matching type.

- The following SystemVerilog datatypes are supported across VCS boundary:
`shortint`, `int`, `longint`, `byte`, `bit`, `logic`, and `reg`.

The following table lists the supported SystemVerilog datatypes with their matching VHDL datatypes.

Table 7 SystemVerilog Datatypes With Their Matching VHDL Datatypes

SystemVerilog Data Types	Integer	Integer Subtype	Bit vector	std_logic vector	std_ulogic vector
Shortint	No	No	Yes	Yes	Yes
Int	Yes	Yes	Yes	Yes	Yes
Longint	No	No	Yes	Yes	Yes
Bit array	Yes	Yes	Yes	Yes	Yes
Logic array	Yes	Yes	Yes	Yes	Yes
Integer	Yes	Yes	Yes	Yes	Yes

Using the hdl_xmr Procedure

The syntax of the `hdl_xmr` procedure is as follows

```
hdl_xmr("source_object", "destination_object", [verbosity]);
```

`source_object`

`source_object` can be a VHDL signal or Verilog register or net. An absolute path or a relative path to the object can be specified.

Use an absolute path instead of a relative path, if the source node resides in VHDL part of the code or if the hierarchical path has a VHDL layer.

`destination_object`

`destination_object` could be a VHDL signal or a Verilog register. An absolute path or a relative path to the object can be specified.

Note:

Use an absolute path instead of a relative path, if the hierarchical path contains a VHDL layer. Verilog net type as a destination object is not supported.

`verbosity`

Third optional argument to the `hdl_xmr` call is a verbosity index. If the argument is not specified then the default value is '0', otherwise possible integer values are '0' or '1'. Value '0' indicates no verbosity, and value '1' enables verbosity. If you specify '1', then every time a value of the source object is copied onto the destination object, a message is displayed.

To use the `hdl_xmr` procedure, you should include the XMR package in your VHDL source code as shown below:

```
Library Synopsys;
Use Synopsys.hdl_xmr_pkg.all;
```

You can call the `hdl_xmr` procedure concurrently or within a process having no sensitivity list and a wait, at the end of the process block, as shown in the following example:

```
hdl_xmr(":vh:vl:cout0", ":vh:coutin_xmr");
hdl_xmr("/vh/vl/cout0", "/vh/in[3]", 1);
```

When there is an escaped/extended-identifier instance present in the source and if it is required to be specified, the hierarchical path must be enclosed within “\” when a call is made.

For example, when a signal is inside an escaped instance and it is referred to as “/VH/VL/\U_VL/U_ESC /signal1”, the same signal inside the `hdl_xmr` procedure must be referred to as “/VH/VL/\U_VL/U_ESC\/signal1”, essentially the hierarchical instance

path is enclosed with “\”. You can use the `hdl_xmr` procedure as shown in the following example:

```
hdl_xmr("/VH/VL/\U_VL/U_ESC\/signal1", "/VH	signal2", 1);
```

Similarly, if there is an escaped instance, `\U_NETLIST/NETLIST_TOP` with the `NETLSIT_TOP/U_AND_2.Z` signal, the same signal inside the `hdl_xmr` procedure must be referred to as:

```
hdl_xmr("/TBTOP/U_TOP/U_NETLIST/\U_NETLIST/NETLIST_TOP\NETLIST_TOP/U_AND_2.Z", "/TBTOP/CLK", 1);
```

Using hdl_xmr With Generate Blocks

You can use the following syntax for names within generate blocks:

- For generate block in Verilog, the name for instances within generate block that has escaped name is “`escaped_generate_name.instance_name`”
- For generate block in VHDL, the name for instances within generate block that has extended name is “`extended_generate_name.instance_name`”

Using the \$hdl_xmr Task

The syntax of the `$hdl_xmr` procedure is as follows

```
$hdl_xmr("source_object" , "destination_object",
          [verbosity]);
```

`source_object`

`source_object` could be a VHDL signal or Verilog register or net. An absolute path or a relative path to the object can be specified.

Use an absolute path instead of a relative path, if the source node resides in VHDL part of the code or if the hierarchical path has a VHDL layer.

`destination_object`

`destination_object` could be a VHDL signal or a Verilog register. An absolute path or a relative path to the object can be specified.

Note:

Use an absolute path instead of a relative path, if the hierarchical path contains a VHDL layer. Verilog net type as a destination object is not supported.

`verbosity`

Third optional argument to the `hdl_xmr` call is a verbosity index. If the argument is not specified then the default value is '0', otherwise possible integer values are '0' or '1'. Value '0' indicates no verbosity. When verbosity is required, that is, '1' is the third argument, then

every time when the value of the source object is copied on to the destination object, a message is displayed.

You can use the `$hdl_xmr` system task as shown in the following example:

```
initial begin
$hdl_xmr("vl.vh.clk", "vl.vclk");
$hdl_xmr("/vl/vh/reset_n", "/vl/vrst_n[0]", 0);
$hdl_xmr("vl:vh:state[3:0]", "vl:state[4:7]", 1);
end
```

When there is an escaped/extended-identifier instance present in the source and if it is required to be specified, the hierarchical path should be enclosed within “\” when a call is made.

For example, when a signal is inside an escaped instance and it is referred to as "VL.\U_VL/U_ESC .VH.Signal1", the same signal inside the `hdl_xmr` task must be referred to as "VL.\U_VL/U_ESC\VH.Signal1", essentially the hierarchical instance path is enclosed with "\". You can use the `$hdl_xmr` system task as shown in the following example:

```
$hdl_xmr("VL.\U_VL/U_ESC\VH.Signal1", "VL.signal2", 1);
```

Similarly, if there is an escaped instance \U_NETLIST/NETLIST_TOP with the NETLSIT_TOP.U_AND_2.Z signal, the same signal inside the `$hdl_xmr` task is referred to as:

```
$hdl_xmr("top.U_NETLIST.\U_NETLIST/NETLIST_TOP\
.NETLIST_TOP.U_AND_2.Z", "top.CLK", 1);
```

Using `$hdl_xmr` With Generate Blocks

You can use the following syntax for names within generate blocks:

- For generate block in Verilog, the name for instances within generate block that has escaped name is “\escaped_generate_name.instance_name\\”
- For generate block in VHDL, the name for instances within generate block that has extended name is “\extended_generate_name\.instance_name”

Use Model

Analysis

```
% vlogan [vlogan_options] file4.v file5.v file6.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

Note:

Specify the VHDL bottom most entity first, then move up in order.

Elaboration

```
% vcs [vcs_options] -debug_access+r+cbk top_cfg/entity/module
```

Simulation

```
% simv [simv_options]
```

Examples

- `hdl_xmr` with escaped/extended identifiers

Consider the following design code — *test.v* and *test.vhdl* that are cross referenced using escaped/extended identifiers.

test.v

```
cat test.v
`timescale 1ns/1ns
module top;
reg clk;
middle `m_inst/m_inst ();
initial
begin
$hdl_xmr ("top.clk",
"top.\`m_inst/m_inst\`.leaf_inst.\`I1/I1\`.vhtop", 1);
$hdl_xmr ("top.clk",
"/top/\`m_inst/m_inst\`/leaf_inst/\`I1/I1\`/vhtop", 1);
clk = 0;
#10 clk = 1 ;
#10 $finish;
end

endmodule

module middle ;
leaf leaf_inst();
endmodule
```

test.vhdl

```
library ieee;
use ieee.std_logic_1164.all ;

entity vhleaf is

end vhleaf ;

architecture a of vhleaf is

signal vhtop : std_logic ;
begin
```

```

end a ;

library ieee;
use ieee.std_logic_1164.all ;
library synopsys;
use synopsys.hdl_xmr_pkg.all ;

entity leaf is

end leaf ;

architecture a of leaf is

component vhleaf
end component ;

begin

  hdl_xmr ("top.clk",
    "top.\m_inst/m_inst.leaf_inst.\I1/I1\.vhtop", 1);
  hdl_xmr ("top.clk",
    "/top/\m_inst/m_inst/leaf_inst/\I1/I1\/.vhtop", 1);
  \I1/I1\ : vhleaf
end a ;

```

If the call ids are made from Verilog, both escaped and extended identifier must be enclosed within “\”. Whereas, if the call is made from VHDL, both escaped and extended identifier must be enclosed within “\\”. The `hdl_xmr` procedures for this example are defined as shown in the following table:

	Call From	Format	hdl_xmr Procedures and Tasks
Escape identifier in Verilog - \m_inst/m_inst; Extended identifier in VHDL - \I1/I1\ : vhleaf;	Verilog	dot	\$hdl_xmr ("top.clk", "top.\m_inst/m_inst\\.leaf_inst.\I1/I1\.vhtop", 1);
		slash	\$hdl_xmr ("top.clk", "/top/\m_inst/m_inst\\/leaf_inst/\I1/I1\\/.vhtop", 1);
Escape identifier in Verilog - \m_inst/m_inst; Extended identifier in VHDL - \I1/I1\ : vhleaf;	VHDL	dot	hdl_xmr ("top.clk", "top.\m_inst/m_inst\\.leaf_inst.\I1/I1\.vhtop", 1);
		slash	hdl_xmr ("top.clk", "/top/\m_inst/m_inst\\/leaf_inst/\I1/I1\\/.vhtop", 1);

- hdl_xmr with generate blocks

Consider the following design code— *test.v* and *test.vhdl* that are cross referenced using generate blocks.

test.v

```
`timescale 1ns/1ns
module top;
reg clk;
middle m_inst ();
initial
begin
$hdl_xmr ("top.clk",
"top.m_inst.\gen[0].leaf_inst\\.\GEN[0]\\I1.vtop", 1);
$hdl_xmr ("top.clk",
"/top/m_inst/ \gen[0].leaf_inst\\/\ \GEN[0]\\I1/vtop", 1);

clk = 0;
#10 clk = 1 ;
#10 $finish;
end
endmodule

module middle ;
generate
begin : \gen[0]
leaf leaf_inst();
end
endgenerate
endmodule
```

test.vhdl

```
library ieee;
use ieee.std_logic_1164.all ;

entity vhleaf is
end vhleaf;

architecture a of vhleaf is
signal vhtop : std_logic ;
begin
end a ;

library ieee;
use ieee.std_logic_1164.all ;
library synopsys;
use synopsys.hdl_xmr_pkg.all ;
```

```

entity leaf is
end leaf ;

architecture a of leaf is
component vhleaf
end component ;

begin
  hdl_xmr ("top.clk",
    "top.m_inst.\gen[0].leaf_inst\\.GEN[0]\\I1.vtop", 1);
  hdl_xmr ("top.clk",
    "/top/m_inst/\gen[0].leaf_inst//GEN[0]/I1/vtop", 1);

\GEN[0]\ : if ( true ) generate
  I1 : vhleaf ;
end generate \GEN[0]\ ;
end a ;
  
```

The `hdl_xmr` tasks for this example are defined as shown in the following table:

	Call From	Format	hdl_xmr tasks
Generate instance in Verilog - <code>generate begin : \gen[0] leaf leaf_inst(); end endgenerate</code> Generate instance in VHDL- <code>\GEN[0]\ : if (true) generate I1 : vhleaf ; end generate \GEN[0]\ ;</code>	Verilog	dot	\$hdl_xmr ("top.clk", "top.m_inst.\gen[0].leaf_inst\\.GEN[0]\\I1.vtop", 1);
		slash	\$hdl_xmr ("top.clk", "/top/m_inst/\gen[0].leaf_inst//GEN[0]/I1/vtop", 1);

The `hdl_xmr` procedures for this example are shown in the following table:

	Call From	Format	hdl_xmr Procedures
Generate instance in Verilog - <code>generate begin : \gen[0] leaf leaf_inst(); end endgenerate</code> Generate instance in VHDL - <code>\GEN[0]\ : if (true) generate I1 : vhleaf ; end generate \GEN[0]\ ;</code>	VHDL	dot	hdl_xmr ("top.clk", "top.m_inst.\gen[0].leaf_inst\\.GEN[0]\\I1.vtop", 1);
		slash	hdl_xmr ("top.clk", "/top/m_inst/\gen[0].leaf_inst//GEN[0]/I1/vtop", 1);

\$hdl_xmr Support for VHDL Variables

VCS supports the usage of VHDL objects of type, variable, in the \$hdl_xmr system task. This support enables you to use VHDL variables, as source or destination, in the \$hdl_xmr (not hdl_xmr in VHDL side) call.

Use Model

In Verilog source, you should call \$hdl_xmr as:

```
$hdl_xmr (<"source variable">, <"destination signal">, <verbosity_value>)
$hdl_xmr (<"source signal">, <"destination variable">, <verbosity_value>)
```

You can specify the source variable and the destination variable in a relative or an absolute path. The last integer value, *verbosity_value*, is optional. It is only used for verbosity. The variable object is the VHDL object.

To enable the support for \$hdl_xmr with VHDL variables, you must use one of the following compile-time options:

- vcs <top> -debug_access* -vdbg_watch
- vcs <top> -debug_access+all

Note:

- In VHDL variables, you must pass the `-vdbg_watch` option along with the `-debug_access*` option. If you are using the `-debug_access+all` option, then there is no need to pass the `-vdbg_watch` option.
- For mixed HDL designs, you must use the `-debug_access*` option for the \$hdl_xmr system task to work.

Data Type Support and Usage Examples

The following table shows data type support and usage examples:

Table 8 Data Type Support and Usage Examples

Verilog Data Types	VHDL Data Types for Variable
reg	bit/std_logic/std_ulogicVHDL record elements. Datatypes for record elements can be bit/std_logic/std_ulogic.

Table 8 Data Type Support and Usage Examples (Continued)

Verilog Data Types	VHDL Data Types for Variable
<pre>module tb;reg r1,r2; reg [0:3] r3,r4;leaf inst1();initial begin \$hdl_xmr("inst1.r1","r1",1); \$hdl_xmr("r2",inst1.r2",1); \$hdl_xmr("inst1.r1","r3[1:1]",1); \$hdl_xmr("r4[1:1]",inst1.r2",1); \$hdl_xmr("inst1.rec.r1","r1",1); \$hdl_xmr("r2",inst1.rec.r2",1); \$hdl_xmr("inst1.rec.r1","r3[1:1]",1); \$hdl_xmr("r4[1:1]",inst1.rec.r2",1); end endmodule</pre>	<pre>entity leaf is end leaf; architecture beh of leaf istype pkt is recordr1 : bit; r2 : std_logic;end record; shared variable rec : pkt ;shared variable r1 : std_logic ;shared variable r2 : std_ulogic ; begin end;</pre>
reg vector	bit_vector/std_logic_vector/signed/unsigned/integer/natural VHDL record elements. Datatypes for record elements can be bit_vector/ std_logic_vector/signed/ unsigned/integer/natural
<pre>module tb;reg [31:0] r1,r2,r3,r4; leaf inst1();initial begin \$hdl_xmr("inst1.r1","r1",1); \$hdl_xmr("r2",inst1.r2",1); \$hdl_xmr("inst1.r1[15:0]","r3[31:16]",1); \$hdl_xmr("r4[15:0]",inst1.r2[15:0"],1); \$hdl_xmr("inst1.rec.r1","r1",1); \$hdl_xmr("r2",inst1.rec.r2",1); \$hdl_xmr("r4[3:0]",inst1.rec.r2[3:0"],1); end endmodule</pre>	<pre>entity leaf is end leaf; architecture beh of leaf istype pkt is recordr1 : natural; r2 : std_logic_vector(31 downto 0);end record; shared variable rec : pkt;shared variable r1,r2 : std_logic_vector(31 downto 0): begin end;</pre>
reg mda	vhdl mda. Base data type for array elements can be bit/std_logic/std_ulogic/bit_vector/std_logic_vector/integer/natural
<pre>module tb;reg [31:0] r1,r2,r3 [0:7] reg [31:0] r4; leaf inst1();initial begin \$hdl_xmr("inst1.r1","r1",1); \$hdl_xmr("r2",inst1.r2",1); \$hdl_xmr("inst1.r3","r3",1); \$hdl_xmr("r4",inst1.r2[1"],1); \$hdl_xmr("inst1.r1[2]","r4",1); \$hdl_xmr("r2[2]",inst1.r2[2"],1); end endmodule</pre>	<pre>entity leaf is end leaf; architecture beh of leaf istype ram is array(0 to 7) of std_logic_vector(31 downto 0);type ram1 is array(0 to 7) of bit_vector(31 downto 0);type ram2 is array(0 to 7) of natural;shared variable r1 : ram;shared variable r2 : ram1;shared variable r3 : ram2; begin end;</pre>
realreal real mda Note: Verilog real vectors are not supported.	vhdl realreal field of vhdl record real mda

Table 8 Data Type Support and Usage Examples (Continued)

Verilog Data Types	VHDL Data Types for Variable
<pre>module tb;real r1 [0:7];real r2; leaf inst1();initial begin \$hdl_xmr("inst1.r1","r1",1); \$hdl_xmr("r2",inst1.r2",1); \$hdl_xmr("r2",inst1.r1[1"],1); \$hdl_xmr("inst1.r1[1]", "r2",1); end endmodule</pre>	<pre>entity leaf is end leaf; architecture beh of leaf istype ram is array(0 to 7) of real;shared variable r1 : ram;shared variable r2 : real;begin end;</pre>
packed struct array of packed struct Data types for elements of packed struct : reg/logicreg/logic vector real	<pre>vhdl record array of vhdl recordsData types for elements of vhdl record:bit/std_logic/std_ulegicbit_vector/std_[u]logic_vector/signed/unsigned/natural/integerreal</pre>
<pre>module tb; typedef struct packed {reg [31:0] t ; reg [15:0] b;} st;st r1,r2;st r3 [0:1];leaf inst1(); initial begin \$hdl_xmr("r2","inst1.r2",1);\$hdl_xmr("inst1.r1 ","r1",1);\$hdl_xmr("inst1.r3","r3",1);\$hdl_xmr ("inst1.r3[1]", "r3[1]",1);\$hdl_xmr("inst1.r3[0] ","r1",1);\$hdl_xmr("r2","inst1.r3[1"]);end endmodule</pre>	<pre>entity leaf is end leaf; architecture beh of leaf is type rec is record a1 : integer ;a2 : bit_vector(15 downto 0);end record; shared variable r1,r2 : rec; type arr is array(0 to 1) of rec;shared variable r3 : arr;begin end beh;</pre>

Support for Native XMR force and release

VCS supports `force` and `release` on VHDL objects with native Verilog XMRs without debug capabilities by using the `-mx_force` option at runtime.

Use Model

Use the following option at runtime:

```
% simv -mx_force
```

Example

The following example demonstrates the use of `force` and `release`. The new supported values are highlighted in the example.

Example 7 Usage of force and release

The content of the `test.v` file is as follows:

```
`timescale 1ns/1ns
module top;

reg clk, rst;
wire [3:0] count;

dut inst (,,count);
```

```

always
begin
  force top.inst.clk = 1'b0;
#5;
  force top.inst.clk = 1'b1;
#5;
end
initial
begin
  force top.inst.rst = 1'b1;
#3;
  release top.inst.rst;
#1;
  force top.inst.rst = 1'b0;
#155;
$finish;
end
initial $monitor("time = %0t rst = %b count
  =%0d",$time,top.inst.rst,count);
endmodule

```

The content of the test.vhd file is as follows:

```

library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_unsigned.all;

entity dut is
  port(
    clk : in std_logic;
    rst : in std_logic;
    count : out std_logic_vector(3 downto 0)
  );
end entity dut;

architecture arch of dut is
begin
  process(clk,rst)
    variable temp_count : std_logic_vector( 3 downto 0);
  begin
    if(rst='1') then
      temp_count := (others => '0');
    elsif(clk'event and clk='1') then
      temp_count := temp_count + 1;
    end if;
    count <= temp_count;
  end process;
end arch;

```

Run the example using the following commands:

- % vhdlan -full64 test.vhd
- % vlogan -full64 test.v
- % vcs -full64 top
- % simv -mx_force

The output is as follows:

```
time = 0 rst = 1 count =0
time = 3 rst = z count =0
time = 4 rst = 0 count =0
time = 5 rst = 0 count =1
time = 15 rst = 0 count =2
time = 25 rst = 0 count =3
time = 35 rst = 0 count =4
time = 45 rst = 0 count =5
time = 55 rst = 0 count =6
time = 65 rst = 0 count =7
time = 75 rst = 0 count =8
time = 85 rst = 0 count =9
time = 95 rst = 0 count =10
time = 105 rst = 0 count =11
time = 115 rst = 0 count =12
time = 125 rst = 0 count =13
time = 135 rst = 0 count =14
time = 145 rst = 0 count =15
time = 155 rst = 0 count =0
```

Limitations

- This feature is only supported on **-full64 mode**.

The following forms of XMRs are not supported:

- XMRs to VHDL array elements, bits of a bit-vector and bit-slice of a bit-vector. For example,

VHDL:

```
type vecarr is array (0 to 2) of std_logic_vector(3 downto 0);
signal vecsigarr1 : vecarr := ( ("0000"), ("1111"), ("ZX01") );
```

Verilog:

```
force top.VHDL.vecsigarr1[1] = 4'b0011;
```

VHDL::

```
signal sig1 : std_logic_vector(3 downto 0) := "0011";
```

Verilog::

```
force top.VHDL.sig1[1] = 1'b0;
force top.VHDL.sig1[2:1] = 2'bx1;
```

- XMRs to the VHDL user-defined types. For example,

VHDL:

```
library ieee;
use ieee.fixed_pkg.all;
signal a : ufixed(3 downto -3);
```

Verilog::

```
force top.VHDL.a = 3.14;
```

- XMRs to the VHDL aliased signals. For example,

VHDL:

```
type real_vector is array (2 downto 0) of real;
type slogic_vector is array(1 downto 0) of std_logic;
signal r1 : real_vector;
signal real1 : real;
signal temp : slogic_vector;
alias my_real : real is r1(1);
alias my_real1 : real is real1;
alias r2 : std_logic is temp(1);
```

Verilog:

```
force top.VHDL.my_real = "3.14";
force top.VHDL.my_real1 = "42.42";
force top.VHDL.r2 = 1'bz;
```

VHDL XMRs cannot be used in the following scenarios:

- Assign/continuous assign. For example,

Verilog:

```
reg [3:0] local_sig1;
top.VHDL.sig1 = 4'b0011;
assign TOP.VHDL.sig2 = local_sig1;
```

- XMRs used for reading and writing. For example,

Verilog:

```
force top.VHDL.sig1 = 4'b01zx;
$display($time, " sig1: %b", top.VHDL.sig1);
```

- XMRs to VHDL unconnected ports used for both reading and writing. For example,

VHDL:

```
entity dut is port (clk : in std_logic; count: out std_logic_vector(3
downto 0)); end entity dut;
```

Verilog:

```
wire [3:0] count;
dut VHDL(, count); // clk port is open
Initial begin
  force top.VHDL.clk = 1'b1;
  #3 release top.VHDL.clk;
end
initial $monitor($time, " clk: %b", top.VHDL.clk);
```

Verilog Configurations and Libmaps

Library mapping files are an alternative to the de facto standard way of specifying Verilog library directories and files with the `-v`, `-y`, and `+libext+ext` compile-time/analysis options and the `'uselib` compiler directive.

Configurations use the contents of library mapping files to specify what source code to use to resolve instances in other parts of your source code.

Library mapping and configurations are described in *SystemVerilog LRM IEEE Std. 1800-2012*. It specifies that SystemVerilog interfaces can be assigned to logical libraries.

This section discusses the following topics:

- [Library Mapping Files](#)
- [Configurations](#)
- [Hierarchical Configurations](#)
- [The -top Compile-Time Option](#)
- [Limitations of Configurations](#)
- [Use Model](#)
- [Example](#)
- [Using the -liblist Option](#)
- [Design Cells and Library Cells](#)
- [Library Search Order Rules](#)

- [Example Testcase Files](#)
- [Usage Examples for Library Search Order Rules for Verilog or SystemVerilog Designs](#)

Library Mapping Files

A library mapping file enables you to specify logical libraries and assign source files to these libraries. You can specify one or more logical libraries in the library mapping file. If you specify more than one logical library, you are also specifying the search order VCS uses to resolve instances in your design.

The following is an example of the contents of a library mapping file:

```
library lib1 /net/design1/design1_1/*.v;
library lib2 /net/design1/design1_2/*.v;
```

Note:

Path names can be absolute or relative to the current directory that contains the library mapping file.

In this example of the library mapping file, there are two logical libraries. VCS searches the source code assigned to *lib1* first to resolve module instances (or user-defined primitive or SystemVerilog interface instances) because that logical library is listed first in the library mapping file.

When you use a library mapping file, source files that are not assigned to a logical library in this file are assigned to the default logical library named *work*.

You specify the library mapping file with the `-libmap` during compilation/analysis.

Resolving ‘include Compiler Directives

The source file in a logical library might include the `'include` compiler directive. If so, you can include the `-inmdir` option on the line in the library mapping file that declares the logical library, for example:

```
library gatelib /net/design1/gatelib/*.v -inmdir /net/
design1/specllib, /net/design1/spec2lib;
```

Note:

The `-inmdir` option specified in the library mapping file overrides the `+inmdir` option specified in the VCS command line.

Configurations

Verilog 2001 configurations are sets of rules that specify what source code is used for particular instances.

Verilog 2001 introduces the concept of configurations and it also introduces the concept of cells. A cell is like a VHDL design unit. A module definition is a type of cell, as it is a user-

defined primitive. Similarly, a configuration is also a cell. A SystemVerilog interface and testbench program block are also types of cells.

Configurations provide the following functionalities:

- Specifies a library search order for resolving cell instances (as does a library mapping file).
- Specifies overrides to the logical library search order for the specified instances.
- Specifies overrides to the logical library search order for all instances of the specified cells.

You can define a configuration in a library mapping file or in any type of Verilog source file outside the module definition as shown in the [Example](#).

Configurations can be mapped to a logical library like any other type of cell.

Configuration Syntax

A configuration contains the following statements:

```
config config_identifier;
design [library_identifier.]cell_identifier;
config_rule_statement;
endconfig
```

where,

`config`

A keyword that begins a configuration.

`config_identifier`

A name you enter for the configuration.

`design`

A keyword that starts a `design` statement for specifying the top of the design.

`[library_identifier.]cell_identifier;`

Specifies the top-level module (or top-level modules) in the design and the logical library for this module (modules).

`config_rule_statement`

Zero, one, or more of the following clauses: `default`, `instance`, or `cell`.

`endconfig`

A keyword that ends a configuration.

The default Clause

The `default` clause specifies logical libraries in which to search to resolve a default cell instance. A default cell instance is an instance in the design that is not specified in a subsequent `instance` or `cell` clause in the configuration.

You specify these libraries with the `liblist` keyword. The following is an example of a `default` clause:

```
default liblist lib1 lib2;
```

This `default` clause specifies resolving default instances in the logical libraries names *lib1* and *lib2*.

Note:

- Do not enter a comma (,) between logical libraries.
- The default logical library work, if not listed in the list of logical libraries, is appended to the list of logical libraries and VCS searches the source files in work last.

The instance Clause

The `instance` clause specifies details about a specific instance. These details depend on the use of the `liblist` or `use` keywords:

`liblist`

Specifies the logical libraries to search to resolve the instance.

`use`

Specifies that the instance is an instance of the specified cell in the specified logical library.

The following are examples of the `instance` clause:

```
instance top.dev1 liblist lib1 lib2;
```

This `instance` clause tells VCS to resolve instance *top.dev1* with the cells assigned to logical libraries *lib1* and *lib2*;

```
instance top.dev1.gml use lib2.gizmult;
```

This `instance` clause tells VCS that *top.dev1.gml* is an instance of the cell named *gizmult* in the logical library *lib2*.

The cell Clause

The `cell` clause is similar to the `instance` clause except that it specifies details about all instances of a cell definition instead of specifying details about a particular instance. These details depend on the use of the `liblist` or `use` keywords:

`liblist`

Specifies the logical libraries to search to resolve all instances of the cell.

`use`

The specified cell's definition is in the specified library.

Hierarchical Configurations

A design can have more than one configuration. You can, for example, define a configuration that specifies the source code you use in particular instances in a subhierarchy, then you can define a configuration for a higher level of the design.

Suppose, for example, a subhierarchy of a design was an eight-bit adder and you have RTL Verilog code describing the adder in a logical library named `rtllib` and you have gate-level code describing the adder in a logical library named `gatelib`. If, for example, you wanted the gate-level code used for the 0 (zero) bit of the adder and the RTL level code used for the other seven bits, the configuration might appear as:

```
config cfg1;
design aLib.eight_adder;
default liblist rtllib;
instance adder.fulladd0 liblist gatelib;
endconfig
```

Now, if you instantiate this eight-bit adder eight times to make a 64-bit adder, you would use configuration `cfg1` for the first instance of the eight-bit adder, but not in any other instance. A configuration that performs this function is as follows:

```
config cfg2;
design bLib.64_adder;
default liblist bLib;
instance top.64add0 use work.cfg1:config;
endconfig
```

The -top Compile-Time Option

VCS has the `-top` compile-time option for specifying the configuration that describes the top-level configuration or module of the design. For example:

```
vcs -top top_cfg ...
```

If you have coded your design to have more than one top-level module, you can enter more than one `-top` option, or you can append arguments to the option using the plus delimiter. For example:

```
-top top_cfg+test+
```

Using the `-top` option tells VCS not to create extraneous top-level modules, that is, one that you do not specify.

Limitations of Configurations

In the current implementation, Verilog configurations have the following limitations:

- You cannot specify source code for user-defined primitives in a configuration.
- The VPI functionality, described in Section 33.7 “Displaying library binding information” in the *SystemVerilog LRM IEEE Std. 1800-2012*, is not implemented.

Use Model

Analysis

```
% vlogan -libmap libmap.v [vlogan_options] file1.v \
file2.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

Note:

Specify the VHDL bottom-most entity first, then move up in order.

Elaboration

```
% vcs [vcs_options] top_cfg/entity/config
```

Simulation

```
% simv [sim_options]
```

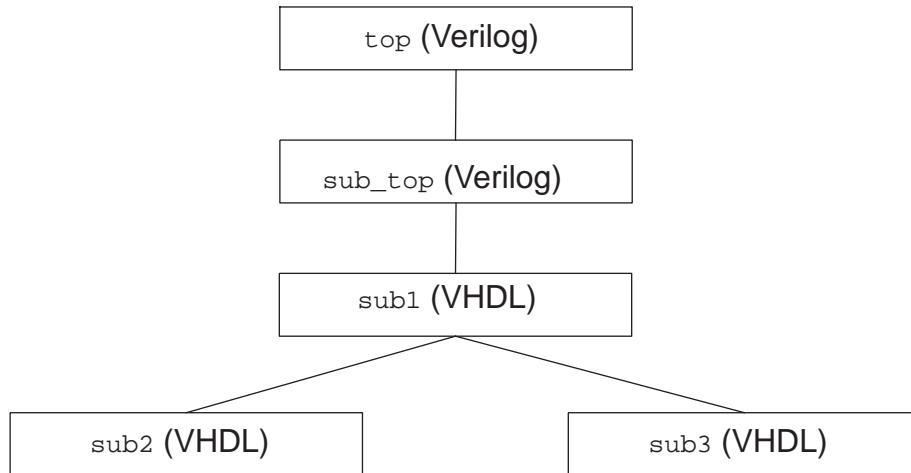
Example

A design can have more than one configuration. You can, for example, define a configuration that specifies the source code you use in a particular instance of a subhierarchy, then you can define a configuration for a higher level of the design.

For example, you have a design with VHDL-top design with the top entity as `top` instantiating a Verilog-top module, `sub_top`. This Verilog module, `sub_top`, further

instantiates a VHDL entity, `sub1` and the VHDL entity, `sub1`, instantiates VHDL entities, `sub2` and `sub3` as shown below:

Figure 2 Verilog Module and VHDL Entity Instantiating VHDL Entries



Suppose, you have the Verilog version of the entities, `sub1` and `sub2`, and need to compile and simulate the design with the Verilog version of `sub1` and the VHDL version of `sub2`. You can achieve this by defining configuration blocks in the Verilog source file outside the module definition or in a separate file as shown below:

To bind the Verilog version of `sub1`, define a configuration block in `top.v` (outside the module definition) as shown below:

```

//---top.v---
Module sub_top (...);
  u_sub1 sub1 (...;
endmodule

config top_cfg;
  design work.top;
  instance top.u_sub1 use work.sub1_cfg:config
endconfig
  
```

or in a separate file as shown below:

```

config top_cfg;
  design work.top;
  instance top.u_sub1 use work.sub1_cfg:config
endconfig
  
```

To bind the VHDL version of "sub2", define a configuration block in sub1.v (outside the module definition) as shown below:

```
//---sub1.v---
Module sub1(...);
  u_sub2 sub2 (...);
  u_sub3 sub3 (...);
endmodule

config sub1_cfg;
  design work.sub1;
  instance sub1.u_sub2 use work.CFG_SUB2_BEH:config
endconfig
```

or in a separate file as shown below:

```
config sub1_cfg;
  design work.sub1;
  instance sub1.u_sub2 use work.CFG_SUB2_BEH:config
endconfig
```

The VHDL files, sub2.vhd and sub3.vhd, are as shown below:

```
---Sub2.vhd---
Entity SUB2 is
  Port ( ... );
End SUB2;

Architecture BEH of SUB2 is
Begin
  Process
    ...
  End process;
End BEH;

Configuration work.CFG_SUB2_BEH of SUB2 is
  For BEH

  End for;
End CFG_SUB2_BEH;
---Sub3.vhd---
Entity SUB3 is
  Port ( ... );
End SUB3;
Architecture BEH of SUB3 is
Begin
  Process
    ...
  End process;
End BEH;

Configuration work.CFG_SUB3_BEH of SUB3 is
  For BEH
```

```
End for;
End CFG_SUB3_BEH;
```

The usage model for the above example is shown below:

Analysis

```
% vlogan top.v sub1.v -libmap libmap.v
% vhdlan sub2.vhd sub3.vhd
```

Note:

Specify the VHDL bottom-most entity first, then move up in order.

Elaboration

```
% vcs top
```

Simulation

```
% simv
```

Supported Features

Verilog configuration supports the following features:

- Verilog configurations in mixed HDL designs can configure Verilog instances and boundary VHDL instances (that is, VHDL entity instantiations in a Verilog module). However, the Verilog configuration cannot configure any sub-tree below the VHDL instance in a Verilog module. To configure the sub-tree below the boundary VHDL instances, a separate Verilog configuration must be instantiated in the VHDL design unit.
- Supports direct or component instantiation. It also supports Verilog configuration specification within VHDL.
- The instance resolution happens based on the resolution rules applicable for the instantiating unit. For example, if the unit is in Verilog, then Verilog rules apply, or if the unit is in VHDL, then VHDL rules apply.
- A VHDL design can have multiple Verilog instances with same module name, but with different implementations. They should be analyzed into different logical libraries.
- A VHDL design can instantiate Verilog configuration like VHDL configuration. However, the configuration and the Verilog module that it is configuring must be analyzed in the same logical library as per parent VHDL rules.

- All configuration rules in Verilog configuration for binding instances are supported.
- While resolving the Verilog configuration, the library resolution happens as per the rules mentioned in the IEEE Verilog LRM Std 1364-2005 Section 13.3.1.5. The library order in the `synopsys_sim.setup` file for searching the Verilog or VHDL cell will be ignored.

Limitations of Configurations

In the current implementation, Verilog configurations have the following limitations:

- Verilog configuration cannot have VHDL DUT in the design statement.
- Verilog configurations cannot configure pure VHDL designs.
- The hierarchical path in the instance-based rule of Verilog configuration cannot go through the VHDL instance. The hierarchical path should be pure Verilog with target Verilog or VHDL instance.
- Direct instantiation of the Verilog configuration inside a VHDL generate statement is not supported.
- SystemC with Verilog configurations is not supported for VHDL top-design topology.
- A separate compile flow with Verilog configurations used in mixed HDL designs is not supported.
- Array of instances is not supported.

Using the `-liblist` Option

You can specify the `-liblist` option at elaboration time as follows:

```
-liblist logic_lib1+logic_lib2+
```

It specifies the library search order for unresolved module or entity instantiated in Verilog.

When `-liblist` is specified, VCS starts searching libraries in the order specified with `-liblist`. It neither honors the library order specified in the `synopsys_sim.setup` file, nor look at the other libraries present in the `synopsys_sim.setup` file. If VCS does not find the module definition in the libraries specified with the `-liblist` option, it generates an error message.

In presence of Verilog configuration, VCS first tries to resolve the instances with the configuration rules provided in the configuration file. If no instances are found, then VCS looks for `-liblist`.

In the following example, `-liblist L2` is used to find the instance, `top.11.12`:

Example

```
cat level1.v
*****
```

Chapter 4: Compiling/Elaborating the Design

Key Compilation or Elaboration Features

```

module level1;
    level2 l2();
    initial $display("%l %m level1 (design)");
endmodule

cat file.v
*****
module level1;
    level2 l2();
    initial $display("%l %m level1 (library)");
endmodule

module level2;
    initial $display("%l %m level2 (library)");
    level3 l3();
endmodule

cat file1.v
*****
module level3;
    initial $display("%l %m level3 (library)");
endmodule

cat dummy.v
*****
module dummy;
    level1 l();
endmodule

cat dummy1.v
*****
module dummy;
    level3 l();
endmodule

cat top.v
*****
module top;
    level1 l1();
endmodule

cat topcfg.v
*****
config topcfg;
    design L1.top;
    instance top.l1 liblist L3;

```

```

      default liblist L2 L1;
endconfig

cat synopsys_sim.setup
WORK > DEFAULT
DEFAULT : ./work
L1 : ./lib1
L2 : ./lib2
L3 : ./lib3

cat run
*****
vlogan -sverilog level1.v -work L3
vlogan -sverilog dummy1.v -v file1.v -work L3
vlogan -sverilog dummy.v -v file.v -work L2
vlogan -sverilog file1.v -work L2
vlogan -sverilog top.v -work L1
vlogan -sverilog topcfg.v -work L1
vcs L1.topcfg -diag libconfig -libmap_verbose -liblist L2

```

The following options enable you to search in different order:

- `-liblist_work`
- `-liblist_nocelldiff`

The `-liblist_work` elaboration option enables VCS to first search in the parent work library when attempting to resolve a design cell. This option is relevant only when the Verilog configuration file is not used.

The `-liblist_nocelldiff` elaboration option disables the differentiation between design cell and library cell. You can use this option at elaboration time to override the default precedence order.

Design Cells and Library Cells

VCS designates Verilog modules as either design cells or library cells. When analyzing files without `-v/-y`, Verilog modules are treated as design cells.

When analyzing files using `-v/-y`, Verilog modules resolved from `-v/-y` are treated as library cells.

For example:

- Consider the following command-line:

```
% vlogan a.v
```

If `a.v` contains module `a`, then `a` is treated as the design cell.

- Consider the following command-line where `a.v` is passed with `-v`:

```
% vlogan top.v -v a.v
```

If `top.v` needs module `a` defined in `a.v`, then `a` is treated as the library cell.

By default, design cells take precedence over the library cells while searching for a module definition. That is, if a library cell and a design cell have the same module name, then VCS first searches for the design cell in the applicable libraries. If VCS cannot find the design cell, it will then search for the library cell.

For example, consider the following command:

```
% vcs top.v b_prime.v -v b.v
```

If both `b_prime.v` and `b.v` define the `b` module needed by `top.v`, then VCS gives first preference to the `b` module defined in `b_prime.v` while resolving the instance of the `b` module defined inside the `top` module.

You can use the `-liblist_nocelldiff` option at elaboration time to override the default precedence order. This option does not differentiate between design cells and library cells.

Note:

- If the design and library cell definitions of a module are dumped in the same library, then VCS stores the last dumped design definition. For example, consider the following commands:

```
% vlogan level1.v -work L1
% vlogan top.v -v level1.v -work L1
```

In this case, both design cell and library cell definitions of the `level1` module are analyzed in library `L1` one after another.

Same is applicable for multiple design definitions also and VCS stores the last dumped design definition.

- Design cells take precedence over library cells if the `-liblist_nocelldiff` is not passed at elaboration time.

Library Search Order Rules

This section describes the library search order rules for module definitions when Verilog configuration file is used with `-v/-y` and `-liblist` options. This section consists of the following subsections:

- [Library Search Order Rules for Mixed HDL Designs](#)
- [Library Search Order Rules for Verilog or SystemVerilog Designs](#)

Library Search Order Rules for Mixed HDL Designs

VCS searches for the same language instances as that of the parent. If the same language child instance is not available, then VCS searches for other language instance.

For example, if both Verilog and VHDL child instances are available in the same library, a VHDL parent gives priority for a VHDL child instance over a Verilog child instance. Also, the Verilog parent gives priority for the Verilog child instance over the VHDL child instance. Further, VHDL parent search order is defined by the VHDL language and Verilog parent search order for various conditions are explained in detail in the subsequent sections.

For any Verilog parent design cell, Verilog configuration is honored until the VHDL instance is hit. Subsequent VHDL hierarchy can be configured using VHDL configurations until Verilog instance is hit. The subsequent Verilog hierarchy beneath VHDL can be configured using appropriate Verilog configurations. In absence of appropriate Verilog configuration, regular search order is followed.

The subsequent sections discuss library search order rules for mixed HDL designs with and without the Verilog configuration file. The library search order rules are listed in order of priority, where the first rule is given preference before the second rule.

Library Search Order Rules for Mixed HDL Designs Without Configuration File

Library Search order rules for mixed HDL designs without a configuration file are as follows:

1. VHDL-VHDL Scenario:

Binding rule of VHDL is applied. The `-liblist` option does not have any impact on this scenario.

2. VHDL-Verilog Scenario:

Binding rule of VHDL is applied. The `-liblist` option does not have any impact on this scenario.

3. Verilog-VHDL Scenario:

The VHDL instantiation gets resolved to a VHDL entity as per the library order specified in `-liblist`.

The `-liblist` option is applicable for Verilog parent only.

Library Search Order Rules for Mixed HDL Designs With Configuration File

Library Search order rules for mixed HDL designs with a configuration file are as follows:

1. VHDL-VHDL Scenario:

Binding rule of VHDL is applied. The `-liblist` option does not have any impact on this scenario.

2. VHDL-Verilog Scenario:

Binding rule of VHDL is applied. The `-liblist` option does not have any impact on this scenario.

3. Verilog-VHDL Scenario:

For Verilog-top designs, Verilog cells are searched, if not found, then VHDL cells are searched as per the rules. For any Verilog instantiated in the VHDL, the configuration rules are not applicable.

Library Search Order Rules for Verilog or SystemVerilog Designs

The library search order rules are listed in order of priority, where the first rule is given preference before the second rule. Each rule contains multiple sub-rules, which are listed in the order of priority. VCS first determines the rule of highest priority, and then searches the sub-rules in that rule, in the order of priority.

This section consists of the following subsections:

- [Library Search Order Rules for Verilog or SystemVerilog Designs With a Configuration File](#)
- [Library Search Order Rules for Verilog or SystemVerilog Designs Without a Configuration File](#)

Library Search Order Rules for Verilog or SystemVerilog Designs With a Configuration File

RULE1: `uselib

1. Any library cell pre-resolved from a ``uselib` file during `vlogan`:
 - a. If a cell is resolved from ``uselib`, then no configuration rules are applied to an instance in it if it was pre-resolved in the ``uselib` library. For usage example, see [Example-1](#)
 - b. If the ``uselib` instance is configured to resolve a design cell, then configuration rules are applied for any hierarchy under it. For usage example, see [Example-2](#)
 - c. If a cell is resolved from ``uselib`, and an instance in it was not pre-resolved from ``uselib` during analysis, then if it finds a design cell during elaboration, it is taken.

In this case, the hierarchy underneath `uselib can be configured. For usage example, see [Example-3](#)

- d. If a design cell for an instance is found during vlogan itself, VCS does not look for `uselib or -v/-y. For usage example, see [Example-4](#)

RULE2: Instance use Clause

1. Configuration instance use clause
 - a. VCS looks for an instance in the use clause specified in the configuration. For usage example, see [Example-5](#)
 - b. If not found, VCS generates an error message.

RULE3: Instance liblist Clause

1. Configuration instance liblist clause
 - a. Resolves the design cell using the configuration instance liblist clause, if the liblist argument is not empty. For usage example, see [Example-6.1](#).
 - b. Looks for the design cell in the parent cell library, if liblist argument is empty. For usage example, see [Example-6.2](#).
 - c. Looks for the design cell in libraries specified with -liblist at the command line. For usage example, see [Example-6.3](#).
 - d. If the cell was analyzed using -v or -y in the same vlogan invocation as the parent.
 - e. Looks for the library cell in the configuration instance liblist clause, if the liblist argument is not empty.
 - f. Looks for the library cell in the parent cell library, if the liblist argument is empty.
 - g. Looks for library cells in the libraries specified with -liblist at the command line.

RULE4: Cell use Clause

1. Configuration Cell use Clause
 - a. Looks for the design cell in the use clause.
 - b. If not found, VCS generates an error message.

RULE5: Cell liblist Clause

1. Configuration cell `liblist` clause
 - a. Resolves the design cell using the configuration cell `liblist` clause, if the `liblist` argument is not empty.
 - b. Looks for the design cell in the parent cell library, if the `liblist` argument is empty.
 - c. Looks for the design cell in the libraries specified with `-liblist` at the command line.
 - d. If the cell was analyzed using `-v` or `-y` in the same `vlog` invocation as the parent.
 - e. Looks for the library cell in the configuration cell `liblist` clause, if the `liblist` argument is empty.
 - f. Looks for the library cell in the libraries, if the `liblist` argument is not empty.
 - g. Looks for the libraries specified with `-liblist` at the command line.

RULE6: Configuration Inherited liblist Clause

1. Configuration inherited `liblist` clause (derived from instance `liblist` or from cell `liblist`)
 - a. Looks for the design cell in the configuration `liblist` (instance or cell clause)
 - b. Looks for the design cell in the parent cell library, if the `liblist` argument is empty.
 - c. Looks for the libraries specified with `-liblist` at the command line.
 - d. If the cell was analyzed using `-v` or `-y` in the same `vlog` invocation as the parent.
 - e. Looks for the library cell in the configuration cell `liblist` clause.
 - f. Looks for the library cell in the parent cell library, if the `liblist` argument is empty.
 - g. Looks for the libraries specified with `-liblist` at the command line.

RULE7: Configuration Default liblist Clause

1. Configuration default `liblist` clause
 - a. Looks for the design cell in the configuration default `liblist`. For usage example, see [Example-7.1](#).
 - b. Looks for the design cell in the parent cell library, if there are no arguments in the configuration default `liblist`.

- c. Looks for the design cell definition in the libraries specified with `-liblist` at the command line. For usage example, see [Example-7.2](#).
- d. If the cell was analyzed using `-v` or `-y` in the same vlogan invocation as the parent.
- e. Looks for the library cell definition in the configuration default `liblist`.
- f. Looks for the library cell definition in the parent cell library, if there are no arguments in the configuration default `liblist`.
- g. Looks for the library cell definition in the libraries specified with `-liblist` at the command line. For usage example, see [Example-7.3](#).

RULE8: No Configuration Rule Applies

1. No configuration rule applies
 - a. Looks for the design cell in the parent cell library. For usage example, see [Example-8.1](#).
 - b. Looks for the libraries specified with `-liblist` at the command line. For usage example, see [Example-8.2](#).
 - c. Looks for the library cell in the parent cell library.
 - d. Looks for the libraries specified with `-liblist` at the command line.

Library Search Order Rules for Verilog or SystemVerilog Designs Without a Configuration File

-liblist

Specify `-liblist` at the command line.

Search Order:

1. Look in the libraries specified at the command line.
2. If not found, VCS generates an error message.

For usage example, see [Example-9](#).

-liblist_work

This option tells VCS to first look in the parent work library when trying to resolve a design cell. That is, while resolving module instances, this option ensures to always start search for child modules from the parent work library before searching in the other analyzed libraries.

This elaboration option is relevant only for non-configuration flow (that is, when you do not use the Verilog configuration file).

Search Order:

1. Look for the design cell in the parent cell library.
2. Look for the libraries specified in `-liblist` at the command line, or look for the libraries specified in the `synopsys_sim.setup` file if `-liblist` is not passed.
3. Not applicable in the configuration flow. It does not apply in the configuration flow. It is ignored if top is configured.
4. In case of , `-liblist_work` applies only for Verilog modules, and does not have any impact on the VHDL module.

For usage example, see [Example-10](#).

-liblist_work and -liblist

`-liblist_work` takes precedence over `-liblist`.

-liblist_nocelldiff

If this option is used, VCS does not differentiate between design cells and library cells. By default, VCS prefers design cells over library cells while resolving instances. It follows the configuration rules as applied to the `-liblist` options. For usage example, see [Example-11](#).

Example Testcase Files

Consider the following design files:

Testcase: `file1.v`

```
module a(in1, in2, out1);
  input in1, in2;
  output out1;
  initial
    $display ("Instance %m Module a from file1.v %l");
    b b1(in1, in2, out1);
endmodule

module t1;

  reg r1, r2;
  wire w1;

  a a1(r1,r2,w1);

  initial
```

```
$display ( "Instance %m Module t1 from file1.v %l");
endmodule
```

Testcase: file2.v

```
module a(in3, in4, out2);
  input in3, in4;
  output out2;

  initial
    $display ( "Instance %m Module a from file2.v %l");
    b b1(in3,in4,out2);
  endmodule

  module t2;
    reg r2, r4;
    wire w2;

    a a1(r2,r4,w2);

    initial
      $display ( "Instance %m Module t2 from file2.v %l");
    endmodule
```

Testcase: modb_1.v

```
module b(in1, in2, out1);
  input in1, in2;
  output out1;
  initial
    $display ( "Instance %m Module b from modb_1.v %l");
  endmodule
```

Testcase: modb_2.v

```
module b(in3, in4, out2);
  input in3, in4;
  output out2;
  initial
    $display ( "Instance %m Module b from modb_2.v %l");
  endmodule
```

Testcase: top.v

```
module top;
  t1 t1_1();
  t2 t2_1();

  initial
    $display ( "Instance %m Module top from top.v %l");
  endmodule
```

Usage Examples for Library Search Order Rules for Verilog or SystemVerilog Designs

Usage Examples for RULE1

This section describes the examples for sub-rules listed under [RULE1:`uselib](#) of the [Library Search Order Rules for Verilog or SystemVerilog Designs](#) section.

Example-1

Consider the following design files:

Testcase: file.v

```
module level1;
    level2 l2();
    initial $display("%l %m level1 (uselib)");
endmodule

module level2;
    initial $display("%l %m level2 (uselib)");
endmodule
```

Testcase: level2.v

```
module level2;
    initial $display("%l %m level2 (design)");
endmodule

module M;
    initial $display("%l %m M");
endmodule
```

Testcase: top.v

```
module top;
    `uselib file="file.v"
    level1 l1();
endmodule
```

Configuration file: cfg.v

```
config topcfg;
    design L1.top;
    instance top.l1.l2 use L2.level2;
endconfig
```

VCS commands:

```
mkdir L1 L2
vlogan -sverilog level2.v -work L2
vlogan -sverilog top.v -work L1
vlogan -sverilog cfg.v -work L1
```

```
vcs L1.topcfg -diag libconfig -libmap_verbose
simv
```

synopsys_sim.setup file:

```
WORK > DEFAULT
DEFAULT : ./work
L1 : ./lib1
L2 : ./lib2
L3 : ./lib3
```

Description

Library search order rules to resolve the top.11.12 instance are as follows:

1. The configuration file cfg.v specifies a configuration rule for the top.11.12 instance (as the top.11.12 instance uses L2.level2), which indicates to resolve the 12 definition from the L2.level2 library.
2. As per the search rule, as the cell level1 (instance 11) is resolved from the `uselib file in top.v, instance 12 under 11 is also resolved from the `uselib library only.
3. Instance 12 cannot be resolved from the configuration rule as given in cfg.v.
4. Configuration verbose output shows that 12 is resolved from the `uselib library.

```
instance: top.11.12
  rule: `uselib
  module: L1.level2
```

Example-2

Consider the following design files:

Testcase: file.v

```
module level1;
    level2 l2();
    initial $display("%l %m level1 (uselib)");
endmodule

module level2;
    initial $display("%l %m level2 (uselib)");
endmodule
```

Testcase: level1.v

```
module level1;
    level2 l2();
    initial $display("%l %m level1 (design)");
endmodule

module M;
```

```
initial $display("%l %m M");
endmodule
```

Testcase: top.v

```
module top;
  `uselib file="file.v"
  level1 l1();
endmodule
```

Configuration file: cfg.v

```
config topcfg;
  instance top.l1 use L2.level1;
  instance top.l1.l2 use L2.M;
endconfig
```

VCS commands:

```
mkdir L1
vlogan -sverilog level1.v -work L2
vlogan -sverilog top.v -work L1
vlogan -sverilog topcfg.v -work L1
vcs L1.topcfg -diag libconfig -libmap_verbose
```

synopsys_sim.setup:

```
WORK > DEFAULT
DEFAULT : ./work
L1 : ./lib1
L2 : ./lib2
L3 : ./lib3
```

Description

Library search order rules to resolve the `top.l1` instance are as follows:

- As there is a configuration rule mentioned in the `cfg.v` file (The `top.l1` instance uses `L2.level1`) to resolve the definition from `L2.level1`, instance `l1` is resolved as per the configuration rule.
- The `top.l1.l2` instance is also resolved as per the configuration rule and uses `L2.M`.

```
instance: top.l1
rule: instance top.l1 uses L2.level1;
module: L2.level1

instance: top.l1.l2
rule: instance top.l1.l2 uses L2.M;
module: L2.M
```

Example-3

Consider the following design files:

Testcase: file.v

```
module level1;
    level2 l2();
    initial $display("%l %m level1 (uselib)");
endmodule
```

```
module level2;
    initial $display("%l %m level2 (uselib)");
endmodule
```

Testcase: level2.v

```
module level2;
    initial $display("%l %m level2 (design)");
endmodule
```

```
module M;
    initial $display("%l %m M");
endmodule
```

Testcase: top.v

```
module top;
    `uselib file="file.v"
    level1 l1();
endmodule
```

Configuration file: cfg.v

```
config topcfg;
    design L1.top;
    instance top.l1.l2 use L2.M;
endconfig
```

VCS commands:

```
mkdir L1 L2
vlogan -sverilog level2.v -work L2
vlogan -sverilog top.v -work L1
vlogan -sverilog cfg.v -work L1
vcs L1.topcfg -diag libconfig -libmap_verbose
```

synopsys_sim.setup file:

```
WORK > DEFAULT
DEFAULT : ./work
L1 : ./lib1
L2 : ./lib2
L3 : ./lib3
```

Description

Library search order rules to resolve the top.11.12 instance are as follows:

1. The level1 (instance 11) cell is resolved from the `uselib file, but the 12 instance is not resolved during vlog analysis.
2. Configuration rules are applied to resolve the 12 instance, and the instance is resolved from the L2.M library.

```
instance: top.11.12
rule: instance top.11.12 use L2.M;
module: L2.M
```

Example-4

Consider the following design files:

Testcase: file.v

```
module level1;
    level2 l2();
    initial $display("%l %m level1 (uselib)");
endmodule
```

Testcase: level1.v

```
module level1;
    initial $display("%l %m level1 (design)");
endmodule

module level2;
    initial $display("%l %m level2 (design)");
endmodule
module M;
    initial $display("%l %m M");
endmodule
```

Testcase: top.v

```
module top;
    `uselib file="file.v"
    level1 l1();
endmodule
```

Configuration file: cfg.v

```
config topcfg;
    design L1.top;
endconfig
```

VCS commands:

```
mkdir L1 L2
vlogan -sverilog level2.v -work L2
vlogan -sverilog top.v level2.v -work L1
vlogan -sverilog cfg.v -work L1
vcs L1.topcfg -diag libconfig -libmap_verbose
```

synopsys_sim.setup:

```
WORK > DEFAULT
DEFAULT : ./work
L1 : ./lib1
L2 : ./lib2
L3 : ./lib3
```

Description

To resolve `top.11`, VCS does not look at ``uselib` or `-v/-y` as the design cell `level2` for an instance `12` is found during `vlogan` itself.

```
instance: top.11
rule: parent cell's library
module: L1.level1
```

Usage Examples for RULE2

This section describes the examples for sub-rules listed under [RULE2: Instance use Clause of the Library Search Order Rules for Verilog or SystemVerilog Designs](#) section.

Consider the design files and hierarchy mentioned in the [Example Testcase Files](#) section.

Example-5

Consider the following configuration file and VCS commands:

Configuration file: `cfg.v`

```
config topcfg;
  design lib3.top;
  instance top.t2_1.a1 use LIB2.a
endconfig
```

VCS commands:

```
vlogan modb_1.v -work lib_b1 +v2k -q
vlogan modb_2.v -work lib_b2 +v2k -q
vlogan file1.v -work lib1 +v2k -q
vlogan file2.v -work lib2 +v2k -q
vlogan top.v -work lib3 +v2k -q
vlogan +v2k -work work cfg.v -q
vcs work.cfg -liblist lib1+lib2+lib3+lib_b1+lib_b2 -diag libconfig
```

Description

Library search order rules to resolve the `top.t2_1.a` instance are as follows:

1. VCS searches for a module definition of `a` in `LIB2.a` as provided by the configuration rules.
2. VCS search completes once it finds the definition in `LIB2.a`.

```
instance: top.t2_1.a1
rule: instance top.t2_1.a1 use LIB2.a;
module: LIB2.a
```

Usage Examples for RULE3

This section describes the examples for sub-rules listed under [RULE3: Instance liblist Clause](#) of the [Library Search Order Rules for Verilog or SystemVerilog Designs](#) section.

Example-6

Consider the design files and hierarchy mentioned in the [Example Testcase Files](#) section.

Configuration file: `cfg.v`

```
config cfg;
  design lib3.top;
  instance top.t2_1.a1 liblist
  instance top.t1_1.a1 liblist lib1
endconfig
```

VCS commands:

```
vlogan +v2k modb_2.v -work lib_b2
vlogan +v2k modb_1.v -work lib_b1
vlogan +v2k file2.v -work lib2
vlogan +v2k file2.v -work lib3
vlogan +v2k file1.v -work lib1
vlogan +v2k -work lib3 top.v
vlogan +v2k cfg.v
vcs work.cfg -liblist lib2+lib1+lib3+lib_b1+lib_b2 -diag libconfig
-libmap_verbose
simv
```

Example-6.1

VCS resolves the design cell using the configuration `instance liblist clause` if `liblist` is not empty.

Consider the design files, the configuration file, and VCS commands mentioned in [Example-6](#).

Description

Library search order rules to resolve the `top.t1_1.a1` instance are as follows:

1. Search for the module definition of `a` in `liblist lib1` as provided by the configuration rule in `cfg.v` (`instance top.t1_1.a1 liblist lib1`)
2. Search completes if the definition is found in `lib1`.
3. Configuration verbose output:

```
instance: top.t1_1.a1
rule: instance top.t1_1.a1 liblist lib1;
module: LIB1.a
```

Example-6.2

VCS looks for the design cell in parent cell library, if the `liblist` argument is empty.

Consider the design files, the configuration file, and VCS commands mentioned in [Example-6](#).

Description

Library search order rules to resolve the `top.t2_1.a1` instance are as follows:

1. The `liblist` argument in the configuration rule is empty (`instance top.t2_1.a1 liblist`).
2. If the argument after `liblist` is empty, then VCS looks for the module definition in the parent cell library.
3. VCS looks for the module definition of `a` in the parent cell library which is `t2_1`.
4. Parent cell library of `t2_1` is in `LIB3`.
5. VCS tries to resolve the module definition of `a` also in `LIB3`.
6. Search completes if the definition of `a` is found.
7. Configuration verbose output

```
instance: top.t2_1.a1
rule: instance top.t2_1.a1 liblist;
module: LIB3.a
```

Example-6.3

VCS looks for libraries specified in `-liblist` at the command line.

Consider the design files, the configuration file, and VCS commands mentioned in [Example-6](#).

Description

1. For the `top.t1_1.a1.b1` instance, there is no specific rule applied in the configuration file.
2. VCS looks for the module definition `a` first in the parent cell library, if the default configuration is not present in the configuration file. In this example, the parent cell library of the `a` module is `lib1`. VCS does not find the `b` module definition in `lib1`.
3. VCS then looks for libraries specified at `-liblist` at the command line in the order provided.
4. VCS searches in `LIB2`. If not found in `LIB2`, VCS searches in `LIB1`. Then, VCS searches in `LIB3`. If it finds the definition, search gets complete.
5. Configuration verbose output:

```
instance: top.t1_1.a1.b1
rule: default library search order
module: LIB_B1.b
```

Usage Examples for RULE7

This section describes the examples for sub-rules listed under [RULE7: Configuration Default liblist Clause](#) of the [Library Search Order Rules for Verilog or SystemVerilog Designs](#) section.

Example-7

Consider the design files and hierarchy mentioned in the [Example Testcase Files](#) section.

Configuration file: cfg.v

```
config cfg;
  design lib3.top;
  instance top.t2_1.a1 liblist
    default liblist lib_b2 lib2;
endconfig
```

VCS commands:

```
vlogan +v2k modb_2.v -work lib_b2
vlogan +v2k modb_1.v -work lib_b1
vlogan +v2k file2.v -work lib2
vlogan +v2k file2.v -work lib3
vlogan +v2k file1.v -work lib1
vlogan +v2k -work lib3 top.v
vlogan +v2k cfg.v
vcs work.cfg -liblist lib2+lib1+lib3+lib_b1+lib_b2 -diag libconfig
-libmap_verbose
```

Example-7.1

VCS looks for the design cell in the configuration default liblist.

Consider the design files, the configuration file, and VCS commands mentioned in [Example-7](#).

Description

Library search order rules to resolve the `top.t1_1.a1.b1` instance are as follows:

1. VCS looks for default libraries specified in the default liblist in the configuration file (`default liblist lib_b2 lib1 lib2`)
2. VCS finds the definition in `LIB_B2`, and the search gets completed.

```
instance: top.t1_1.a1.b1
rule: default liblist lib_b2 lib2;
module: LIB_B2.b
```

Example-7.2

VCS looks for the design cell definition in the libraries specified with `-liblist` at the command line.

Consider the design files, the configuration file, and VCS commands mentioned in [Example-7](#).

Description

Library search order rules to resolve the `top.t1_1` instance are as follows:

1. VCS searches in the default configuration liblist provided in the configuration file, `cfg.v` (`default liblist lib_b2 lib2`).
2. VCS could not find in any of the default libraries `lib_b2` and `lib2` definition of `t1_1`.
3. VCS looks for the libraries specified with `-liblist` at the command line (`lib2+lib1+lib3+lib_b1+lib_b2`).
4. VCS starts looking for the `t1_1` definition and finds in `LIB1`.
5. VCS search completes.

```
instance: top.t1_1
rule: default library search order
module: LIB1.t1
```

Example-7.3

VCS looks for the library cell definition in the libraries specified in the `-liblist` at the command line.

Consider the design files mentioned in [Example-7](#).

Configuration file: `cfg.v`

```
config cfg;
    design lib3.top;
    instance top.t2_1.a1 liblist;
    instance top.t1_1.a1 liblist lib3
        default liblist lib_b2 lib1 lib2;
endconfig
```

VCS commands:

```
vlogan +v2k file2.v -work lib3 -v modb_2.v
vlogan +v2k file2.v -work lib3
vlogan +v2k file1.v -v modb_1.v -work lib_b1
vlogan +v2k file1.v -work lib1
vlogan +v2k -work lib3 top.v
vlogan +v2k cfg.v
vcs work.cfg -liblist lib1+lib3+lib_b1+lib_b2 -diag libconfig
    -libmap_verbose
```

Description

Library search order rules to resolve the `top.t2_1.a1.b1` instance are as follows:

1. VCS searches in the default library list `lib_b2`, `lib`, and `lib2`. However, it could not find the definition.
2. VCS then looks at the libraries specified in `-liblist` at the command line in the order provided `lib1+lib3+lib_b1+lib_b2`.
3. VCS finds the definition in `LIB3` and search gets completed.

```
instance: top.t2_1.a1.b1
rule: default library search order
module: LIB3.b
```

Usage Examples for RULE8

This section describes the examples for sub-rules listed under [RULE8: No Configuration Rule Applies](#) of the [Library Search Order Rules for Verilog or SystemVerilog Designs](#) section.

Example-8

Consider the design files and hierarchy mentioned in the [Example Testcase Files](#) section.

Configuration file: `cfg.v`

```
config cfg;
    design lib3.top;
endconfig
```

VCS commands:

```
vlogan +v2k modb_2.v -work lib_b2
vlogan +v2k modb_1.v -work lib_b1
vlogan +v2k file2.v -work lib2
vlogan +v2k file2.v -work lib3
vlogan +v2k file1.v -work lib1
vlogan +v2k -work lib3 top.v
vlogan +v2k cfg.v
vcs work.cfg -liblist lib2+lib1+lib3+lib_b1+lib_b2 -diag libconfig
-libmap_verbose
```

Example-8.1

VCS looks for the design cell in the parent cell library.

Consider the design files, the configuration file, and VCS commands mentioned in [Example-8](#).

Description

Library search order rules to resolve the `top.t2_1.a1` instance are as follows:

1. No configuration rule applies for the `top.t2_1.a1` instance, and there is no default `liblist` configuration present. Therefore, VCS looks at the parent cell library.
2. Parent cell library for the parent module `t2` is `LIB3`.
3. VCS finds the definition of `a` in `LIB3` and the search gets completed.

```
instance: top.t2_1.a1
rule: parent cell's library
module: LIB3.a
```

Example-8.2

VCS looks for the libraries specified with `-liblist` at the command line.

Consider the design files, the configuration file, and VCS commands mentioned in [Example-8](#).

Description

Library search order rules to resolve the `top.t2_1.a1.b1` instance are as follows:

1. No configuration rules applies for instance `top.t2_1.a1.b1`.
2. VCS tries to look into the parent cell library but could not find the definition.
3. VCS looks for the libraries mentioned with `-liblist` at the command line.
4. VCS searches in the libraries specified in the order specified at the command line.
5. VCS finds the definition in `LIB_B1`, and then the search gets completed.

```
instance: top.t2_1.a1.b1
rule: default library search order
module: LIB_B1.b
```

Usage Examples for Library Search Order Rules for Verilog or SystemVerilog Designs Without a Configuration File

This section describes the examples for library search order rules for Verilog or SystemVerilog designs without a configuration file.

Example-9

Consider the design files and hierarchy mentioned in the [Example Testcase Files](#) section.

VCS commands:

```
vlogan +v2k modb_2.v -work lib_b2
vlogan +v2k modb_1.v -work lib_b1
vlogan +v2k file2.v -work lib2
vlogan +v2k file1.v -work lib1
vlogan +v2k -work lib3 top.v
vcs top -liblist lib1+lib2+lib3+lib_b1+lib_b2 -diag libconfig
-libmap_verbose -liblist_nocelldiff
```

Description

Library search order rules to resolve the top.t2_1.a1.b1 instance are as follows:

1. Looks for the definition of module b.
2. Starts scanning the library files provided with -liblist from left to right.
3. Looks in library lib1. If it does not find in lib1, it looks in library lib2. If it does not find in lib2, it looks in library lib3.
4. Looks in library lib_b1, finds the definition of b. The search gets completed.

```
instance: top.t2_1.a1.b1
rule: default library search order
module: LIB_B1.b
```

Example-10

Consider the design files and hierarchy mentioned in the [Example Testcase Files](#) section.

VCS commands:

```
vlogan +v2k modb_2.v -work lib_b2
vlogan +v2k modb_2.v -work lib3
vlogan +v2k modb_1.v -work lib_b1
vlogan +v2k modb_1.v -work lib3
vlogan +v2k file2.v -work lib2
vlogan +v2k file2.v -work lib3
```

```
vlogan +v2k file1.v -work lib1
vlogan +v2k file1.v -work lib3
vlogan +v2k -work lib3 top.v
vcs top -liblist_work -diag libconfig -libmap_verbose
```

Description

Library search order rules to resolve the `top.t2_1.a1.b1` instance are as follows:

1. Looks for the design cell `b`.
2. As `-liblist_work` is present at elaboration time, VCS first tries to search the parent work library.
3. Parent module of `b` is module `a`, and is available in the `LIB3` library.
4. VCS tries to search the definition of `b` in `LIB3`.
5. VCS finds the definition of `b` in `LIB3`, and the search gets completed.

```
instance: top.t2_1.a1.b1
rule: parent cell's library
module: LIB3.b
```

Example-11

Consider the design files and hierarchy mentioned in the [Example Testcase Files](#) section.

VCS commands:

```
vlogan +v2k modb_2.v -work lib_b2
vlogan +v2k modb_1.v -work lib_b1
vlogan +v2k file2.v -v modb_2.v -work lib2
vlogan +v2k file2.v -work lib3
vlogan +v2k file1.v -work lib1
vlogan +v2k -work lib3 top.v
vcs top -liblist lib1+lib2+lib3+lib_b1+lib_b2 -diag libconfig
-libmap_verbose -liblist_nocelldiff
```

Description

Library search order rules to resolve the `top.t2_1.a1.b1` instance are as follows:

1. Looks for the definition of module `b` in library `LIB2` (library cell) and in `LIB3` (design cell).
2. With `-liblist_nocelldiff`, definition of `b` is resolved from library `LIB2` as it is available in `LIB2`.
3. `LIB2` comes first in the order of `-liblist (lib1+lib2+lib3+lib_b1+lib_b2)` provided at elaboration time.

```
instance: top.t2_1.a1.b1
rule: default library search order
module: LIB2.b
```

4. With no presence of `-liblist_nocelldiff` in the same example, definition of `b` is resolved from the `LIB_B1`(design cell) library as it is available in `LIB_B1`.

```
instance: top.t2_1.a1.b1
rule: default library search order
module: LIB_B1.b
```

Lint Warning Message for Missing 'endcelldefine'

You can tell VCS to display a lint warning message if your Verilog or SystemVerilog code contains a `'celldefine` compiler directive without a corresponding `'endcelldefine` compiler directive and vice versa.

You enable this warning message with the `+lint=CDUB` VCS compile-time option `vlogan` command-line option. The CDUB argument stands for “Compiler Directives Unbalanced.”

The examples in this section show the warning message and the source code that results in its display.

Example 8 Source Code With Missing 'endcelldefine'

```
`celldefine
module mod;
endmodule
```

In this example, there is no corresponding `'endcelldefine` compiler directive.

In VCS two-step flow, if you execute the following `vcs` command:

```
vcs exp1.v +lint=CDUB
```

VCS displays the following Lint warning message:

```
Lint-[CDUB] Compiler directive unbalanced
exp1.v, 1
  Unbalanced compiler directive is detected : `celldefine
  has no matching `endcelldefine.
  Please make sure that all directives are balanced.
```

In VCS, `vlogan` also displays this lint warning message when you execute the following command:

```
vlogan exp1.v +lint=CDUB
```

The source code in [Example 9](#) does not display this warning message when you include the `+lint=CDUB`.

Example 9 Source Code With 'celldefine and 'endcelldefine

```
`celldefine
module mod;
endmodule
`endcelldefine
```

It does not display the warning message because there is the `'endcelldefine` compiler directive after the `'celldefine` compiler directive in the source code.

Instead of the `'endcelldefine` compiler directive, you can substitute the `'resetall` compiler directive, as shown in [Example 10](#).

Example 10 Source Code With 'celldefine and 'resetall

```
`celldefine
module mod;
endmodule
`resetall
```

The source code in both [Example 9](#) and [Example 10](#) does not result in the warning message when you include the `+lint=CDUB` option.

Also with the `+lint=CDUB` option, if your source code contains the `'endcelldefine` compiler directive without the preceding and corresponding `'celldefine` compiler directive, you see a similar warning message.

Example 11 'endcelldefine Without a Preceding and Corresponding 'celldefine

```
module mod;
endmodule
`endcelldefine
```

With the `+lint=CDUB` option, this source code results in the following lint warning message:

```
Lint-[CDUB] Compiler directive unbalanced
exp6.v, 3
  Unbalanced compiler directive is detected : `endcelldefine
  has no matching `celldefine.
  Please make sure that all directives are balanced.
```

With the `+lint=CDUB` option, it is not just that the number of `'endcelldefine` compiler directives must be equal to the number of `'celldefine` compiler directives. The `'endcelldefine` compiler directive must follow the `'celldefine` compiler directive before there is another `'celldefine` compiler directive.

Example 12 Equal Number of 'celldefine and 'endcelldefine, But Not in the Required Sequence

```
`celldefine  \\ line 1
module mod;
endmodule
```

```
'celldefine
module schmodule;
endmodule

`endcelldefine

`endcelldefine \\ line 11
```

In [Example 12](#), the number of `'celldefine` compiler directives matches the number of `'endcelldefine` compiler directives, but they are not in a corresponding sequence, which results in the following lint warning messages:

```
Lint-[CDUB] Compiler directive unbalanced
exp5.v, 1
  Unbalanced compiler directive is detected : `celldefine
  has no matching `endcelldefine.
  Please make sure that all directives are balanced.
```

```
Lint-[CDUB] Compiler directive unbalanced
exp5.v, 11
  Unbalanced compiler directive is detected : `endcelldefine
  has no matching `celldefine.
  Please make sure that all directives are balanced.
```

Limitation

The `'celldefine/'endcelldefine` compiler directives must be matched serially. Recursive `'celldefine/'endcelldefine` directives are not supported with the `+lint=CDUB` option and keyword argument, for example:

Example 13 Recursive 'celldefine/"endcelldefine Compiler Directives

```
'celldefine
'celldefine
module dev (....,...);
'celldefine
'celldefine
module dev (....,...);
  ...
endmodule
'endcelldefine
'endcelldefine
```

[Example 13](#) shows redundant and unnecessary `'celldefine` and `'endcelldefine` compiler directives, but does not prevent compilation. The `+lint=CDUB` option and

keyword argument triggers the unbalanced message of Lint compiler directives when VCS reads another '`celldefine`' directive before reading an '`endcelldefine`' directive,

Error/Warning/Lint Message Control

You can control error, warning, and lint messages in the following two ways:

- For `-error`, `-suppress`, `+lint`, and `+warn` compile options, see [Controlling Error/Warning/Lint Messages Using Compile-Time Options on page 214](#).
- With a configuration file that you specify with the following compile-time option:

`-msg_config=message_configuration_file_name`

Using a configuration file, you can control lint, warning, and error messages that VCS displays according to the following:

- by source file name
- by module name
- by design subhierarchy

See [Controlling Error/Warning/Lint Messages Using a Configuration File](#).

Controlling Error/Warning/Lint Messages Using Compile-Time Options

The `-error`, `-suppress`, `+lint`, and `+warn` options control error and warning messages. With them, you can:

- Disable the display of any lint, warning, or error messages.
- Disable the display of specific messages.
- Limit the display of specific messages to a maximum number that you specify.

To control the display of specific messages, you need their message IDs. A message ID is the character string in a message between the square brackets []. In [Figure 3](#), the message ID is MFACF.

Note:

The `-error` option is also a runtime option.

Figure 3 Message IDs



message ID

Warning-[MFACF] Missing flag argument
 Argument for flag 'verboseLevel' is missing in config statement, it will be ignored.
 Config file : error_id0_id1.cfg, starting at line 4.

The new compile-time options for controlling messages and their syntax are as follows:

```
-error=[no]message_ID[:max_number],...|none|all  

-error=all,noWarn_ID|noLint_ID  

+warn=[no]message_ID[:max_number],...|none|all  

+lint=[no]message_ID[:max_number],...|none|all  

-suppress[=message_ID,...]
```

Note:

The `-error` option is also a runtime option. However, only the following feature is supported at runtime:

```
-error=[no]message_ID[:max_number],...
```

These compile-time options and their arguments are described in the following sections:

- [Controlling Error Messages](#)
- [Controlling Lint Messages](#)
- [Suppressing Lint, Warning, and Error Messages](#)
- [Error Conditions and Messages That Cannot Be Disabled](#)
- [Using Message Control Options Together](#)

Controlling Error Messages

You can control error messages with the `-error` option in the following ways:

- Limit the number of occurrences of an error message to a number you specify. For this, specify the message ID as an argument to the `-error` option along with the specified maximum number of occurrences.
- Disable the display of all error messages which are downgradable with the `none` argument.
- Enables the display of all errors/warnings/lint messages with the `all` argument to the `-error` option.

Upgrading Lint and Warning Messages to Error Messages

If you enter the message ID for a warning or lint message as an argument to the `-error` option, VCS upgrades the condition causing the warning or lint message to an error condition and an error message.

Controlling Warning Messages

Like error messages, you can control warning messages with the `+warn` option in the following ways:

- Limit the number of occurrences of a warning message to a number you specify. For this, specify the message ID as an argument to the `+warn` option along with the specified maximum number of occurrences.
- Disable the display of a particular warning message by entering the keyword `no` as an argument and appending to this keyword the message ID, for example:

`+warn=noTFIPC`

This option disables the display of the error message with the TFIPC message ID.

Note:

- Do not enter a maximum number of occurrences, even if 0, if also appending the `no` keyword to the message ID.
- Disable the display of all warning messages with the `none` argument to the `+warn` option.
- Enable the display of all warning messages with the `all` argument to the `+warn` option.
- Controls the display of all notes. For example,

`+warn=noFCICIO`

This option suppresses the display of the following note:

`Note-[FCICIO] Instance coverage is ON`

Upgrading Lint Messages to Warning Messages

Note:

- All lint/warning messages are suppressible. But only some of the error messages can be downgraded or suppressed.
- You cannot downgrade all error conditions and messages to a warning condition and message. Entering a message ID for an error message that cannot be downgraded as an argument to the `+warn` option results in VCS ignoring the message ID and displaying a warning message similar to the following:

```
Warning-[CSMC] Cannot set message count
Failed to set display count for message id 'TFAFTC' because cannot
set count
for non-warning ID in '+warn' switch.
Specified count is ignored.
```

For an example of this warning see [Example 4: An Error Message That Cannot Be Controlled](#).

This warning message is in response to the `+warn=TFAFTC:2` option, when `TFAFTC` is the ID for the following error message:

```
Error-[TFAFTC] Too few arguments to function/task call
tfatc_err.v, 9
"wrFld4(.bus(1));"
The above function/task call is not done with sufficient arguments.
```

Controlling Lint Messages

Like error and warning messages, you can control lint messages with the `+lint` option in the following ways:

- You can limit the number of occurrences of a lint message to a number you specify. For this, specify the message ID as an argument to the `+lint` option along with the specified maximum number of occurrences.

You can enter a maximum of 0 to disable any display of the message specified by the message ID, see [Example 2: Reducing the Number of lint Messages](#).

Note:

- Do not enter a maximum number of occurrences, even if 0, if also appending the `no` keyword to the message ID.
- Disable the display of all lint messages with the `none` argument to the `+lint` option.

- Enable the display of all lint messages with the `all` argument to the `+lint` option.

Note:

You cannot downgrade an error or warning condition and message to a lint condition and message.

Suppressing Lint, Warning, and Error Messages

The `-suppress` option suppresses lint, warning, and error messages. The `-suppress` option with no argument should suppress all warnings/lint and downgradable error messages

If you enter a message ID argument, and the message is downgradable, VCS does not display that message. You can enter the ID for any lint, warning, or downgradable error message.

The `-suppress` option gives you a message control option that takes a higher precedence when you enter more than one of these options: `-error`, `+warn`, or `+lint`. For more details, see [Using Message Control Options Together](#).

Note:

The `-error` option is also a runtime option.

Error Conditions and Messages That Cannot Be Disabled

Some error conditions always terminate compilation without creating an executable and cannot be controlled or suppressed by the `-error` or `-suppress` options.

- Syntax errors
- Fatal error messages, those from error conditions that immediately halt compilation

Using Message Control Options Together

If you are entering more than one of these message control options, you will need to know their precedence when used together. The order of precedence from highest to lowest is as follows:

1. The `-suppress` option with no arguments, suppresses all possible messages and cannot be overridden by another message control option.
2. The `none` argument has a higher precedence than specifying `all` or a message ID.
3. The order on the `vcs` command line

The following options and arguments have the same intrinsic precedence:

`-suppress=messageID`

<pre>-error=messageID:max +warn=messageID:max +lint=messageID:max</pre>	<pre>-error=all +warn=all +lint=all</pre>
-------------------------------------------------------------------------	-------------------------------------------

Because they have equal intrinsic precedence, the order on the `vcs` command line determines relative precedence. The first of these options on the command line has the least precedence and the last of these has the most.

Message Control Examples

The following examples show how to use these options:

Example 1: Reducing the Number of Warning Messages

If you have small SystemVerilog source file named as `diff_clk_wosvaext.sv` with the following content:

```

1 module top #(Pa = 1);
2 bit a, c, clk;
3 wand b1;
4 wand c1;
5
6 clocking cb2 @ (posedge clk);
7 endclocking
8
9 sequence S2();
10 @ (cb2) $past ($past (a,, $stable ($isunknown (1'bx), @ (negedge
11 clk)), @ (posedge clk)), , $sampled (a), @ (negedge clk));
12 endsequence
13
14 property P1();
15 @ (cb2, posedge clk iff ($stable (b1, @ (posedge clk)))
16 $stable ($past (b1,, ,@ (posedge clk)), @ (negedge clk)));
17 endproperty
18
19 A1: assume property (@(S2) S2 );
20 A2: assume property (@(S2) P1());
21 A3: assume property (@(cb2) disable iff ($stable (c1)) P1);
22 A4: assume property (@(cb2) disable iff ($sampled ($past (c1,, ,@ (clk)))
23 first_match (S2)));
24
25 sequence S3();
26 @ (cb2) S2() ##1 @ (negedge clk) $stable (b1 || $sampled (c1), @ (posedge
27 clk));
28 endsequence
29
30 A5: cover property (@(S2) S3);
31 initial begin
32 a = 1;
```

```

29 repeat (20)
30 #5 clk = !clk;
31 end
32 endmodule

```

If you compile the above system Verilog file with the following command,

```
vcs -sverilog diff_clk_wosvaext.sv
```

VCS displays the following warning messages:

```

Warning-[SVA-LCDNAWPSC] Lead and property/sequence clocks differ
diff_clk_wosvaext.sv, 17
top
  Leading clock of expression does not agree with property/sequence
  clock.
  Leading clock will be applied.
  property/sequence clock: S2
  leading clock: posedge clk

```

```

Warning-[SVA-LCDNAWPSC] Lead and property/sequence clocks differ
diff_clk_wosvaext.sv, 18
top
  Leading clock of expression does not agree with property/sequence
  clock.
  Leading clock will be applied.
  property/sequence clock: S2
  leading clock: top.cb2,posedge clk iff $stable(b1, @(posedge clk))

```

```

Warning-[SVA-LCDNAWPSC] Lead and property/sequence clocks differ
diff_clk_wosvaext.sv, 19
top
  Leading clock of expression does not agree with property/sequence
  clock.
  Leading clock will be applied.
  property/sequence clock: posedge clk
  leading clock: top.cb2,posedge clk iff $stable(b1, @(posedge clk))

```

```

Warning-[SVA-LCDNAWPSC] Lead and property/sequence clocks differ
diff_clk_wosvaext.sv, 26
top
  Leading clock of expression does not agree with property/sequence
  clock.
  Leading clock will be applied.
  property/sequence clock: S2
  leading clock: posedge clk

```

VCS displays the same warning four times, if you want to control the number of warning messages, you can use the compile-time option +warn=warn_ID:n...

For example,

```
vcs -sverilog +warn=SVA-LCDNAWPSC:1 diff_clk_wosvaext.sv
```

VCS limits the warning messages to one.

```
Warning-[SVA-LCDNAWPSC] Lead and property/sequence clocks differ
diff_clk_wosvaext.sv, 17
top
  Leading clock of expression does not agree with property/sequence
  clock.
  Leading clock will be applied.
  property/sequence clock: S2
  leading clock: posedge clk
```

Example 2: Reducing the Number of lint Messages

If you have small SystemVerilog source file named as `top.sv` with the following content:

```
1 `celldefine
2 module sub;
3 endmodule
4
5 `celldefine
6 module sub1;
7 endmodule
8
9 `celldefine
10 module top;
11 sub inst();
12 sub1 inst1();
13 endmodule
```

By default, all lint messages are disabled if you want to enable the lint message, you need to use the `+lint=lint_ID` compile-time option. For example:

```
vcs -sverilog +lint=CDUB top.sv
```

VCS displays the following lint messages during compilation:

```
Lint-[CDUB] Compiler directive unbalanced
top.sv, 1
  Unbalanced compiler directive is detected : `celldefine has no matching
  `endcelldefine.
  Please make sure that all directives are balanced.

Lint-[CDUB] Compiler directive unbalanced
top.sv, 5
  Unbalanced compiler directive is detected : `celldefine has no matching
  `endcelldefine.
  Please make sure that all directives are balanced.

Lint-[CDUB] Compiler directive unbalanced
top.sv, 9
  Unbalanced compiler directive is detected : `celldefine has no matching
  `endcelldefine.
  Please make sure that all directives are balanced.
```

If you want to control the number of lint messages printed in the compile time, you can use +lint=lint_ID:n... For example:

```
vcs -sverilog +lint=CDUB:1 top.sv
```

Now, VCS controls the number of lint messages printed to one:

```
Lint-[CDUB] Compiler directive unbalanced
top.sv, 1
  Unbalanced compiler directive is detected : `celldefine has no matching
  `endcelldefine.
  Please make sure that all directives are balanced
```

Example 3: Upgrading Multiple Warnings to One Error

Consider a Verilog file named *tfpic.v* with the following contents:

```
module top();
  wire a,b,c;
  child child_position_instance(a,b);
  child child_name_instance(.b(b));
endmodule

module child( input a, input b, input c);
endmodule
```

The module child has three input ports, but the module instantiation statements have only two or one port connection.

If you compile the vcs *tfpic.v* source file without message control, VCS displays the following during compilation:

```
Warning-[TFIPC] Too few instance port connections
  The following instance has fewer port connections than the module
  definition
  "tfipc.v", 3: child child_position_instance(a, b);

Warning-[TFIPC] Too few instance port connections
  The following instance has fewer port connections than the module
  definition
  "tfipc.v", 4: child child_name_instance( .b (b));

Warning-[TFIPC] Too few instance port connections
  The following instance has fewer port connections than the module
  definition
  "tfipc.v", 4: child child_name_instance( .b (b));
```

If you recompile specifying that message ID, *TFIPC* is upgraded to an error, and set to display this error message only once:

```
vcs tfpic.v -error=TFIPC:1
```

VCS displays the following error message only once:

```
Error-[TFIPC] Too few instance port connections
  The following instance has fewer port connections than the module
  definition
  "tfipc.v", 3: child child_position_instance(a, b);

1 error
```

Example 4: An Error Message That Cannot Be Controlled

Consider a Verilog file named `tfatc_err.v` with the following content:

```
module top;
  task wrFld4(input string fldName, input int bus = 0,input string
fldName2);
    $display("In wrFld4");
  endtask
  task wrFld4_2(input int bus = 0,input string fldName);
    $display("In wrFld4");
  endtask
  initial begin
    wrFld4(.bus(1));                                // this is line 9
    wrFld4(,1);                                     //                10
    wrFld4_2(.bus(1));                                //                11
  end
endmodule
```

The `wrFld4` task has three input ports and the `wrFld4_2` task has two input ports. However, the task enabling statements for them have only one connection.

VCS displays the following during compilation:

```
Error-[TFAFTC] Too few arguments to function/task call
tfatc_err.v, 9
"wrFld4(.bus(1));"
  The above function/task call is not done with sufficient arguments.

Error-[TFAFTC] Too few arguments to function/task call
tfatc_err.v, 10
"wrFld4(, 1);"
  The above function/task call is not done with sufficient arguments.

Error-[TFAFTC] Too few arguments to function/task call
tfatc_err.v, 10
top, "wrFld4(, 1);"
  The above function/task call is not done with sufficient arguments.

Error-[TFAFTC] Too few arguments to function/task call
tfatc_err.v, 11
top, "wrFld4_2(1);"
  The above function/task call is not done with sufficient arguments.
```

The error message with the ID TFAFTC is displayed four times. If you recompile while specifying that this error message gets displayed only once:

```
vcs tfatc_err.v -sverilog -error=TFAFTC:1
```

VCS displays the following:

```
Warning-[CSMC] Cannot set message count
  Failed to set display count for message id 'TFAFTC' because it cannot
  be
  suppressed.
  Specified count is ignored.
```

Parsing design file 'tfatc_err.v'

```
Error-[TFAFTC] Too few arguments to function/task call
tfatc_err.v, 9
"wrFld4(.bus(1));"
  The above function/task call is not done with sufficient arguments.
```

```
Error-[TFAFTC] Too few arguments to function/task call
tfatc_err.v, 10
"wrFld4(, 1);"
  The above function/task call is not done with sufficient arguments.
```

```
Error-[TFAFTC] Too few arguments to function/task call
tfatc_err.v, 10
top, "wrFld4(, 1);"
  The above function/task call is not done with sufficient arguments.
```

```
Error-[TFAFTC] Too few arguments to function/task call
tfatc_err.v, 11
top, "wrFld4_2(1);"
  The above function/task call is not done with sufficient arguments.
```

1 warning
 4 errors

None of the error messages are disabled and there is a warning saying that VCS cannot limit the display of the message.

Example 5: Syntax Using the -suppress Option

Consider a SystemVerilog file example.sv with the following content:

```
1 module top;
2 wire [5:0]data;
3 longint result,result1,result2,result3,result4;
4 assign data = 6'h2345;
5 initial
6 begin
7 result = $clog2(4294967296);      //2 ** 32
8 result4 = $clog2(2147483648);    //2 ** 31
```

```

9 result3 = $clog2(1073741824);      //2 ** 30
10 result1=2**16;
11 result2=result1*result1;
12 $display("clog: %0d result2 %0d \n",result,result2);
13 $display("clog3: %0d \n",result3);
14 $display("clog43: %0d \n",result4);
15 end
16 endmodule

```

If you compile this file as follows:

```
vcs -sverilog exmaple.sv
```

VCS displays the following warning messages:

```

Warning-[TMBIN] Too many bits in Based Number
example.sv, 4
  The specified width is '6' bits, actually got '16' bits.
  The offending number is : '2345'.

```

```

Warning-[DCTL] Decimal constant too large
example.sv, 7
  Decimal constant is too large to be handled in compilation.
  Absolute value 4294967296 should be smaller than 2147483648.

```

```

Warning-[DCTL] Decimal constant too large
example.sv, 8
  Decimal constant is too large to be handled in compilation.
  Absolute value 2147483648 should be smaller than 2147483648.

```

If you are using the `-suppress` option with the command line all warning messages are suppressed.

For example, if you use the following command:

```
vcs -sverilog -suppress example.sv
```

The `-suppress` option suppresses all warning/lint/downgradable error messages.

Controlling Error/Warning/Lint Messages Using a Configuration File

Using a configuration file, you can control lint, warning, and error messages that VCS displays according to the following:

- Source file name
- Module name
- Design subhierarchy

You control these messages with entries in a configuration file that you specify with the following compile-time option:

```
-msg_config=message_configuration_file_name
```

In this message configuration file, the basic rules are as follows:

- Each configuration entry is enclosed in braces or curly brackets { } ; for example:

```
{ +warn=noTFIPC;
  +file=$VCS_HOME/vmm.sv;
}
```

This entry specifies disabling the warning message with the `TFIPC` message ID about the content of the `vmm.sv` source file in the VCS installation.

- Each entry can have only one message operation *command*, beginning with one of the following keywords:

```
+lint +warn -error -suppress
```

- There can be multiple control conditions specified in the same entry, beginning with the following keywords:

```
+file +module +tree -file -module and -tree
```

- Message operation commands and control conditions that begin with + are for including something; those that begin with - are for excluding something.
- The message operation command and control conditions are separated with a semicolon ; or white space or return.
- Comments can be added to the file using the character types // and #. For example:
`// This is a single line comment. Multi-line comment /* */ is not supported.`
- Any + control condition, such as `+file`, cannot be used together with its corresponding - control condition, such as `-file`, in the same configuration entry.
- VCS reports an error condition if you specify conflicting control conditions for the same message ID.

This section consists of the following subsections:

- [Controlling Lint Messages](#)
- [Controlling Warning Messages](#)
- [Controlling Error Messages](#)
- [Upgrading Lint and Warning Messages to Error Messages](#)
- [Downgrading Error Messages to Warning Messages](#)

- [Suppressing All Types of Messages](#)
- [Enabling and Disabling by Source File](#)
- [Enabling and Disabling by Module Definition](#)
- [Enabling and Disabling by Subhierarchy](#)

Controlling Lint Messages

Lint messages are disabled by default so the lines in a configuration file enable their display.

To enable lint messages with a message operation command in a configuration entry that begins with `+lint=arguments`, use the following arguments:

```
+lint=all
```

To specify that the lines that follow enable the display of all lint messages.

```
+lint=ID1, ID2...
```

A comma separated list of lint message IDs to specify that you want to enable these specific lint messages, for example:

```
+lint=CDUB, NCEID
```

This list of IDs enables the display of the lint messages with `CDUB` and `NCEID` message IDs.

```
+lint=none
```

To specify that the lines that follow disable the display of all lint messages for a particular control condition in a configuration entry.

```
+lint=all, noID1, noID2...
```

A comma separated list of message IDs, each preceded by `no` with no space between `no` and the IDs, to disable these specified lint messages in a configuration entry.

Note the following about the `+lint` message operation command:

- It suppresses lint messages for the specified modules (see [Enabling and Disabling by Module Definition](#)) when you enter the `+lint=none` message operation command.
- It suppresses the specific lint messages for the specified modules when you enter the `+lint=noID` message operation command.

Controlling Warning Messages

To disable warning messages with the `+warn=arguments` message operation command, use the following arguments:

`+warn=none`

To specify that the lines that follow disable the display of all warning messages.

`+warn=noID1,noID2...`

A comma separated list of message IDs, each preceded by `no` with no space between `no` and the IDs, to specify that you want to disable these specific warning messages, for example:

`+warn=noMFACF,noCSMC`

This list of IDs disables the display of the warning messages with `MFACF` and `CSMC` message IDs.

Note the following about the `+warn` message operation command:

- It suppresses warning messages for the specified modules when you enter the `+warn=none` message operation command.
- It suppresses the specific warning messages for the specified modules when you enter the `+warn=noID` message operation command.

Controlling Error Messages

Error messages, like warning messages, are enabled by default. You can use the configuration file to do the following:

- Upgrade lint and warning messages to error messages.
- Downgrade applicable error messages to warning messages (not all error messages are downgradable).

Upgrading Lint and Warning Messages to Error Messages

To upgrade lint and warning messages to error messages, use the `-error=arguments` message operation command in the configuration entry. The arguments you can enter are as follows:

`-error=all`

Upgrades all enabled lint and warning messages to error messages.

`+lint=all -error=all`

Enables all lint messages, then upgrades all enabled lint and warning messages to error. The order of these options is important to get the messages with the appropriate severity

level. If you provide the options in opposite order as `+lint=all -error=all`, then all the lint messages will not be upgraded to the severity level of error.

`-error=ID1, ID2...`

A comma separated list of lint and warning message IDs to upgrade them to error messages, for example:

`-error=CDUB, MFACF`

This list of IDs upgrades the lint message with the ID of `CDUB` and the warning message with the ID of `MFACF` to error messages.

Downgrading Error Messages to Warning Messages

To downgrade error messages to warning messages, use a message operation command in the configuration entry that begins with:

`-error=noID1, noID2...`

The comma separated list is a list of error message IDs, preceded by the keyword `no`, for example:

`-error=noURMI, noETMFBC`

Not all error messages are downgradable. If you enter an error message ID for a non-downgradable error message, you receive a different error message indicating that it is not downgradable.

Note:

- You cannot downgrade all error messages to warning messages with the `-error=none` option.

Suppressing All Types of Messages

You can disable the display of all types of messages - such as informational, lint, warning, and error messages. For this, enter a line in the configuration file beginning with the `-suppress` or `-suppress=arguments` *message operation command except the error messages that cannot be downgraded*.

Note:

The `-suppress` message operation command cannot suppress non-downgradable error messages.

The arguments you can enter are as follows:

`-suppress` without an argument

Suppress all downgradable messages. This message operation command is the equivalent of the `-error=none` message operation command.

```
-suppress=ID1, ID2...
```

A comma separated list of message IDs to suppress specific lint, warning, or error messages, for example:

```
-suppress=CDUB, CSMC
```

This list of IDs suppresses the display of the lint message with the `CDUB` ID and the warning message with the `CSMC` IDs.

Enabling and Disabling by Source File

You can enable or disable lint, warning, and error messages for specific source files. For this, add the following control conditions to message operation command:

```
+file=source_file_list
```

`source_file_list` is a comma separated list of source files without spaces between them, for example:

```
+file=top.sv,introctr.sv,arbit.sv
```

This control condition specifies that the messages enabled in the preceding message operation command are enabled only for the source files named `top.sv`, `introctr.sv`, and `arbit.sv`.

```
-file=source_file_list
```

This control condition is similar to but opposite from `+file=source_file_list`. This control condition specifies the source files not affected by the message operation command.

Enabling and Disabling by Module Definition

You can enable or disable messages for specific module definitions. For this, add the following control conditions to a message operation command in a configuration entry:

```
+module=module_name_list
```

The module name list is a comma separated list of module names, for example:

```
+module=top,introctr,arbit
```

This control condition specifies that the messages enabled in the message operation command are enabled for the contents of the modules named `top`, `introctr`, and `arbit`.

```
-module=module_name_list
```

This control condition is similar to but opposite from `+module=module_name_list`. This control condition specifies the module definitions not affected by the message operation command.

Enabling and Disabling by Subhierarchy

Consider a scenario in which your design includes sub-hierarchies, such as in a Verilog library file that has a top-level module and module definitions hierarchically under it, or some other discrete set of module definitions in a hierarchy with a top-level module, such as in design re-use in a larger design. To enable or disable messages for these subhierarchies, specify the top-level module definition with the following control conditions:

```
+tree=module_name_list
```

The module name list is a comma separated list of top-level module names, for example:

```
+tree=introctr,arbit
```

This control condition specifies that the messages enabled in the message operation command are enabled for module definitions *introctr* and *arbit* and the module definitions hierarchically under them.

```
-tree=module_name_list
```

This control condition is similar to but opposite from `+tree=module_name_list`. This control condition specifies the subhierarchies not affected by the message operation command.

Extracting the Files Used in Elaboration/Compilation

To extract the Extensible Markup Language (XML) files, which are required to create the `top` module, use the `-metadump` compile-time option. Its syntax is as follows:

For two-step flow:

```
% vcs -metadump <design_top>
```

For three-step flow:

```
% vlogan <analyze_options> <source_file>
% vcs -metadump <design_top>
```

Using this syntax, you can generate the list of files required to create the top-level module and create `simv`.

The `-metadump` option generates XML files from which you can get information about all files with the file name and the information about the line number to resolve `simv`.

The arguments or sub-options for the `-metadump` option are as follows:

`-metadump [=hierarchy_name]`

Generates the file list used by the sub hierarchy tree.

`-metadump_only`

Exits the compilation without generating simv.

```
-top "top1+top2" -metadump
```

Supports multiple top modules.

```
-metadump_txt
```

Generates the text files instead of XML files.

The reporting files are in the XML file format. The verilogMetadata.xml file is extracted for the Verilog portion of the design and the vhdlMetadata.xml file is extracted for the VHDL portion of the design. These files can be accessed from the current working directory.

Note:

This section describes the feature in the context of the Unified Use Model (UUM) flow only. The two-step flow is not supported in this implementation.

This section discusses the following topics:

- [XML File Format](#)
- [Limitations](#)

XML File Format

This section describes the format of the XML document. Synopsys does not provide a parser for the XML file and it is suggested to choose to process the file in the way you want.

There are four main sections in the XML file as described in this section.

Section 1

```
<?xml version="1.0" encoding="ISO-8859-1" ?>
<opml version="1.0">
<head>
<title>VCS Dump File for Post Process</title>
<vcsVersion> H-2013.06-SP1 (ENG)</vcsVersion>
<dateCreated>Tue May 14 13:35:00 2013</dateCreated>
</head>
```

The top-level head section describes the basic statistical information about the file, such as the VCS version that is used and the date on which it is created. This information can be used to keep track of when the list of files was extracted.

Section 2

The following example provides the collection of files that are listed in the <filelist> section. These are the complete set of files that are used in the entire design.

```
<fileList>
<file fid="0" path="/remote/path1/directory1/Macro/m.h" />
<file fid="1" path="/remote/path2/directory2/y.h" />
<file fid="2" path="/remote/path1/directory1/Macro_n.h" />
<file fid="3" path="/remote/path1/directory1/Macro/x.h" />
<file fid="4" path="=/remote/path1/directory1/Macro/test.v" />
</fileList>
```

Each of the file `fid` point to a specific path on the file system from where it is picked up. This section also provides a list of all the files that make up `simv`.

In this example, `file fid = "1"` refers to the `/remote/path2/directory2/y.h` file on the file system. The `file fid="0"` refers to the `m.h` file in some other location. The list of files included in this section are both include files and elaboration files.

Section 3

This section provides the information about the files that are included by other files as understood by the VCS parser. It provides a list of include files that are used for elaboration of the design. Note that the list of include files are non-unique. If a file is included by many files, it is displayed as separate lines in this section of the XML file.

```
<!-- Include file list for the whole design -->
<includeFileList>
<incfile fid="0" lineno="10" includeID="3" />
<incfile fid="4" lineno="1" includeID="0" />
<incfile fid="4" lineno="2" includeID="2" />
<incfile fid="4" lineno="3" includeID="3" />
</includeFileList>
```

For example,

```
<incfile fid="0" lineno="10" includeID="3" />
```

is interpreted as follows:

`file fid="0", that is, /remote/path1/directory1/Macro/m.v includes file fid = "3", (/remote/path1/directory1/Macro/x.h) on line number 10.`

Each file picked up by the parser is reported as explained.

In this example, it is noted that `file fid="3"` (that is `/remote/path1/directory1/Macro/x.h`) is included by multiple files, and therefore, shows up multiple times in the list.

Section 4

This section is a unified unique list of files that are included in the design using the ``include`` directive from the previous section, which is a list of unique `includeIDs`.

This section displays only a subset of the list presented in Section 3. As mentioned earlier, `includedfile fid="3"` has been included multiple times by many files and can be seen in the XML entries in the previous section. However, it is reported once in this section.

```
<uniqIncludeFileList>
<includedfile fid="0" />
<includedfile fid="2" />
<includedfile fid="3" />
</uniqIncludeFileList>
```

Example

If the `top` module is specified as a design top in elaboration, VCS gathers `top.v` and `header.v` elaboration files. The `top.v` file is gathered as a non-include file and `header.v` is gathered as an include file. If these two files are provided as input to tools, such as `vlogan`, multiple-definition errors might occur.

```
top.v

`include "header.v"
module top;
bottom bot();
endmodule

header.v

module bottom;
endmodule
not_used.v

module not_used;
endmodule
```

VCS Command Line:

```
% vcs top.v
% vcs header.v
% vcs not_used.v
% vcs top -sverilog -metadump
% simv
```

VCS generates the `verilogMetadata.xml` file, which can be accessed from the current working directory and contains the following in `<body>`:

```
<fileList>

<file fid="0" path="/remote/xxxx/yyy/Documents/header.v" />
<file fid="1" path="/remote/xxxx/yyy/Documents/top.v" />

</fileList>
```

The `verilogMetadata.xml` file lists only `header.v` and `top.v`. However, it does not list the `not_used.v` file as it is not used in simulation.

Note:

For any tool that is capable of parsing the Verilog file and substitute the `include directive, non-include files are sufficient to work.

Example

vllogic.v

```
module vllogic(a,b,c,d,e);
  input a,b,c;
  output d,e;
  wire d,e;
  assign d = a ^ b ^ c;
  assign e = ((a&b) | (b&c) | (a&c));
endmodule
```

vltop.v

```
`include "vllogic.v"
module vltop;
  reg a,b,c;
  wire d,e;
  vhent vh1(.a(a), .b(b), .c(c), .d(d), .e(e));
  initial begin
    $monitor("a=%b b = %b c=%b d=%b e=%b", a,b,c,d,e,$time);
    #2 a = 'b1; b = 'b1; c= 'b0;
    #2 a = 'b0; b = 'b1; c= 'b1;
    #2 a = 'b1; b = 'b0; c= 'b0;
    #5 $finish;
  end
endmodule
```

vhnet.vhd

```
LIBRARY IEEE;
USE IEEE.STD_LOGIC_1164.ALL;
use ieee.std_logic_arith.all;
entity vhent is
  port(a,b,c: in std_logic;
       d,e: out std_logic);
end entity;
architecture structural of vhent is
  component vllogic
    port(a,b,c: in std_logic;
         d,e: out std_logic);
  end component;
begin
  vh1: vllogic port map(a,b,c,d,e);
end structural;
```

```
not_used.vhd

LIBRARY IEEE;
USE IEEE.STD_LOGIC_1164.ALL;
use ieee.std_logic_arith.all;

entity orGate is
    port(      A, B : in std_logic;
               F : out std_logic);
end orGate;

architecture func of orGate is
begin
    F <= A or B;
end func;
```

VCS Command Line:

```
% vhdlan not_used.vhd
% vlogan vlogic.v
% vlogan vttop.v
% vhdlan vhnet.vhd
% vcs vttop -metadump
% simv
```

VCS generates the `verilogMetadata.xml` **file for the Verilog code and the** `vhdlMetadata.xml` **file for the VHDL code, which can be accessed from the current working directory.**

The `vhdlMetadata.xml` file lists the following:

```
<fileList>
<file fid="0" path="/remote/xx/y/yyy/metadump1/vhnet.vhd" />
</fileList>
```

However, the XML file does not list the `not_used.vhd` file as it is not used in simulation.

Note:

To extract the text files instead of XML files, use the `-metadump_txt` compile-time option.

Limitations

The following are the limitations with the `-metadump` option:

- The order of elaboration files cannot be guaranteed. For example, in a three-step flow, if you swap file A and file B for `vlog`, one or more of the following can happen:
 - For the undefined macro references, syntax errors occur if file A uses a macro defined in file B, and file A does not include file B either directly or recursively.
 - A different default net type is picked up if file A uses a default net type defined in file B, and file A does not include file B either directly or recursively.
 - Error might occur in other related compiler directives.

Even with correct ordering of file A and file B, other problems might arise.

For example, in an elaboration, `top1` is named `design top`, and `top1.v` gets copied over to a new directory for insertion and the timescale of `top1.v` is `1ns/1ns`. In the original analyzed database (`AN.DB`), the timescale could either be `1ps/1ps` or `1ns/1ns` based on the order of analysis of `top1.v` and `top2.v`.

```
top1.v
`timescale 1ns/1ns
module top1;
  initial begin
    $display("At %t Hello world\n", $time);
  end
endmodule

top2.v
`timescale 1ps/1ps
module top2;
  initial begin
    $display("Hello world\n");
  end
endmodule
```

The correct ordering does not guarantee simulation of file copies and happens in the same manner as that of original files.

- Elaboration files do not include configuration files, `synopsys_sim.setup`, other non-Verilog files, and non-VHDL files.
- The `-metadump` option does not work with partition compile flow. The metadump file can be generated with single compile flow and the same file works in both single compile and partition compile flow.

Reducing Compile Time for Post-Process Only Debug

If you want to perform only post-process debug with an existing VPD file and recreate simv.daidir, you can use the `-static_dbgen_only` compile-time option and significantly reduce the compilation time.

Use Model

Perform the following steps:

1. Use the following syntax to compile your design file:

```
% vcs -debug_access+pp <other_vcs_options> file_name.v
```

2. Run the simulation to generate the VPD file.

```
% simv <simv_options>
```

3. Use the `-static_dbgen_only` compile-time option as follows to regenerate simv.daidir which is required to do post-process debug.

```
% vcs -static_dbgen_only -debug_access+pp <other_vcs_options>  
file_name.v
```

5

Simulating the Design

This chapter describes the following:

- [Using Verdi](#)
- [Using UCLI](#)
- [Reporting Forces/Injections in a Simulation](#)
- [Key Runtime Features](#)

As described in the section [Simulation](#), you can simulate your design in either interactive mode or batch mode. To simulate your design in interactive mode, you can use Verdi and UCLI. To simulate your design in batch mode, you must use UCLI. For more information, refer to the section entitled, [Batch Mode](#).

Using Verdi

Verdi provides you with a graphical user interface to debug your design. Using Verdi, you can debug the design in interactive mode or in post-processing mode. You must use the same version of VCS and Verdi to ensure problem-free debugging of your simulation.

In the interactive mode, apart from running the simulation, Verdi allows you to do the following:

- View waveforms
- Compare waveforms
- Trace drivers and loads
- View schematics and path schematics
- Execute UCLI/Tcl commands
- Set breakpoints (line, time, event, and so on)
- Line stepping

However, in post-processing mode, an FSDB file is created during simulation, and you use Verdi to:

- View waveforms
- Compare waveforms
- Trace drivers and loads
- View schematics and path schematics

Set `VERDI_HOME` and perform the following:

- Use the following command to invoke simulation in interactive mode using Verdi:

```
% simv -gui
```

- Use the following command to invoke Verdi in post-processing mode:

```
% verdi -ssf novas.fsdb -nologo
```

Note:

The interactive mode of Verdi is not supported when you are running VCS slave mode simulation.

For information on generating an FSDB dump file, see [Debugging with Verdi](#).

For more information on using Verdi, see *Verdi and Siloti Command Reference Guide* under Verdi documentation in SolvNetPlus.

Using UCLI

Unified Command Line Interface (UCLI) provides a common set of commands for interactive simulation. UCLI is the default command line interface for batch mode debugging in VCS.

UCLI commands are based on Tcl, therefore you can use any Tcl command with UCLI. You can also write Tcl procedures and execute them at the UCLI prompt. Using UCLI commands, you can do the following:

- Control simulation
- Dump the FSDB and VPD files
- Save/Restore the simulation state
- Force/Release a signal
- Debug the design using breakpoints, scope/thread information, and built-in macros

UCLI commands are built based on Tcl. Therefore, you can execute any Tcl command or procedures at the UCLI prompt. This provides you with more flexibility to debug the design in interactive mode. The following command starts the simulation from the UCLI prompt:

```
% simv [simv_options] -ucli
```

When you execute the above command, VCS takes you to the UCLI command prompt. To invoke UCLI, ensure that you specify the `-debug_access+r` option during compilation/elaboration. You can then use the `-ucli` option at runtime to enter the UCLI prompt at time 0 as follows:

```
% simv -ucli
ucli%
```

At the ucli prompt, you can execute any UCLI command to debug or run the simulation. You can also specify the list of required UCLI commands in a file, and source it to the UCLI prompt or specify the file as an argument to the runtime option, `-do`, as shown below:

```
% simv -ucli
ucli% source file.cmds

% simv -ucli -do file.cmds
```

Note:

- UCLI is not supported when you are running VCS slave mode simulation.
- You can use the `-ucli` option at runtime even if you have not used some form of the `-debug_access` option during compilation, but in this case only the `run` and `quit` UCLI commands are supported.

Note the following behavioral changes when UCLI is the default command-line interface:

- Command line options, such as `simv -i` or `-do`, only accept UCLI commands
- Interrupting the simulation using `Ctrl+C` takes you to the UCLI prompt by default for debugging your designs
- UCLI include file options (`-i` or `-do`) expect the next argument to be a UCLI script.

```
%> simv -ucli -i ucli_script.inc
```
- The `-R` feature in VCS continues to take you to the old CLI UI, unless you explicitly add `-ucli` to VCS command line.

-ucli2Proc Option

By default, UCLI runs in the simv process. There are a few scenarios which may require running UCLI in its own process, and this is enabled using the `-ucli2Proc` option:

- In SystemC designs, you must specify the `-ucli2Proc` command, if you want to call ‘cbug’ in batch mode (ucli). VCS issues a warning message if you do not specify this command
- When you issue a `restore` command inside a `-i/-do/source`, you must pass the `-ucli2Proc`. This situation is only applicable when there are commands following the `restore` commands that need to be executed in the `do` script
- Any usage of `start/restart/finish/checkpoint/config endofsim/reversedebug` from UCLI needs the `-ucli2Proc` command

For more information about UCLI, click the link *Unified Command-line Interface (UCLI)* if you are using the VCS Online Documentation.

If you are using the PDF interface, see *ucli_ug.pdf* to view the *UCLI User Guide*.

Options for Debugging Using Verdi and UCLI

`-debug_access`

Gives best performance with the ability to generate FSDB/VPD/VCD files for post-process debug. It is the recommended option for post-process debug.

`-debug_access (+<option>)`

Allows you to have more granular control over the debug capabilities in a simulation. The `-debug_access` option enables the dumping of the VPD and FSDB files for post-process debug.

You can specify additional options with the `-debug_access` option to selectively enable the required debug capabilities. You can optimize the simulation performance by enabling only the required debug capabilities.

For more information on the `-debug_access` option, see “Optimizing Simulation Performance for Desired Debug Visibility With the `-debug_access` Option” section.

`-debug_region (=<option>) (+<option>)`

Allows you to have better control over the performance of `-debug_access`. This option enables you to apply debugging capabilities to the desired portion of a design.

You must use the `-debug_region` option along with the `-debug_access` option at compile time. For more information on `-debug_region`, see “Optimizing Simulation Performance for Desired Debug Visibility With the `-debug_access` Option” section.

`+fsdbfile+filename`

FSDB dumping option. This option allows you to specify the FSDB file name. If this option is not specified, the default FSDB file name is `novas.fsdb`.

Example: `+fsdbfile+test1.fsdb`

`-vpd_file <file_name>`

Specifies the name of the generated VPD file. You can also use this option for post-processing where it specifies the name of the VPD file.

`+fsdb+dump_limit=size`

Specifies a size limit for the FSDB file. The default and minimum size limit is 10MB.

Example: `+fsdb+dump_limit=15`

`-vpd_fileswitchsize <size_in_MB>`

Specifies a size for the VPD file. When the VPD file reaches this size, VCS closes this file and opens a new one with the same size.

Reporting Forces/Injections in a Simulation

VCS provides the details of all the forces applied on your design during the simulation in a user-defined ASCII text file. This feature helps you to debug forces by allowing you to view all the forces that are effective in a simulation.

The following sections describe this feature in detail:

- [Use Model](#)
- [Reporting Force/Deposit/Release Information](#)
- [Generating Force List Report for Desired Instance Hierarchies](#)
- [Displaying Different ID for Each Verilog Force](#)
- [Displaying Source Information for VHDL Forces](#)
- [Displaying Source Information for External Forces](#)
- [Viewing Force Information in Interactive Debug Mode](#)

- [Reporting \\$deposit Value Changes](#)
 - [Limitations](#)
-

Use Model

Perform the following steps to use this feature:

- Use the `-force_list` option at compile time, as shown below, to allow the force reporting feature to record language forces/releases.

```
% vcs <debug_option> filename.v -force_list <other_vcs_options>
```

Where,

`<debug_option>`

Reporting external (PLI) forces and deposits requires a minimum of read and callback debug capability. This corresponds to `-debug_access+r+cbk`.

This step does not enable force reporting feature by itself. You must use `-force_list <filename>` at runtime, as shown in the following step. For more information, see [Table 9](#).

- Use the `-force_list` option at runtime, as shown below, to enable force reporting feature and generate an ASCII text file containing information about the forces/deposits/releases applied during the simulation in time order.

```
% simv -force_list <filename>
```

Where, `filename` is the user-defined ASCII file name. It can be relative path or absolute path. Compression is disabled by default. Use the `-force_list_compress` option at runtime to compress the resulting log file with the gzip compression. The log file is saved with the same name, but changes its filename extension by appending `.gz` at the end of it.

For example, for the following command:

```
% simv -force_list report.log -force_list_compress
```

the output file is: `report.log.gz`

Use gunzip to uncompress a force list file. For example, uncompress the above output file as follows:

```
gunzip report.log.gz
```

This results in the original file `report.log` which is uncompressed.

Key Points to Note

- If you use the `-force_list` option at runtime, but not at compile time, only external forces are logged
- If you use the `-force_list` option at both compile time and runtime, then both language forces and external forces are logged
- For mixed signal designs, the `-force_list` option captures the force events on the VHDL/VHPI/hdl_xmr targets. Pure VHDL designs are not supported
- [Table 9](#) describes the usage of the `-force_list` option in detail.

Table 9 Usage of the `-force_list` Option

-force_list at compile time	-force_list at runtime	Language forces logged	External forces logged
No/Yes	No	No	No
No	Yes	No	Yes
Yes	Yes	Yes	Yes

Reporting Force/Deposit/Release Information

The ASCII text file consists of the following parts:

- A header section that includes the information given in [Table 10](#), associated with an ASCII character ID that is 1 to 4 characters long. For more information, see [Header Section](#).
- A time order sorted list of force/release/deposits, as they occur during the simulation, indexed by the ID shown in the header section. For more information, see [Event List Section](#).

Table 10 Force Capture and Log Information

Force type	Time	Instance name of the target node	Module name where force occurred	File / Line data logged	Value
Language force/release/deposit	Simulation time when the node was forced or released.	Hierarchical node name being forced.Example: top.test.child2.a	Name of the module.Example: top	Full path of the file where the force statement occurs, and the line number of the statement in the source file. For example:/home/work/test.v:1234	Value represented as binary except for int, real, and string types.Binary value is prefixed with 'b.Release will not have values.
VPI/ACC/UCLI force/release/deposit	Simulation time when the node was forced or released.	Hierarchical node name being forced.	Not Applicable	Not Applicable	Value represented as binary except for int, real, and string types.Binary value is prefixed with 'b.Release will not have values.

Handling Forces on Bit/Part Select and MDA Word

If the target of the force is bit-select, part-select, or mda word, the appropriate indices is included in the target node name, for example, as follows:

Bit select	top.a.b[2]
Part select	top.a.b[0:3]
MDA bit select	top.c.d[2][3][4]
MDA part select	top.c.d.[2][3][1:4]

Forces recorded can be any object supported by language and PLI forces.

Expressions are evaluated only if they contain constants and/or parameters. For example, `top.a[(1+paramb)*2]` is evaluated to determine the resulting constant index.

Expressions are not evaluated if they contain variables. For example, consider the following code. In this case, only the base vector is captured.

```
logic [0:9] top.a;
  for (i= 0; i < 10; i++)
    begin
      force top.a[i] = 1; // captures the entire vector for top.a and
      not a select of top.a
    end
  end
```

Handling Forces on Concatenated Codes

Forces consisting of more than one signal on a signal line are split up on signal basis.

For example, `force {a,b} = 2'b11` results in two header entries, one for `a` and one for `b`. Both `a` and `b` display the same file/line number, but carry different IDs since they are different nodes.

Output Format

The ASCII text file output consists of the following two sections:

- [Header Section](#)
- [Event List Section](#)

Header Section

The header section contains mapping between forced object list and unique ASCII ID. This section is divided into two parts: Language Forces and External Forces.

Language forces are unique by statement, whereas external forces are unique by node. Multiple language forces on the same node from different lines result in multiple header entries for that node.

Multiple external forces result in one external force header entry for that node. Nodes with both language and external forces have entries in both Language Forces and External Forces parts.

For a unique node, only single ID is used for all entries in header for that particular node.

If VCS finds unsupported force/release event, it labels such event with a reason.

Following is the display format for Language Forces and External Forces:

Language Forces:

ID	Target	Module	File:Line
----	--------	--------	-----------

External Forces (VPI/ACC/UCLI):

ID	Target
----	--------

Header Example

Header Section

Language Forces

ID	Target	Module	File Line
1	top.child1.a	top	/home/user/top.v:10 ** NO_VALUE_CHANGE full mda **

```

2      top.child1.child2.foo    child   /home/user/child.v:125
3      top.child1.a_real     child   /home/user/child.v:127

External Forces

ID      Target

4      top.child1.a_int
5      top.child1.b[0:3]
6      top.child1.b[2]

```

Event List Section

This section displays the following information:

- Time during which the value is forced
- ID from the header
- Type of the force
- Value of force or deposit

Release value is not displayed. [Table 11](#) lists the phrases of the acronyms used in the Event List section.

Table 11 List of Acronyms Displayed in the Event List Section

Acronym	Phrase
LF	Language Force
LR	Language Release
LD	Language Deposit/write
EF	External Force
ER	External Release
ED	External Deposit

Event List Example

```

ID      Type    Value
----  Time:  1  ---
2      LF      'b0
5      EF      'b1111
6      EF      'b1
4      ED      25
----  Time:  2  ---
3      EF      2.14

```

```

3      ER
5      LR
5      LD      'b0110
---    Time:  3  ---

```

The `Value` column displays the integer and real values as decimal values, strings as ASCII characters, and rest of them as binary values prefixed with '`b`'. Long value strings are line-wrapped.

Usage Examples

Example-1

Consider the following testcase `test.v` and the `test.ucli` file which contains UCLI forces.

Example 14 Design Testcase test.v

```

module top;

reg clk,rst,d;
wire q;

DUT dut (clk,rst,d,q);

always #1 clk = ~clk;

initial begin
  clk = 0 ; rst =0 ; d=0 ;

  #5 force rst = 1;
  #5 release rst ;
  #10 force dut.q =1 ;
  #10 release dut.q ;
  #100 $finish ;
end
endmodule

module DUT (clk,rst,d,q);
input clk,rst,d;
output q;
wire q;
reg q_reg;

assign q = q_reg;

always @ (posedge clk)
if (rst) begin
  q_reg <= 0;
end else begin
  q_reg <= d;
end

```

```
endmodule
```

Example 15 test.ucli

```
run 30
force top.dut.q 1
release top.dut.q
run
```

Compile the `test.v` code, as follows:

```
% vcs -debug_access+r+cbk -sverilog -force_list test.v
```

Run the simulation, as follows:

```
% simv -ucli -i test.ucli -force_list report.log
```

Use the following command to view the `report.log` file:

```
% cat report.log
```

Below is the content of the `report.log` file:

```
VCS Force List
    Header Section

    Language Forces

    ID Target      Module      File Line
    1 top.rst      top force.v 13
    1 top.rst      top force.v 14
    2 top.dut.q   top force.v 15
    2 top.dut.q   top force.v 16

    External Forces

    ID      Target
    2 top.dut.q

    Event List Section

    ---- Time: 0 ----
    ---- Time: 5 ----
    1 LF 'b1
    ---- Time: 10 ----
    1 LR
    ---- Time: 20 ----
    2 LF 'b1
    ---- Time: 30 ----
    2 EF 'b1
    2 LF 'b1
```

```
2 ER
2 LR
2 LR
```

Example-2

This example describes the usage of `-force_list` to capture force events on the `hdl_xmr` targets.

Consider the following files:

Example 16 Testcase test_1.v

```
`timescale 1 ns/1 ns
module ex1 (main_bus bus);

reg reg_hdlxmr;
reg [7:0] reg_hdlxmr_vec;

endmodule
```

Example 17 Testcase test_2.v

```
module vtop (wa,wb,wc);

inout wa,wb;
input wc;
main_bus bus (.wa(wa), .wb(wb), .controll1(wc));

ex1 ex11 (.bus(bus));

reg source_inter_reg;
byte vbytel=8'd1,vbyte2=8'd1;
shortint vshort1=16'd1,vshort2=16'd1;
int vint1=32'd1,vint2=32'd1;
longint vlong1=64'd1,vlong2=64'd1;
bit[0:7] vbit1,vbit2;
bit[7:0] vbit3,vbit4;
logic[0:7] vlogic1=8'd1,vlogic2=8'd1;
logic[7:0] vlogic3=8'd1,vlogic4=8'd1;

logic l1=1;

initial
begin
# 10 ;
bus.interfacebyte2 = 8'd10 ;
bus.interfaceshort2 = 16'd10 ;
bus.interfacelong2 = 64'd10 ;
bus.interfacebit4 = 8'd10 ;
bus.interfacebit3 = 8'd10 ;
bus.interfaceint1 = 5 ;
bus.interfaceint2 = 5 ;
bus.interfacelogic3 = 8'd10 ;
```

```
    end
endmodule
```

Example 18 Testcase *interface_datatypes.v*

```
typedef enum {red,blue,green,yellow} color1;
interface main_bus (inout wa, wb, input control1);

reg interface_reg;
byte interfacebyte1 ,interfacebyte2 ;
shortint interfaceshort1 ,interfaceshort2 ;
integer interfaceint1 ;
int interfaceint2 ;
longint interfacelong1 , interfacelong2 ;
bit[0:7] interfacebit1 ,interfacebit2 ;
bit[7:0] interfacebit3 , interfacebit4 ;
logic[0:7] interfacelogic1 ,interfacelogic2 ;
logic[7:0] interfacelogic3 ,interfacelogic4 ;
logic interfacell;
logic[0:1] interfacef1,interfacef2;
color1 intfcolor1 , intfcolor2 ;
modport driver (inout wa,
                inout wb,
                input control1);

modport slave (inout wa,
                inout wb,
                input control1);

endinterface
```

Example 19 Testcase *test.vhd*

```
LIBRARY IEEE;
USE STD.textio.all;
USE IEEE.STD_LOGIC_TEXTIO.all;
USE IEEE.std_logic_1164.ALL;
Library Synopsys;
USE Synopsys.hdl_xmr_pkg.all;

entity ex2_vtop  is
port (wa,wb:INOUT std_logic;control1:in std_logic);
end entity;

architecture arch of ex2_vtop is
component vtop is
port (wa,wb:INOUT std_logic;wc:in std_logic);
end component;
signal w_vector:std_logic_vector(0 to 10);
signal wire_wa,wire_wb,wire_control:std_logic:='0';
signal vhd_sig,vhd_sig1:std_ulogic_vector(0 to 7);
type color_vhd is (red,blue,green,yellow);
```

Chapter 5: Simulating the Design
Reporting Forces/Injections in a Simulation

```

signal vhd_enum,vhd_enum1:color_vhd;
signal vhbytel,vhbyte:std_logic_vector(0 to 7) ;
signal vhshort1,vhshort2:std_logic_vector(0 to 15);
signal vhlond1,vhlong2:std_logic_vector(0 to 63);
signal vhbit1,vhbit2:std_logic_vector(7 downto 0);
signal vhd_sig3,vhd_sig4:std_logic_vector(7 downto 0);
signal vhint1,vhint2:integer;
signal vhbit3,vhbit4:bit_vector(7 downto 0);
signal vh_ustd3,vh_ustd4:std_ulogic_vector(7 downto 0);

begin

--Source interface signals
hdl_xmr(":ex2_vhtop:g1:bus:interfacebyte2","vhbytel",1);
hdl_xmr(".ex2_vhtop.g1.bus.interfaceshort2","vhshort1",1);
hdl_xmr(".ex2_vhtop.g1.bus.interfacelong2","vhlond1",1);
hdl_xmr(".ex2_vhtop.g1.bus.interfacebit4","vhd_sig4",1);
hdl_xmr(".ex2_vhtop.g1.bus.interfacebit3","vhbit3",1);
hdl_xmr(".ex2_vhtop.g1.bus.interfaceint1","vhint1",1);
hdl_xmr(".ex2_vhtop.g1.bus.interfaceint2","vhint2",1);
hdl_xmr(".ex2_vhtop.g1.bus.interfacelogic3","vh_ustd3",1);

g1:vtop port map(wire_wa,wire_wb,wire_control);
wa <= wire_wa;
wb <= wire_wb;

end arch;
```

Execute the following elaboration commands:

```
% vlogan -sverilog interface_datatypes.v
% vlogan -sverilog test_1.v
% vlogan -sverilog test_2.v
% vhdlan test.vhd
% vcs ex2_vhtop -debug_access+r+cbk
```

Execute the following simulation command:

```
% simv -force_list force.log
```

The following output (force.log) shows the force events on the hdl_xmr targets:

```
VCS Force List
Header Section

External Forces

ID      Target

1 /EX2_VHTOP/VHBYTE1
2 /EX2_VHTOP/VHSHORT1
3 /EX2_VHTOP/VHLONG1
```

Generating Force List Report for Desired Instance Hierarchies

The `-force_list` option allows you to generate force list report for the desired instance hierarchies in the design. This feature helps you to get the force list report for the selected hierarchies.

Use the following option at runtime to generate force list report for the desired instance hierarchies in the design:

-force list hier <list file>

Where, `<list_file>` is the configuration file. This file allows you to specify the hierarchy and levels of hierarchy to be dumped in the resultant `-force list` output file.

Note:

To use this feature, you must first compile your designs with the `-force_list` option.

Following is the syntax of the `<list file>` statements:

+/-<instance> <depth>

Where,

+<instance>

Fully rooted path name to a Verilog instance. Dumps the forces on the signals in the instance tree to the report.

<depth> for +<instance>

Specify the level/levels of hierarchy to be dumped. For example, <depth> of 1 means only the signals in <instance> are dumped. <depth> of 0 means all levels of sub-instances are dumped.

-<instance>

The forces on signals in the instance tree are filtered out. They are not dumped to the report.

<depth> for -<instance>

Specify the level/levels of hierarchy to be filtered out. For example, <depth> of 1 means only the signals in <instance> are filtered out. <depth> of 0 means all levels of sub-instances are filtered out.

Examples

The following are the examples:

- [Example-1: Generating force list report for the desired instance hierarchies](#)
- [Example-2: Reporting \\$deposit value changes](#)

Example-1: Generating force list report for the desired instance hierarchies

Example 20 test.v

```
module tb();
  reg r;
  wire oA, oB;
  dut dut(r, oA, oB);
  initial begin
    r = 0;
    #10 force tb.dut.instA.oA = 1;
    #10 force tb.dut.instA.instB.oB = 1;
  end
endmodule

module dut(input r, output oA, oB);
  A instA(r, oA, oB);
endmodule

module A(input r, output oA, oB);
  assign oA = r;

```

```
B instB(r, oB);
endmodule

module B(input r, output oB);
    assign oB = r;
endmodule
```

Example 21 hierarchyConfig.cfg

```
+tb.dut.instA 1
```

Compile test.v as follows:

```
% vcs -debug_access -sverilog -force_list test.v
```

Simulation command:

```
% simv -force_list=uniqueID forceReport.txt -force_list_hier
hierarchyConfig.cfg
```

Following is the forceReport.txt output:

```
VCS Force List
Header Section

Language Forces

ID Target Module File Line

1 tb.dut.instA.oA tb test.v 7

Event List Section

---- Time: 10 ----
1 LF 'b1
```

Example-2: Reporting \$deposit value changes

Example 22 test.v

```
module mymod(o,i,phi);
    output o;
    input i,phi;

    and(o,i,phi);

    initial begin
        $deposit(o,1'b1);
    end
endmodule

module test;
    reg i,phi;
```

```

  wire o;

  mymod m1(o,i,phi);

  initial begin
    i=0; phi=0;
    #5 i=1;
    #5; phi=1;
    #5; phi=0;
    #10; $finish;
  end
endmodule

```

Compile test.v as follows:

```
% vcs -debug_access -sverilog -force_list test.v
```

Simulation command:

```
% simv -force_list=uniqueID forceReport.txt
```

Following is the frc.txt output:

```

VCS Force List
  Header Section

  Language Forces

  ID  Target    Module    File Line
  1  test.m1.o  mymod   prim.v  8

  Event List Section

---- Time: 0 ----
1  LD 'b1

```

Displaying Different ID for Each Verilog Force

The force list feature provides separate IDs for each source of a Verilog force, release, or deposit in the Language Forces section of the force list file.

Enhanced Force List Report

The enhanced force list report looks as follows:

- VCS displays a unique ID for each Verilog force in the Language Forces section.
- The unique ID is used in the Event List Section.

Following is the sample report:

```
VCS Force List
  Header Section

  Language Forces

  ID  Target    Module      File Line

  1  test.a  test t.v 3
  2  test.a  test t.v 4

  Event List Section

  ---- Time: 0 ----
  ---- Time: 1 ----
  1  LF 'b0
  ---- Time: 2 ----
  2  LF 'b1
```

Use Model

To enable this feature, perform the following:

- At compile-time, use the following options:
 - The `-force_list` option. This option enables the force list feature
 - Debug capability `-debug_access+r+cbk` or higher
 - The `-sverilog` option, otherwise old Verilog style force may be missing
- At runtime, use the following option to enable the new enhancements:

`-force_list=uniqueID`

Displaying Source Information for VHDL Forces

The force list report is enhanced to mark the VHDL force/release with the file name and line number of the corresponding force/release assignment statement. For example, see the following report:

```
VCS Force List
  Header Section

  VHDL Language Forces

  ID  Target    File           Line

  1  /top/et/S_1  forceRelease.vhd  23
  2  /top/et/S_1  forceRelease.vhd  25
```

```
3 /top/et/s_2      forceRelease.vhd    28
```

Event List Section

```
---- Time: 0 ----
1 LF 'b1
---- Time: 1 ----
2 LR
---- Time: 2 ----
3 LF 2
---- Time: 3 ----
4 LR
5 LF 32
```

Use Model

To enable this feature, perform the following:

- At compile-time, use the following options:
 - The `-force_list` option. This option enables the force list feature
 - Debug capability `-debug_access+r+cbk` or higher
 - The `-sverilog` option, otherwise old Verilog style force may be missing
- At runtime, use the following option to enable the new enhancements:

`-force_list=uniqueID`

Displaying Source Information for External Forces

VCS provides detailed information about the origin of an external force, release, or deposit in the force list report.

The force list report is enhanced to mark an external force, release, or deposit with the file name and line number of the corresponding force, release, or deposit assignment statement.

[Table 12](#) describes this enhancement in detail.

Table 12 Displaying Source Information for External Forces

	Origin of the force, release, or deposit	Description
1	<code>hdl_xmr_force</code> , <code>hdl_xmr_release</code> , <code>hdl_xmr</code>	Force, release, or deposit that originates from <code>hdl_xmr*</code> procedures is marked with the Verilog/VHDL source file name and line number calling the <code>hdl_xmr</code> procedure, if one exists.

Table 12 Displaying Source Information for External Forces (Continued)

	Origin of the force, release, or deposit	Description
		Force, release, or deposit that originates from sourcing a file with the UCLI <code>Tcl do</code> command is marked with the TCL source file name and line number. There is no file name and line number information when the <code>Tcl source</code> command is used.
2	UCLI	Force, release, or deposit originating from an interactive command is marked as <code>INTERACTIVE</code> , and no file name and line number is given.
		Force, release, or deposit originating from <code>mhpi_ucliTclExec</code> is marked as <code>UCLI_FROM_MHPI</code> , and no file name and line number is given.
		No file name and line number is given for the <code>force</code> command executed in the UCLI <code>stop -command</code> script.
		Third-party tools and other PLI applications are typically not run from a design file name/line number. Third-party tools are run from a VPI/VHPI synchronization callback. Therefore, there is no design file name/line number information.
3	VPI, VHPI, MHPI, VCSD, and Third-party tools	Force, release, or deposit originating from the user-defined system function activation is marked with the design file name and line number for the activation, if one exists.
		If a force, release, or deposit originates from a simulation callback, no file name and line number is given.

Enhanced Force List Report

Following is the sample report:

External Forces

ID Target

```
1 top.tsanadigmld.vccsa    t.v, 11
2 top.tsanadigmld.vccst    t.v, 12
3 top.tsanadigmld.vcc_ts   t.v, 13
```

Following is the output when the UCLI force is used:

```
top.tsanadigmld.vccsa    my.tcl, 11
```

Following is the output when the VPI force is used in a third-party PLI called with the `$user_pli` user-defined task in the `testbench.v` Verilog file:

```
top.tsanadigmld.vccsa testbench.v, 11
```

Use Model

To enable this feature, perform the following:

- At compile-time, use the following options:
 - The `-force_list` option. This option enables the force list feature
 - Debug capability `-debug_access+r+cbk` or higher
 - The `-sverilog` option, otherwise old Verilog style force may be missing
- At runtime, use the following option to enable the new enhancements:

`-force_list=uniqueID`

Viewing Force Information in Interactive Debug Mode

VCS allows you to view all active forces at the current simulation time in interactive debug mode using the `find_forces` UCLI command.

Use model

Compile the design as follows:

```
% vcs -debug_access+r+cbk -sverilog -force_list test.v
```

Run the simulation as follows:

```
% simv -force_list -ucli
```

Use the `find_forces` UCLI command at the `ucli` prompt. This command reports the current active forces in the design or scope.

```
ucli% find_forces <option_name>
```

Following is the use model of the `find_forces` UCLI command:

```
find_forces <nid>
find_forces -scope <scope_name> [-level <level_number>] [-file <file_name>]
```

Where,

- `nid` is a nested identifier (hierarchical path name).
- `-scope` allows you to specify the full scope path name to find force.
- `-level` allows you to specify the level of hierarchy to do force search. Default value is 0 (search all sub-scopes).
- `-file <file_name>`: Specify a file to report the result. By default, report is generated as UCLI output.

Examples:

- `ucli% find_forces -scope tb`
Searches tb scope only
- `ucli% find_forces -scope tb/DUT/* -level 3`
Searches DUT and additional 2 levels under DUT
- `ucli% find_forces tb/DUT/clk_i_node`
Specifies whether the node clk_i_node is forced
- `ucli% find_forces -scope tb/DUT/* -level 0 file.txt`
Searches all levels under DUT and writes to file.txt.

Note:

- You must specify a signal name when `-scope` is not specified. Otherwise, VCS issues an error message.
- A force is reported only if it is forced. If the force start time (for example, using `hdl_xmr_force`) is beyond current time, but the actual command was already executed, it will not be reported in the force list.

Example

Consider the following files:

Example 23 test.sv

```
module test;
  reg [0:3] a;
  reg clk;
  wire aa,b;
  assign aa=a[1];

  dut dt (clk,aa,b);
  initial
```

Chapter 5: Simulating the Design
 Reporting Forces/Injections in a Simulation

```

begin
    clk=0;
    a = 4'b1010;
    #10 a= 4'b1110;
    #10 force a[0] =1'bX;
    #10 release a[0];
    #100 $finish;
end
initial begin
    forever #1 clk = ~clk;
end
endmodule
module top;
reg [0:3] a;
reg clk;
wire aa,b;
assign aa=a[1];

dut dt (clk,aa,b);
endmodule
module dut(input clk,a,output b);
    assign b = a;
endmodule

```

Example 24 frc.tcl

```

run 1
catch {find_forces -scope test} err
puts $err
force test.a 3'b1010
catch {find_forces -scope test -level 0} err
puts $err
run 1
force test.dt.a 3'b0011
run 1
catch {find_forces -scope test.dt -file frc.txt} err
puts $err
exec cat frc.txt
run
quit

```

Execute the following commands:

```
%vcs -debug_access+r+cbk+f -sverilog -force_list test.sv
%./simv -force_list -ucli -i frc.tcl
```

Output:

```

ucli% run 1
1 s
ucli% catch {find_forces -scope test} err
0

```

```

ucli% puts $err

ucli% force test.a 3'b1010
ucli% catch {find_forces -scope test -level 0} err
0
ucli% puts $err
{test.a[0]} freeze 'b1 Design
{test.a} freeze 'b1010 External

ucli% run 1
2 s
ucli% force test.dt.a 3'b0011
ucli% run 1
3 s
ucli% catch {find_forces -scope test.dt -file frc.txt} err
0
ucli% puts $err
{test.dt.a} freeze 'b1 External
Result successfully reported to: frc.txt
ucli% exec cat frc.txt
{test.dt.a} freeze 'b1 External
ucli% run

```

Reporting \$deposit Value Changes

The `-force_list` option dumps only the `$deposit` value change events in the force list report. It does not dump regular value changes caused by the design activity. A unique ID is assigned for each `$deposit` statement in the force list report. This feature helps you to identify value changes caused by `$deposit`.

Limitations

- This feature is not supported for pure VHDL designs.
- The `-R` option is not supported.
- Language forces from encrypted code are not reported.
- Nodes that are forced by more than one line in the Verilog source are not analyzed for the exact line that is driving the value. If a force/release event occurs on the node, VCS records the event, but not the file and line that exactly caused the event. VCS only lists the possible force drivers, not the exact driver.
- Force on entire mda is not supported for an event list capture. For example, consider the following code:

```

logic [1:0] foo [1:0][1:0] = 8'b11111111;
logic [1:0] baz [1:0][1:0];
initial begin

```

```
#1 force baz = foo;      //force entire mda baz
end
```

VCS does not record this value change on the force at #1, it just captures the header information for this force. VCS provides a comment in the header saying that value changes for this force event will not be recorded.

- Values in the event list for the language forces represent the current value at the time of the force, and not necessarily the forced value. This becomes an issue when multiple force statements influence one or more but not all of the same bits in the part selects and full vectors. As an example of this limitation, consider the following code:

```
logic foo [0:3] = 4'b0000;
Initial begin
    #0 force foo[0] = 1;           //creates force list header item 1 for the
                                    //force on bit select of foo[0].
    #1 force foo[0:2] = 'b111; //creates force list header item2 for the
                                //force on part select foo[0:2], note bit 0.
    #1 release foo[0:2] = 'b000; //creates a new header entry using the
                                //same id as the above line.
    #1 foo[0:2] = 'b000;          //reset bits
    #1 force foo[0] = 1; //creates a new header entry using same id as
                        //the first force on this node.
end
```

Following is the output report for the above code. Comments are added to the following report for illustration purposes only.

```
Language Forces
ID Target Module File Line
1 top.foo[0] top test.v 11
2 top.foo[0:2] top test.v 12
2 top.foo[0:2] top test.v 17
1 top.foo[0] top test.v 18
Event List Section
---- Time: 0 ----
2 LF 'b100      //at time 0, force occurs on id 1, so values for bits
1 and 2 on id2 are not forced.
1 LF 'b1        //this is the real force at this time, but forcelist
cannot know whether the force is from id 1 or id 2.
---- Time: 1 ----
2 LF 'b111     // this is the real force, no issue here because all
values are forced.
1 LF 'b1
---- Time: 2 ----
2 LR           // release fires, this is ok.
---- Time: 4 ----
2 LF 'b100     //similar to time 0, bits 1 and 2 are not forced but
reported.
1 LF 'b1       //this is the real force.
```

Key Runtime Features

Key runtime features includes:

- Overriding Generics at Runtime
 - Passing Values from the Runtime Command Line
 - Saving and Restarting the Simulation
 - Specifying Long Time Before Stopping the Simulation
 - Using -f Runtime Option
 - Resolving RTL Simulation Races in Mixed HDL Designs
 - Resolving RTL Simulation Races in Verilog Designs
 - Preventing Time 0 Race Conditions
 - Supporting Simulation Executable to Return Non-Zero Value on Error Results
 - Supporting Memory Load and Dump Task Verbosity
-

Overriding Generics at Runtime

Using the `-g`, `-gen` or `-generics` runtime option, you can change the following types of VHDL generics at runtime:

- Any generic that stays in VHDL and is not propagated directly or indirectly into Verilog
- Any generic that does not shape the tree or define the widths of ports through mixed HDL boundary
- Generics like delays, file names, and timing checks control

The use model is as follows:

```
% simv -g generics_file
```

The `-g`, `-gen` or `-generics` option takes a command file as an argument. You must specify the hierarchical path of the generic and a new value to override. A sample `generics_file` is as follows:

```
% cat generics_file

assign 1 /TOP/LEN
assign "OK.dat" /TOP/G1/vhdl1/FILE_NAME
assign (4 ns) /TOP/G1/VHDL1/delay
assign 16 /TOP/width
assign 4 /TOP/add_width
```

Use Model

Analysis

```
% vlogan [vlogan_options] file1.v file2.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

Note:

Specify the VHDL bottommost entity first, then move up in order.

Elaboration

```
% vcs [vcs_options] top_cfg/entity/config
```

Simulation

```
% simv [sim_options] -g cmd.file
```

Example

Consider the following example:

```
--spmem.vhd---

LIBRARY ieee;
USE ieee.std_logic_1164.ALL;
USE ieee.std_logic_signed.All;

ENTITY spmem IS
generic (
    add_width : integer := 3;
    delay : time := 2 ns;
    file_name : string := "empty.dat";
    WIDTH : integer := 8);
PORT (
    clk      : IN std_logic;
    reset    : IN std_logic;
    add     : IN std_logic_vector(add_width -1 downto 0);
    Data_In : IN std_logic_vector(WIDTH -1 DOWNTO 0);
    Data_Out : OUT std_logic_vector(WIDTH -1 DOWNTO 0);
    WR      : IN std_logic);
END spmem;

ARCHITECTURE spmem_v1 OF spmem IS

TYPE data_array IS ARRAY (integer range <>) OF
    std_logic_vector(7 DOWNTO 0);

SIGNAL data : data_array(0 to (2** add_width) );
BEGIN -- spmem_v1
PROCESS (clk, reset)
```

Chapter 5: Simulating the Design

Key Runtime Features

```

BEGIN -- PROCESS

    IF (reset = '0') THEN
        data_out <= (OTHERS => 'Z');

    ELSIF clk'event AND clk = '1' THEN
        IF (WR = '0') THEN
            data(conv_integer(addr)) <= data_in after delay;
        END IF;
        data_out <= data(conv_integer(addr));
    END IF;

END PROCESS;

END spmem_v1;

--TOP.vhd---

library IEEE;
use IEEE.std_logic_1164.all;

entity top is
    generic ( add_width : integer := 3;
              delay : time := 2 ns;
              file_name : string := "empty.dat";
              WIDTH : integer := 8;
              LEN : integer := 1 );
    PORT (
        clk      : IN std_logic;
        reset    : IN std_logic;
        add      : IN std_logic_vector(add_width -1 downto 0);
        Data_In : IN std_logic_vector(WIDTH -1 DOWNTO 0);
        Data_Out: OUT std_logic_vector(WIDTH -1 DOWNTO 0);
        WR       : IN std_logic);
END top;

architecture top_arch of top is
component spmem
generic ( add_width : integer := 3;
          delay : time := 2 ns;
          file_name : string := "empty.dat";
          WIDTH : integer := 8 );
PORT (
    clk      : IN std_logic;
    reset    : IN std_logic;
    add      : IN std_logic_vector(add_width -1 downto 0);
    data_In : IN std_logic_vector(WIDTH -1 DOWNTO 0);
    data_Out: OUT std_logic_vector(WIDTH -1 DOWNTO 0);
    WR       : IN std_logic);
END component;

begin -- top_arch

```

```

G1: if LEN=1 generate
    INST1 : spmem generic map (add_width,delay,file_name,width)
                port map (clk,reset,add,data_in,data_out,wr);
end generate G1;

G2: if LEN=2 generate
    INST2 : spmem generic map (add_width,delay,file_name,width)
                port map (clk,reset,add,data_in,data_out,wr);
end generate G2;

end top_arch;

```

In the above example, you can override the generics at runtime. The use model is as follows:

Analysis

```
% vhdlan spec_mem.vhd TOP.vhd
```

Note:

Specify the VHDL bottommost entity first, then move up in order.

Elaboration

```
% vcs TOP
```

Simulation

```
% simv -g generics_file
```

Following is the `generics_file`:

```

assign 1 /TOP/LEN
assign "OK.dat" /TOP/G1/INST1/FILE_NAME
assign (4 ns) /TOP/G1/INST1/delay
assign 16 /TOP/width
assign 4 /TOP/add_width

```

As per `generics_file`, VCS overrides the generics `LEN`, `width`, and `add_width` in the `TOP.vhd` file, and `FILE_NAME` and `delay` generics defined in the `spmem.vhd` file.

Passing Values from the Runtime Command Line

The `$value$plusargs` system function can pass a value to a signal from the `simv` runtime command line using `plusarg`. The syntax is as follows:

```
integer = $value$plusargs("plusarg_format",signalname);
```

The *plusarg_format* argument specifies a user-defined runtime option for passing a value to the specified signal. It specifies the text of the option and the radix of the value that you pass to the signal.

The following code example contains this system function:

```
module valueplusargs;
reg [31:0] r1;
integer status;

initial
begin
$monitor("r1=%0d at %0t",r1,$time);
#1 r1=0;
#1 status=$value$plusargs("r1=%d",r1);
end
endmodule
```

If you enter the following `simv` command line:

```
% simv +r1=10
```

The `$monitor` system task displays the following:

```
r1=x at 0
r1=0 at 1
r1=10 at 2
```

Using -f Runtime Option

You can use the `-f <filename.f>` runtime option to specify user-defined arguments in a file. These arguments are those that you specify on the `simv` command line. This option works for all mixed HDL designs, pure VHDL, and pure Verilog designs.

Limitations

- Nested file inclusion is not supported
- Environment expansion is not supported
- Complex string options are not supported
- You cannot specify multiple options on the same line. This is illustrated in the below example:

```
%simv -f <filename.f>
      filename.f

      -ova_report
      -lca
```

```
-cm_name foo
...
...
```

Saving and Restarting the Simulation

You can use the `$save` and `$restart` system tasks to save the checkpoints of the simulation at arbitrary times. The resulting checkpoint files can be executed at a later time, causing simulation to resume at the point immediately following the save.

Note:

- Save and restart using the `$save` and `$restart` system tasks is for the designs having both DUT and the testbench in Verilog HDL. You can also use the UCLI `save` and `restart` feature. For more information, see *Unified Command-line Interface User Guide*.
- You can restore the gzipped snapshots using system tasks or the UCLI `restore` command.
- The files which are created and closed using Verilog system tasks before saving the simulation state are also retained while restoring the simulation. To enable this feature, pass the `-save_closed_files` option at runtime during saving the simulation state.

Benefits of save and restart include:

- Regular checkpoints for interactively debugging problems found during long batch runs
- Use of plusargs to start action such as `$dumpvars` on restart
- Execution of common simulation system tasks such as `$reset` just once in a regression

Restrictions of save and restart include:

- Requires extra Verilog code to manage save and restart
- Must duplicate start-up code if handling plusargs on restart
- File I/O suspend and resume in PLI applications must be given special consideration
- Simprofile flow is not supported
- Not supported with the `+vcs+loopreport` option

- The `-covg_disable_cg` runtime option takes effect during the beginning of simulation. Therefore, if you run the option only during the restore step or remove the option only from the restore step, it does not have any effect.
- During restore, you must pass the same coverage options that are passed during saving the simulation state. The changed coverage options do not take effect during the restore step.

Save and Restart Example

[Example 25](#) illustrates the basic functionality of save and restart.

The `$save` call is scheduled and executed as the last event in the current time. Therefore, the events delayed with `#1` are the first to be processed upon restart.

Example 25 Save and Restart Example

```
% cat test.v
module simple_restart;
initial begin
#10
$display("one");
$save("test.chk");
$display("two");
#1 // make the following occur at restart
$display("three");
#10
$display("four");
end
endmodule
```

Now compile the example source file:

```
% vcs test.v
```

Run the simulation:

```
% simv
```

VCS displays the following:

```
one
two
$save: Creating test.chk from current state of simv...
three
four
```

To restart the simulation from the state saved in the check file, enter:

```
% simv -r test.chk
```

VCS displays the following:

```
Restart of a saved simulation
three
four
```

Save and Restart File I/O

VCS remembers the files you opened via `$fopen` and reopens them when you restart the simulation. If no file with the old file name exists, VCS opens a new file with the old file name. If a file exists having the same name and length at the time you saved the old file, then VCS appends further output to that file. Otherwise, VCS attempts to open a file with a file name equal to the old file name plus the suffix `.N`. If a file with this name already exists, VCS exits with an error message.

If your simulation contains PLI routines that do file I/O, the routines must detect both the save and restart events, closing and reopening files as needed. You can detect `save` and `restart` calls using `misctf` callbacks with reasons `reason_save` and `reason_restart`.

When running the saved checkpoint file, be sure to rename it so that further `$save` calls do not overwrite the binary you are running. There is no way from within the Verilog source code to determine if you are in a previously saved and restarted simulation, therefore, you cannot suppress the `$save` calls in a restarted binary.

Saving and Restoring Files During Save and Restore

You can save all files that are open in read or write mode at the time of save using the following runtime options. All these files are saved in the directory named:

`<name_of_the_saved_image>.FILES.`

`-save`

Saves all open files in writable mode.

`-save_file <file name> | <directory name>`

Saves all open files in writable mode, and all files that open in read-only mode, depending on the option you specify:

- With `<file name>`, saves the specified open file in read/write mode.
- With `<directory name>`, saves all files in the specified directory open in read/write mode.

`-save_file_skip <file name> | <directory name>`

This allows you to skip saving one or more files depending on the option:

- With `<file name>`, skips saving the specified file that is open in read/write mode.
- With `<directory name>`, skips all files in the specified directory that are open in read/write mode.

Restoring the Saved Files from the Previous Saved Session

At restore time, you can remap any old path where files were open at the time of save to the new place where restore searches using the `-pathmap` option.

Example:

```
%simv -pathmap <file_with_pathmaps>
```

where,

```
<file_with_pathmaps>:  
<old_directory_path_name>:<new_directory_path_name>
```

Save and Restart With Runtime Options

If your simulation behavior depends on the existence of runtime `plusargs` or any other runtime action (such as reading a vector file), be aware that the restarted simulation uses the values from the original run unless you add special code to process runtime events after the restart action. Depending on the complexity of your environment and your usage of the save and restart feature, this can be a significant task.

For example, if you load a memory image with `$readmemb` at the beginning of the simulation and want to be able to restart from a checkpoint with a different memory image, you must add Verilog code to load the memory image after every `$save` call. This ensures that at the beginning of any restart the correct memory image is loaded before simulation begins. A reasonable way to manage this is to create a task to handle processing arguments, and call this task at the start of execution, and after each save.

The following example illustrates this in greater detail. The first run optimizes simulation speed by omitting the `+dump` option. If a bug is found, the latest checkpoint file is run with the `+dump` option to enable signal dumping.

```
// file test.v
module dumpvars();
task processargs;
begin
  if ($test$plusargs("dump")) begin
    $dumpvars;
  end
end
end task
//normal start comes here
initial begin
```

```

processargs;
end
// checkpoint every 1000 time units
always
#1000 begin
// save some old restarts
$system("mv -f save.1 save.2");
$system("mv -f save save.1");
$save("save");
#1 processargs;
end
endmodule
// The design itself here
module top();
.....
endmodule

```

Additional Save and Restore Options

VCS supports the following additional save and restore options:

- `-save_options=SUBOPTION1,SUBOPTION2,...`
 - `-save_options=continue_log -l simv.log`

Continues writing to the log file to provide a coherent log of the full execution, which contains entries from the simulation before save and from after restore but no messages about the snapshot restore itself.

- `-save_options=skip_zero_bytes`

Handles snapshots with a lot of 0-bytes more efficiently. Try this if the memory usage of the restored simulation is much larger than that of the simulation before save.

- `-save_options=use_gzip`

Uses the gzip compressor for the memory dump.

- `-save_options=use_lz4`

Uses the lz4 compressor for the memory dump.

- `-no_save`

Deactivates the Save/Restore functionality (thereby avoiding, for example, process restarts that would be necessary to prepare for Save/Restore).

- `-save_uncompressed`

Saves memory dump in an uncompressed form.

- `-daidir=/path/to/simv.daidir`

Specifies the path to the `simv.daidir` directory in case it has been moved.

For example, this option is required in a scenario where the simulation is saved in DIR1, `simv` and `simv.daidir` are moved to DIR2, and the simulation is restored in DIR3.

Specifying Long Time Before Stopping the Simulation

You can use the `+vcs+stop+time` runtime option to specify the simulation time when VCS stops the simulation. This works if the `time` value you specify is less than 232 or 4,294,967,296. You can also use the `+vcs+finish+time` runtime option to specify when VCS either stops or ends the simulation, provided that the `time` value is less than 232.

For `time` values greater than 232, you must follow a special procedure that uses two arguments with the `+vcs+stop` or `+vcs+finish` runtime options, as shown below:

```
+vcs+stop+<first argument>+<second argument>
+vcs+finish+<first argument>+<second argument>
```

This procedure is as follows:

For example, if you want a time value of 10,000,000,000 (10 billion):

1. Divide the large `time` value by 232.

In this example:

$$\frac{10,000,000,000}{4,294,967,296} = 2.33$$

2. Narrow down this quotient to the nearest whole number. This whole number is the second argument.

In this example, you would narrow down to 2.

3. Multiply 232 with the second argument (that is, 2), and then subtract the obtained result from the large time value (that is, subtract 2 X 232 from the large `time` value), as shown below:

$$10,000,000,000 - (2 * 4,294,967,296) = (1,410,065,408)$$

This difference is the first argument.

You now have the first and second argument. Therefore, in this example, to specify stopping simulation at time 10,000,00,000, you would enter the following runtime option:

```
+vcs+stop+1410065408+2
```

VCS can do some of this work for you by using the following source code:

```
module wide_time;
  time wide;
  initial
    begin
      wide = 64'd10_000_000_000;
      $display("Hi=%0d, Lo=%0d", wide[63:32], wide[31:0]);
    end
endmodule
```

VCS displays the following:

Hi=2, Lo=1410065408

Preventing Time 0 Race Conditions

At simulation time 0, VCS executes always blocks where any of the signals in the event control expression that follows the `always` keyword (the sensitivity list) initializes at time 0.

For example, consider the following code:

```
module top;
  reg rst;
  wire w1,w2;
  initial
    rst=1;
    bottom bottom1 (rst,w1,w2);
endmodule

module bottom (rst,q1,q2);
  output q1,q2;
  input rst;
  reg rq1,rq2;

  assign q1=rq1;
  assign q2=rq2;

  always @ rst
begin
  rq1=1'b0;
  rq2=1'b0;
  $display("This always block executed!");
end
endmodule
```

With other Verilog simulators, there are two possibilities at time 0:

- The simulator executes the initial block first, initializing `reg rst`, then the simulator evaluates the event control sensitivity list for the `always` block and executes the `always` block because the simulator initialized `rst`.
 - The simulator evaluates the event control sensitivity list for the `always` block, and so far, `reg rst` has not changed its value during this time step. Therefore, the simulator does not execute the `always` block. Then the simulator executes the `initial` block and initializes `rst`. When this occurs, the simulator does not re-evaluate the event control sensitivity list for the `always` block.
-

Resolving RTL Simulation Races in Mixed HDL Designs

A race between data and clock signal occurs when both signals change at the same simulation time and both are input to the same sequential element (flip-flop or latch). However, it is expected that the clock arrives before data and samples the previous settled value of data. When clock arrives after or at the same cycle as data, the new value of data is sampled which causes incorrect results.

VCS helps resolve these RTL simulation races in mixed VHDL-Verilog designs. The following section illustrates how to resolve race conditions.

Recommended Approach to Resolve Race Conditions

It is recommended to use the following methodologies to resolve the race conditions:

1. Use the VHDL `-sn=+rdr` race detector option and try to manually fix the RTL code. If it is not possible to modify the RTL code, use the race detector output as an input to the configuration file. For the use model, see section [Using the VHDL Race Detector](#).
2. Use the `-sn=+rfx` automatic option to accelerate the scalars (clocks) over the boundary and insert NBA delays only for the data drivers that directly provide stimulus to the Verilog boundary. For information about the using the configuration file, see section [Using the Race Configuration File](#).

Using the VHDL Race Detector

If a simulation race is already detected, delay the data to fix the race. To resolve the simulation race using the VHDL race detector, execute the following steps:

1. To insert delays, modify the VHDL code and add “`after 0.0 sec`” on drivers for data signals reported as racy. The delay can be a non-zero delay or a non-blocking assignment zero delay specified as follows in the source code:

```
signal <= data after 0.0 sec;
```

This is legal VHDL and works with any tool. However, VCS recognizes it as a VHDL Non-Blocking Assignment (NBA). To enable VHDL NBA assignment, use the `vcs -sn=+nbavhd` option.

2. Enable the race detector using the `-sn=+rdr` option, as shown in the following command:

```
% vcs -sn=+rdr; simv
```

This option generates the `snps_vcs_vhdl_race.log` file, which can be used as an input to the configuration file to insert NBA delays on those signals.

Using the Race Configuration File

To enable the configuration input, use the `-sn=+nbacfg` option with `snps_vcs_vhdl_nba.cfg` as an input file. The configuration input file contains a list of signals for which NBA delays are inserted.

The configuration file with the NBA delays can be generated by the race detector using the following commands:

```
% vcs -sn=+rdr; simv
% cp snps_vcs_vhdl_race.log snps_vcs_vhdl_nba.cfg
% vcs -sn=+nbacfg; simv
```

OR

The configuration file with the NBA delays can be generated by the automatic race solution along with the log file (`-sn=+rfx+nbalog`). This can be used as a starting list and enables you to tune the configuration list:

```
% vcs -sn=+rfx+nbalog; simv
% cp snps_vcs_vhdl_nba.log snps_vcs_vhdl_nba.cfg
% vcs -sn=+nbacfg; simv
```

Resolving RTL Simulation Races in Verilog Designs

A race between data and clock signal occurs when both signals change at the same simulation time and both are input to the same sequential element (flip-flop). However, it is

expected that the clock arrives before data and samples the previous settled value of data. When clock arrives after or at the same cycle as data, the new value of data is sampled which causes incorrect results.

VCS helps resolve these RTL simulation races in Verilog design. The following section illustrates how to resolve race conditions.

Recommended Approach to Resolve Race Conditions

It is recommended to use the Verilog `-deraceclockdata` option to enable the clock-data resolution for your entire design. For more information, see section [Using Clock-Data Resolution](#).

Using Clock-Data Resolution

Using clock-data resolution ensures that the previous value of the data is always sampled.

This significantly improves the verification productivity. There is no simulation mismatch due to races on flops while migrating to a new release or modifying options that are provided to the VCS command line.

Use Model

To enable the clock-data resolution for your entire design, use the Verilog `-deraceclockdata` option.

```
% vcs -deraceclockdata <other vcs options>
```

By default, the `-deraceclockdata` option samples memory up to 8 MB. You can use the `-deraceclockdata=fullmem` option to remove the restriction on the memory size.

Example

Consider the following test case:

```
module dff(q, d, clk, clr);
    output q;
    input d, clk, clr;
    reg q;

    always @ (posedge clk , negedge clr) begin
        if (!clr)
            q <= 1'b0;
        else
            q <= d;
    end
endmodule

//=====top module=====
module top(clk, d, clr, out);
    input clk, d, clr;
    output out;
```

```

wire clk2;
wire q1;

dff div2(clk2, ~clk2, clk, clr);
dff f1(q1, d, clk, clr);
dff f2(out, q1, clk2, clr);
endmodule

//=====testbench=====
module tb;
    reg clk;
    reg clr;
    reg d;

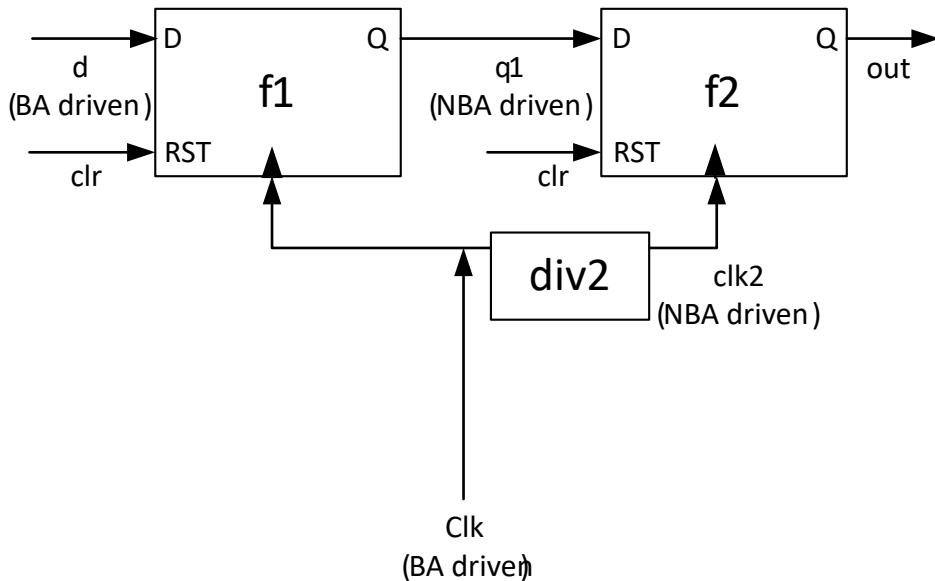
    wire dout;
    top t1(clk,d,clr,dout);
    initial begin
        clk = 1'b0;
        d = 1'b0;
        clr <= 1'b0;

        fork
            forever clk = #5 ~clk;
            #25 d = 1'b1;
            #45 d = 1'b0;
            #9 clr <= 1'b1;
        join_none

        #70 $finish();
    end
endmodule

```

This example has three flops. For f1 flop, the clock and the data changes at the same time in the blocking assignment region. For f2 flop, both clock and data are changing in the NBA region.



Compile and run the test case using the following command line:

```
% vcs -sverilog -deraceclockdata test.v
% simv
```

To compile the test case with FSDB dumping enabled:

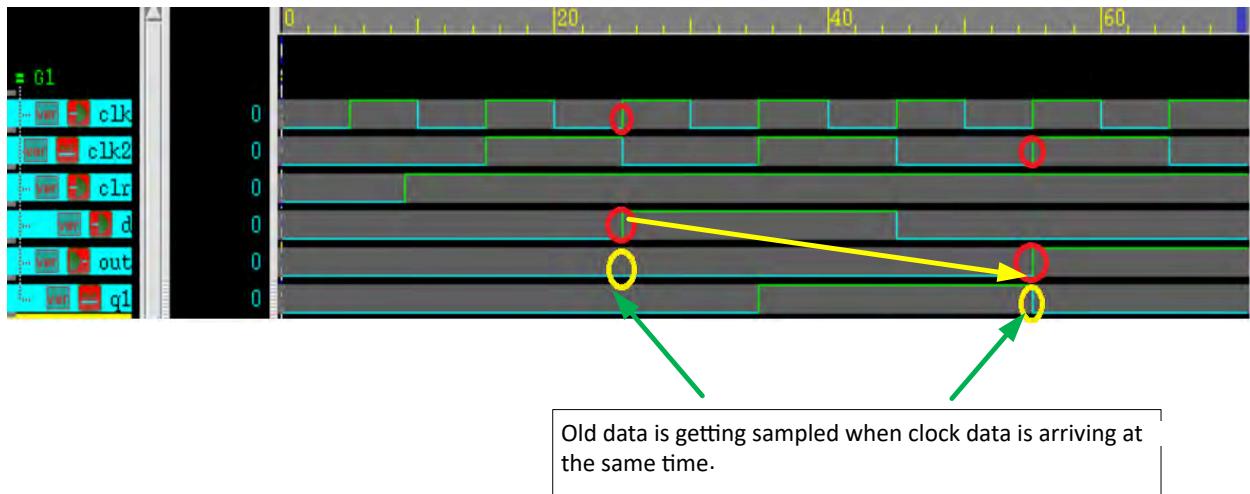
- Enable the FSDB dumping by calling the `$fsdbDumpvars` dumping task in the initial block of the `tb` module.
- Use the following command line:

```
% vcs -sverilog -deraceclockdata test.v -debug_access
-debug_region+cell
```

Note:

You must set the `VERDI_HOME` environment variable for FSDB dumping.

Import the design and load the generated FSDB file into the Verdi platform. The following waveform is generated:



As shown in the waveform, it is able to consistently sample the clock first using the `-deraceclockdata` option. Therefore, it picks up the previous value of the data.

Limitations

The feature has the following limitations:

- The feature is supported for inferred flops in Verilog only.
- Process blocks containing following constructs are ignored:
 - Call to impure function
 - Having immediate assertions inside
 - Dynamic variables
 - System Task Calls other than `$display` and `$monitor`
- Flops modeled using both blocking assignment region and non-blocking assignment region (in same process) are not supported.
- UDPs with synchronous control are not supported.

Supporting Simulation Executable to Return Non-Zero Value on Error Results

Simulation executable generated by VCS returns non-zero value in case of errors, fatal errors, and assertion failures.

The simulation executable return values on errors, fatal errors, and assertion values are:

- 0 (no indication)
- 1 (as in runtime crash or system crash)
- 2 (error)
- 3 (fatal)

The possible scenarios and return error value for the scenarios are listed in the following table:

Table 13 List of Scenarios and Return Error Values

Scenario	Return Error Value
\$fatal/UVM_FATAL/OVM_FATAL/VMM_FATAL	3
\$error/UVM_ERROR/ OVM_ERROR/ VMM_ERROR/ Errors promoted from warning messages to errors	2
NLP ERROR	2
Assertion failure Verilog	2
\$warning /UVM_WARNING/ OVM_WARNING/ VMM_WARNING	0
NLP WARNING	0
Unique/priority RT warnings	0
-xzcheck	0

Note:

The `-assert quiet` and `-assert quiet1` runtime options cannot override the exit status in case of assertion failures. The exit status value still must be value 2 in case of assertion failures.

Use of `-error` runtime option generates non-zero return values in case of errors, fatal errors, as well as in case of errors resulting from warning messages.

If the messages are suppressed using the `-error` runtime option, non-zero return values are not generated.

If a simulation has several errors, fatal errors, and/or assertion failures, the most severe status must be returned. For example, if simulation has both `$error` and `$fatal` messages, the returned status must be of value 3 as in case of `$fatal` scenario.

Use Model

The following is the use model for this feature:

- Compile time

The compile time is same as the previous use model. There is no change needed.

- Runtime

```
% simv -exitstatus
% echo $status
```

This returns a value based on the type of exit

Limitation

Following is the limitation of this feature:

- VMM is not supported.

Supporting Memory Load and Dump Task Verbosity

If you use the `-diag sys_task_mem` compile-time option, `$writememh`, `$writememb`, `$readmemh` and `$readmemb` system tasks get into a verbose mode and displays the following information:

- Full path of the file being written by `$writememh` and `$writememb` and being read by `$readmemh` and `$readmemb`, such as `/foo/bar/filename`.
- The full hierarchical instance path of the module from where `$writememh`, `$writememb`, `$readmemh`, and `$readmemb` is invoked, such as `soc.a1.b1.c1.my_module_instance`. All instances are reported.
- The name of the module template from where it is displayed, such as `my_module`.
- The full path to the file that includes the module that contains the `$writememh`, `$writememb`, `$readmemh`, and `$readmemb`, such as `/baz/qux/my_module.vs`.

Syntax:

```
$writememh ( filename , memory_name [ , start_addr [ ,
finish_addr ] ] ) ;
```

```
$writememb ( filename , memory_name [ , start_addr [ ,
finish_addr ] ] ) ;
```

The system tasks \$writememh and \$writememb dump memory array contents to files that are readable by \$readmemh and \$readmemb respectively.

```
$readmemh ( mem_name , start_address , finish_address , string { ,
string } ) ;
```

```
$readmemb ( mem_name , start_address , finish_address , string { ,
string } ) ;
```

The system tasks \$readmemh and \$readmemb load data into memory mem_name from a character string.

The \$readmemh and \$readmemb system tasks take memory data values and addresses as string literal arguments. These strings take the same format as the strings that appear in the input files and are passed as arguments to \$readmemh and \$readmemb. The start_address and finish_address indicate the bounds for the data to be stored in the memory.

Use Model

The following is the use model to support memory load and dump task verbosity:

```
%vcs -diag sys_task_mem
```

The following examples illustrate the usage of \$writemem and \$readmem system tasks:

Example 26 Example to illustrate \$writemem system task

```
//test.v
module test;
reg [31:0] mem[0:11];

initial begin
  $writememh("./datah.dat", mem);
end
endmodule
```

Run the example using the following commands:

```
% vcs test.v -diag sys_task_mem
% simv
```

It generates the following output:

```
Note-[STASK_WMEM] Encountered Memory Write Task
/home/user/task/test.v, 5
  At module test, Instance test
    Writing to file /home/user/task/datah.dat.
```

Example 27 Example to illustrate \$readmem system task

```
//test.v
module test;
reg [31:0] mem[0:11];

initial begin
  $readmemh("./data.dat", mem);
end
endmodule
```

Run the example using the following commands:

```
% vcs test.v -diag sys_task_mem
% simv
```

It generates the following output:

```
Note-[STASK_RMEM] Encountered Memory Read Task
/home/user/task/test.v, 5
  At module test, Instance test
    Reading from file /home/user/task/data.dat.
```

6

The Unified Simulation Profiler

The unified simulation profiler reports the amount of CPU time and machine memory used by the Verilog, SystemVerilog, and VHDL parts of the design. For SystemC parts, the unified profiler reports the CPU time and memory.

This information is summarized at a high-level and also further categorized in detail at the module level, instance level, and based on constructs such as `always` procedures.

The reports are generated by the `profrpt` profile report generator. The supported report formats are text, JSON, and HTML.

The major sections in this unified profiler documentation are as follows:

- [The Use Model](#)
- [HTML Profiler Reports](#)
- [Generating Simprofile Dynamic Report](#)
- [Advanced Simprofile Usages](#)
- [Line-Based CPU Time Profiler](#)
- [Isolating the Cost of Garbage Collection](#)
- [Isolating the Cost of Loading Design Database](#)
- [Third-Party Shared Library Profiler Report](#)
- [VHDL Unified Simulation Profiler Report](#)
- [The HSIM View With the Simulation Profiler Report](#)
- [Limitations](#)

The Use Model

Compile Time Option

Compile your design using the `-simprofile` compile-time option.

Note:

If this is not the first compilation of your design, delete the `csrc` and `simv.daidir` directories and `simv` executable file before this step. Incremental compilation is not yet supported for the unified profiler.

The `-simprofile` compile-time option also has optional arguments:

`-simprofile=time`

Specifies compiling the design and testbench for collecting both CPU time and machine memory profile information. Then at runtime specifies collecting CPU time profile information.

`-simprofile=mem`

Specifies compiling the design and testbench for collecting both CPU time and machine memory profile information. Then at runtime, specifies collecting machine memory profile information.

With these arguments, you can omit the `-simprofile` runtime option if you want to collect the type of profile information that you have specified at compile time.

If at runtime you want VCS to collect different types of profile information than the type you specified at compile time, you can specify different type with the runtime option.

For example:

```
%> vcs source.v -simprofile=time
%> simv -simprofile mem
```

Runtime Options

At runtime, you can specify the type of data VCS collects during simulation by entering the `-simprofile` runtime option with a keyword argument or sub-option in the following way:

```
% simv -simprofile <sub-options>
```

These keyword arguments or sub-options are as follows:

`time`

The `time` argument specifies collecting CPU time profile information.

`mem`

The `mem` argument specifies collecting machine memory profile information.

`fastmem`

Using the `fastmem` argument, the tool collects the memory profile information in an optimized manner which reduces the time taken in memory profiling. For example,

```
%> simv -simprofile fastmem
```

You can use the `fastmem` argument when you are working on a large design and want to simulate memory profile as fast as time profile.

`time+flamegraph`

You can use the `time+flamegraph` sub option to enable the Time FlameGraph view in the Simprofile HTML report. For details, refer to the [The Time FlameGraph View](#) section.

`mem+rpm`

Use the `mem+rpm` sub option to enable the Real Time Profile Monitoring (RPM) feature. For more details, refer to the [Generating Simprofile Dynamic Report](#) section.

`noprof`

Tells VCS not to collect profiling information at runtime. Synopsys recommends entering this runtime option and keyword argument or sub-option instead of simply omitting the `-simprofile` runtime option. See [Omitting Profiling at Runtime](#).

`noreport`

After simulation, but before you run the `profrpt` profile report generator there is a summary of profile information available to you, see [Running the profrpt Profile Report Generator](#). The `noreport` tells VCS to collect profile information at runtime but not write the `profileReport.html` file or the `profileReport` directory after simulation, see [Omitting Profile Report Writing after Runtime](#)

`-simprofile_dir_path <pathname>`

By default, VCS creates the profile database and the directory named `simprofile_dir` that contains all the profile information gathered during simulation, in the directory that contains the `simv` executable.

You can specify a different directory for the profile database with the `-simprofile_dir_path pathname` runtime option.

For example,

```
% simv -simprofile time -simprofile_dir_path /tmp/SUBDIR1
```

To use this option, the directories in the pathname must already exist. VCS does not create the directories in the pathname.

`-simprofile_report <reportname>`

By default, VCS writes the profile report named `profileReport.html` and the corresponding `profileReport` directory that contains the profile report information. You can enter the `-simprofile_report <reportname>` runtime option to specify a different name for this file and directory.

For example:

```
% simv -simprofile_report memory_rppt_default_constraints
```

This example creates the following profile report and report directory named `memory_rppt_default_constraints.html` file and `memory_rppt_default_constraints` directory respectively.

`-simprofile no-altstack` or `-simprofile allow-user-sigaltstack`

VCS simulation profiler sets up the signal stack size for sampling up to 256 KB. If the simulation runs with any external application that lowers this stack size, VCS generates the following error:

```
Error-[SIMPROFILE-STACK-TOO-SMALL] Stack size validation failed
```

The stack size of the signal handler is overridden by a smaller value 163840 (current value: 262144)

To proceed with simulation, modify the corresponding `sigaltstack` invocation with at least 256KB or use one of the following workarounds:

- Specify '`-simprofile no-altstack`', then the profiler uses the regular stack.
- Specify '`-simprofile allow-user-sigaltstack`', then the profiler uses the specified stack.

To resolve the error, modify the external application or use one of the following runtime options:

- `-simprofile no-altstack`
- `-simprofile allow-user-sigaltstack`

Omitting Profiling at Runtime

If you compiled the design to collect profile data by entering the `-simprofile` compile-time option, but decide to forgo the performance cost of collecting profile data during simulation, you enter the `-simprofile noprof` runtime option and the keyword argument.

When you do so, VCS does not create the `profileReport.html` file or the `profileReport` directory, but does create the `simprofile_dir` directory. However, this `simprofile_dir` directory remains empty.

Omitting the `-simprofile` runtime option after compiling with the `-simprofile` compile-time option is not recommended.

If you compile your design with the `-simprofile` compile-time option, but omit the `-simprofile` runtime option when you run the simulation, VCS by default, creates the `simprofile_dir` and `profileReport` directories and write the `profileReport.html` in the current directory that only contains information about the simulation time.

The `simprofile_dir` directory never contains any information that you can read. It does contain database files that come with a performance cost.

If `simprofile_dir`, the directory `profileReport`, and the file `profileReport.html` exist and you run the test, then VCS renames them to `simprofile_dir.integer`, `profileReport.integer`, `profileReport.integer.html` respectively, so that the files are not overwritten.

Omitting Profile Report Writing after Runtime

If you have compiled your design to collect profile data and want VCS to collect profile data during simulation, but you do not want VCS to write the `profileReport.html` file or the `profileReport` directory, enter the `-simprofile noreport` runtime option and the keyword argument.

When you do this you still have the profile database and can obtain profile information after simulation with a `profrpt` command line.

Running the profrpt Profile Report Generator

After simulation, but before you run the `profrpt` profile report generator, VCS provides a summary report of profile information.

At the end of simulation VCS writes the `simprofile_dir` and `profileReport` directories and the `profileReport.html` file.

The `simprofile_dir` directory contains the databases that are read by the `profrpt` profile report generator to write profile reports in a separate step after the simulation.

The `profileReport.html` file can tell you the total simulation time and the location of the profiler databases.

You can run the `profrpt` profile report generator with the `profrpt` command line. The syntax of this command line is as follows:

```
profrpt simprofile_dir [-view view1 [+view2 [...]]]
[-h|-help] [-format text|html|json|ALL] [-output <name>]
[-filter percentage] [-snapshot [delta|incr|delta+incr]]
[-timeline [dynamic_memory_type_or_class +...]]
```

Where:

`simprofile_dir`

Specifies the profile database directory that VCS writes at runtime. The default name is `simprofile_dir`. You can enable the writing of this database with the `-simprofile` compile-time option and specify the kind of data in the database with the `-simprofile` runtime option.

`-view view1[+view2[+...]]`

Specifies the views you want to see in the reports, see [Specifying Views](#). You must specify this option.

`-h | -help`

Displays help information about the `profrpt` command-line options.

`-format text|html|json|ALL`

Specifies whether the report files are generated in the text files, HTML files, JSON files, or in all the formats (by specifying the `ALL` keyword). The default format is HTML. Some views, like the accumulative views, are only available in HTML format.

However, if you want to generate report in any two formats, you can specify the `-format` option as `-format html -format text`.

You can specify text format reports with the `-format text` or `-format all` option and argument on the `profrpt` command line. Text format views are merged together into a text file named `profileReport.txt` in the current directory.

If you run the `profrpt` report generator more than once, the utility overwrites the `profileReport.txt` file in the current directory so that its profile information is from the last run.

When you specify the text format reports, the `profrpt` utility also creates separate text files for each view in the profile report directory, these separate text files for each view have names such as `PeakMemInstanceView.txt` or `TimeConstr.txt`.

`-output <name>`

Specifies the name of the output directory for the profile report. If the report format is in HTML (which is the default format), `profrpt` writes in the current directory that is an HTML index file with the name `<name>.html`. This HTML index file contains hypertext links to the HTML files in the output directory.

VCS writes the `profileReport` directory and the HTML index file `profileReport.html` at the end of simulation. So, if you omit the `-output` option, `profrpt` renames this directory and file `profileReport.integer` and `profileReport.integer.html` and then

writes a new `profileReport` directory and `profileReport.html` file. This new directory and the file contain post-processing information from the database.

If the specified directory and file already exists, a warning message is generated and the `profrpt` creates a new output directory and file and renames the older output `name.integer` and `name.integer.html` to differentiate them from the new directory and file.

`-filter <percentage>`

Specifies the minimum percentage of the machine memory or the CPU time that a module, instance, or construct needs to use before `profrpt` enables reporting about it in the output views and reports. The default limit is 0.5%. For a more granular report enter a small percentage, such as `-filter 0.0001`.

Filtering is not applicable to the time and memory summary views and the dynamic memory timeline report.

`-snapshot [delta|incr|delta+incr]`

Specifies writing a series of snapshot profile reports for SystemVerilog dynamic memories. It writes a snapshot report each time a dynamic memory uses a specified different amount of machine memory. For information on specifying this amount, and more on the snapshot mechanism, see [The Snapshot Mechanism](#).

`-timeline [dynamic_memory_type_or_class +...]`

Specifies two things:

- Timeline reports for SystemVerilog dynamic memories.
- Snapshot reports using the default delta threshold of 5%.

If you omit the `dynamic_memory_type_or_class +...` argument or arguments, `profrpt` writes all the dynamic class timeline views.

For information on the keyword arguments or sub-options for specifying the types of SystemVerilog dynamic memories in the timeline reports, see [Specifying Timeline Reports](#).

Specifying Views

You must enter the `-view` option on the `profrpt` command line.

The views you can specify with the `-view` option depend on the type of report that `profrpt` is writing, which depends on the argument to the `-simprofile` runtime option.

The arguments and the views that you can specify are as follows:

CPU time views

`time_summary`

To specify writing the time summary view.

`time_inst`

To specify writing the time instance view that shows the CPU time used by the various module, program, and interface instances in a design. For VHDL, this view also reports the CPU time used by the various entity/architecture instances in a design.

`time_mod`

To specify writing the time module view that shows the CPU time used by the various module, program, and interface definitions in a design. For VHDL, this view also reports the CPU time used by the various entity/architecture definitions in a design.

`time_constr`

To specify writing the time construct view that shows the CPU time used by constructs, such as the `always` procedures.

`time_solver`

To specify generating the Time Constraint Solver view.

`time_callercallee`

To specify the Caller/Callee view for CPU time information. For more details, refer to the [The Caller-Callee Views](#) section.

`time_pli`

To write the Time PLI/DPI/DirectC views.

`time_all`

Specifies writing all the supported CPU time views.

Machine memory views

`mem_summary`

To specify writing the peak memory summary view, which is when your design used the most machine memory.

`mem_inst`

To specify writing the peak memory instance view that shows the peak memory used by the various module, program, and interface instances in a design. For VHDL, this view also reports the peak memory used by the various entity/architecture instances in a design.

`mem_mod`

To specify writing the peak memory module view that shows the peak memory used by the various module, program, and interface definitions in a design. For VHDL, this view also reports the peak memory used by the various entity/architecture definitions in a design.

`mem_constr`

To specify writing the peak memory construct view that shows the peak memory used by constructs.

`dynamic_mem`

To specify writing the dynamic memory peak view.

`dynamic_mem+stack`

To specify writing the dynamic memory peak view, and machine memory stack traces. The stack traces can help you determine which callers consume the most memory. For more information, see [Recording and Viewing Memory Stack Traces](#).

`mem_solver`

To specify generating the Memory Constraint Solver view.

`mem_callercallee`

To specify the Caller/Callee view for the machine memory information. For more details, refer to the [The Caller-Callee Views](#) section.

`mem_pli`

To write the Memory PLI/DPI/DirectC views.

`mem_all`

To specify writing all the supported machine memory views. This argument also enables machine memory stack traces.

Both memory and time profiler:

`ALL`

To specify writing all supported views. The `profprt` output is the HTML or text files for all these views, including machine memory stack traces.

The Snapshot Mechanism

The keyword arguments, or sub options, that you include after the `-snapshot` option control the snapshot mechanism.

```
% simv -simprofile -snapshot <sub-option>
```

The sub options are as follows:

`delta`

A numerical value (not a keyword) specifying the delta threshold before next snapshot. For example, `-snapshot 8.5` specifies a delta threshold of 8.5%. Thus, `profrpt` writes another snapshot report when a dynamic memory uses 8.5% more machine memory or 8.5% less machine memory.

`incr`

A keyword specifying the generation of another snapshot only when the machine memory for a SystemVerilog dynamic memory increases by 5%.

`delta+incr`

Specifies generation of another snapshot when the amount of machine memory used by a SystemVerilog dynamic memory increases (but not decreases) by the specified delta threshold.

If you enter no arguments or sub-options, the profiler uses the default delta threshold of 5%, and enables a new snapshot when the amount of machine memory used by a SystemVerilog dynamic memory increases or decreases by 5%.

Specifying Timeline Reports

The `-timeline` option specifies writing timeline reports in the following way:

```
% simv -simprofile -timeline <sub-option>
```

The keyword arguments or sub-options that you include after the `-timeline` option specify the types of SystemVerilog dynamic memories in the timeline reports. You can also specify a SystemVerilog class by name. Its dynamic memories are included in the timeline reports.

The arguments or sub-options for the `-timeline` option are as follows:

`vcs_ST`

keyword for string dynamic memories

`vcs_ET`

keyword for event dynamic memories

`vcs_DA`

keyword for dynamic arrays

`vcs_SQ`

keyword for queues

```
vcs_AA
keyword for associative arrays
class
a class name, not a keyword, specifying a class
ALL
keyword specifying all types of dynamic memories
```

If you enter the `-timeline` option without an argument or sub-option, `profrpt` writes timeline reports for all dynamic memories. Thus, the keyword `ALL` as an argument or sub-option is same as entering no argument or sub-option.

Recording and Viewing Memory Stack Traces

You can use the unified profiler to record stack traces whenever machine memory is allocated. The stack traces can help you determine which callers consume the most memory.

You can enable memory stack traces with the `dynamic_mem+stack`, `mem_all`, or `ALL` arguments to the `profrpt -view` option.

The following file, named `check.v`, is used to produce a sample stack trace report.

```
class Packet;
    bit[100000:0] b;
    function new();
        b = 0;
    endfunction
endclass

Packet pp[int];
int cindex = 0;
reg r;

program p;
    function Packet AllocPacket();
        begin
            AllocPacket = new;
        end
    endfunction

    task A;
        begin
            fork
                B();
                C();
            join
        end
    endtask
```

```

task B;
    int i;
    Packet lpp[int];
begin
    $display("B called");
    for (i=0; i < 100000; i++)
        pp[i] = AllocPacket();
end
endtask

task C;
    int i;
    Packet lpp[int];
begin
    $display("C called");
    for (i=0; i < 10000; i++)
        lpp[i] = AllocPacket();
end
endtask

initial
begin
    A();
end
endprogram

```

The following command sequence generates the stack trace report for the `check.v` example:

```

% vcs check.v -simprofile -sverilog

% simv -simprofile mem

% profrpt simprofile_dir -view dynamic_mem+stack

```

[Figure 4](#) shows the HTML stack trace report for the `check.v` example. The stack trace information is at the bottom of the view.

Figure 4 The Machine Memory Dynamic Object View for the Peak Snapshot

Dynamic Memory View (clock: 0)				
Dynamic Object	Instance Number	Memory	Percentage	
▶ Packet	110000	1315.92 MB	100.00 %	
▶ AllocPacket	100000	1196.29 MB	90.91 %	
▶ AllocPacket	10000	119.63 MB	9.09 %	
▶ AssociativeArray	N/A	128 B	0.00 %	
▶ String	1	48 B	0.00 %	
		1315.92 MB	100%	

Page: 1

Stack Information		
#0	AllocPacket	/file_system/big_design/VCS_user_files/stack.sv:15
#1	C	stack.sv:44
#2	A	stack.sv:23
#3	p	stack.sv:50

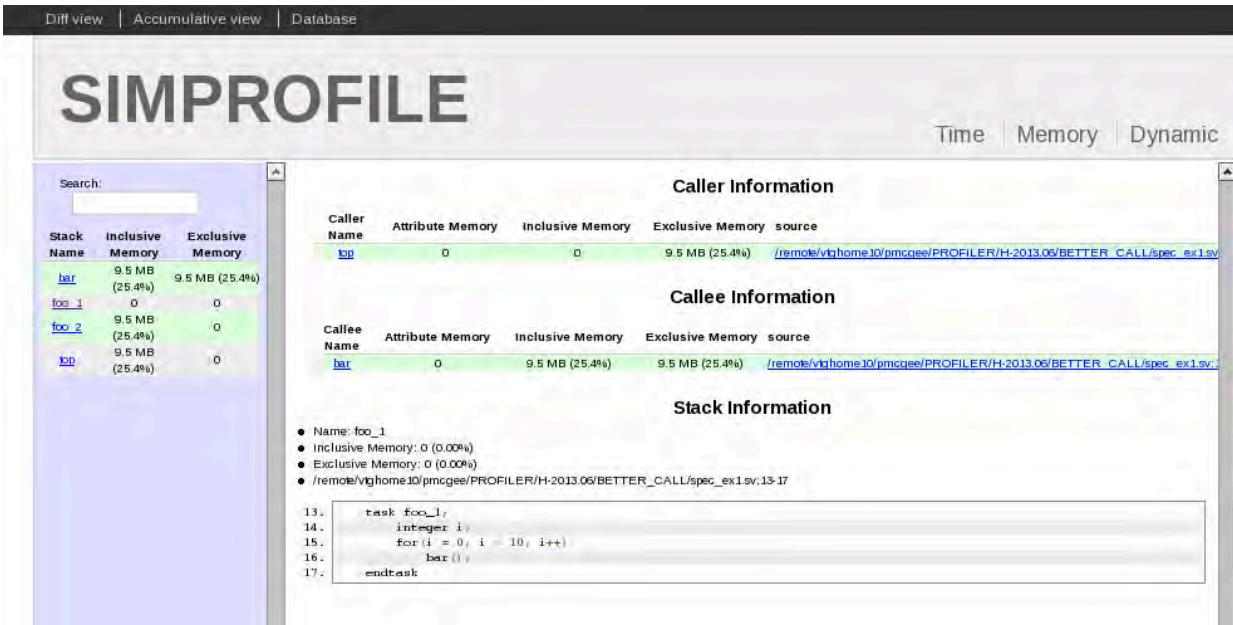
The Caller-Callee Views

The unified simulation profiler shows the hierarchical constructs that call other hierarchical constructs (these are caller constructs) and the hierarchical constructs that are called by other constructs (these are callee constructs). These new views are as follows:

- The Caller-Callee Memory View, see [Figure 5](#)
- The Caller-Callee Time view

The concepts and the organization is same in both of these views.

Figure 5 The Caller-Callee Memory View



The caller-callee views are only supported in HTML format.

For example, a hierarchical construct, such as a user-defined task can contain a task enabling statement that starts another user-defined construct.

Other views can tell you when a construct consumes the most CPU time or machine memory. But the reason a construct uses so much of these resources is not apparent. Consider the source code in [Example 28](#).

Example 28 Tasks That Consume Resources

```

module top;
    integer a[$];
    integer i;

    initial begin
        i = 0;
        foo_1();
        foo_2();
    end

    task bar;
        a.push_back(i);
        i++;
    endtask

    task foo_1;
        integer i;
        for(i = 0; i < 10; i++)

```

```

        bar();
endtask

task foo_2;
    integer j;
    for(j = 0; j < 1024 * 1024; j++)
        bar();
endtask

endmodule

```

When you run the above code and profile for machine memory, you can view and examine the Memory Construct view, as shown in [Figure 6](#).

Figure 6 The Memory Construct View

Memory Construct View (clock:0)		
Name	Size	Percentage
▼ Task		
bar	22.11 MB	34.01 %
total	21.83 MB	33.59 %
Page: 1		

In the Memory Construct View, you can see that the task named bar consumed most of the machine memory during simulations. It is only the construct that uses enough resources to warrant inclusion in this file.

The question remains why is task bar called (or enabled) so many times that it consumes so much of this resource? The unified profiler has a new view that shows us why. It is called the Memory Caller-Callee view (and its corresponding Caller-Callee Time view).

Let us consider different panes of the Caller-Callee Memory view as shown in [Figure 5](#).

Figure 7 The Left Pane of the Caller-Callee Memory View

The screenshot shows a light yellow background with a search bar labeled "Search:" at the top. Below it is a table with three columns: "Stack Name", "Inclusive Memory", and "Exclusive Memory". The table lists four entries: "bar" with 9.5 MB (51.5%) inclusive and 9.5 MB (51.5%) exclusive; "foo_1" with 0 MB inclusive and 0 MB exclusive; "foo_2" with 9.5 MB (51.5%) inclusive and 0 MB exclusive; and "top" with 9.5 MB (51.5%) inclusive and 0 MB exclusive.

Stack Name	Inclusive Memory	Exclusive Memory
bar	9.5 MB (51.5%)	9.5 MB (51.5%)
foo_1	0	0
foo_2	9.5 MB (51.5%)	0
top	9.5 MB (51.5%)	0

Unlike other left panes of views, which are a place for selecting profile databases and views, this left pane contains:

- a search field for searching for constructs, such as modules, tasks, and other hierarchical constructs in the call stack
- A list of hierarchical constructs and their inclusive and exclusive memory usage.

As shown in [Figure 7](#), the call stack for this example contains user-defined tasks bar foo_1, foo_2, and top-level module top.

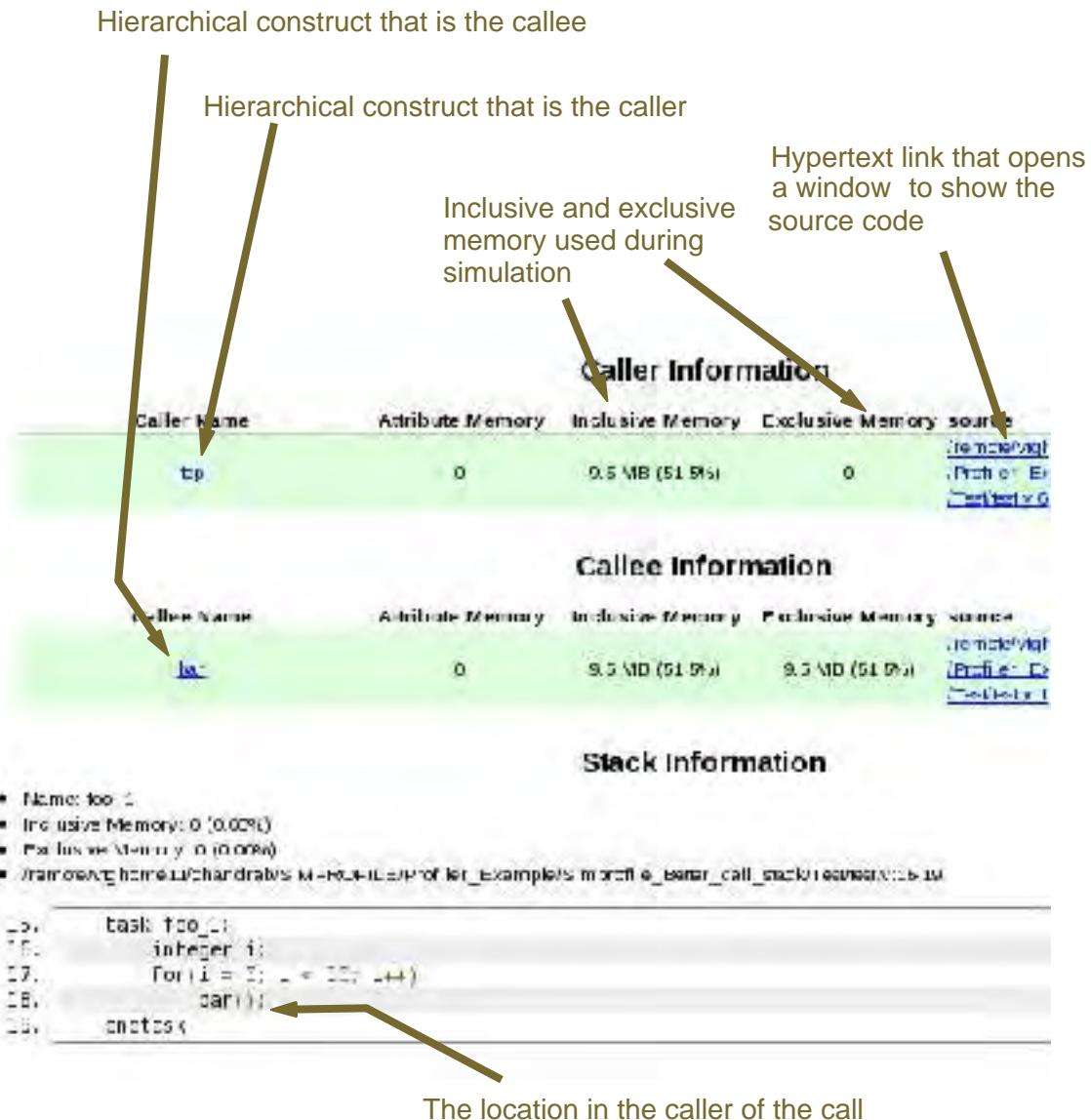
Task bar is a callee, tasks foo_1, foo_2, and module top are callers.

As in other views, inclusive is the resource used by the hierarchical construct and all such other constructs hierarchically under it. Exclusive is the amount or the resource used by the construct itself.

For an extensive list of callers and callees, there is a search field.

As shown in [Figure 7](#), task bar uses most of the exclusive machine memory that is used by the code example.

Figure 8 The Right Division of the Caller-Callee Memory View



In Figure 8, the caller of the task bar is top-level module top that contains task foo_1 and foo_2, which contain task enabling statement that calls the task bar. The *Stack Information* section of the view shows the source code of task foo_1.

As in other views, inclusive memory is the amount of memory used by the hierarchical construct and those other hierarchical constructs that are under it in the design hierarchy.

As in other views, exclusive memory is the amount of memory used solely by the hierarchical construct.

Also as in other views, attribute memory is the amount of memory used by each caller construct in the design hierarchy.

The Time FlameGraph View

VCS supports the native user-level time profiling and provides the following two methods to get the call stack:

FlameGraph

The Simprofile HTML report includes the *Time FlameGraph* view when the time flame graph feature is enabled. This view allows you to navigate and visualize time profiling of tasks and functions.

Use Model

The following is the use model to enable the *Time FlameGraph* view in the Simprofile HTML report:

```
%vcs -simprofile  
%simv -simprofile time+flamegraph
```

The following use model enables the *Time FlameGraph* view in the Simprofile HTML report and applies the custom YAML rules to the original flame graph:

```
%simv -simprofile time+flamegraph+flamegraph_config=<filename of yaml>
```

If you want to change the original flame graph, you must convert your rules to the YAML configuration and provide them to simprofile. According to the YAML configuration, simprofile adds virtual labels to the call stack and generates the required flame graph.

The YAML rule must begin with a string title and contain the following fields:

Required fields

- **priority:** Unsigned integer. The lower values mean that the generated virtual label is closer to the bottom of the flame graph.
- **type:** **The type can be time or path. When the type is set to time, you need to define the time range. When the type is set to path, the paths are compared with the filename of the latest frame of the call trace.**
- **condition:** **Specifies a list of strings. The condition is satisfied if any one of the conditions in the list is met. When you set type as time, the value you provide has the following meaning:**

10: Timeframe 10.

-(dash): From the beginning to the end of the simulation.

20-(dash): From the timeframe 20 to the end of the simulation.

-(dash)20: From the beginning of the simulation to timeframe 20.

30-40: From timeframe 30 to timeframe 40.

When you set type as path, the list of strings is treated as the python regex rule.

Optional field

- label_type: Specifies one of the following informations:
 - title: When the label_type is set to title, it specifies the name of the frame as the rule title.
 - basename: When the label_type is set to basename, the name of the frame is considered as the basename of the path.
 - regex: When the label_type is set to regex, the name of the frame is the first Capturing Group in regex. However, when the rule does not match or is not provided, the name of the frame is the rule title.

The frame generated from the rule is filled with the light blue color.

The following is the sample configuration:

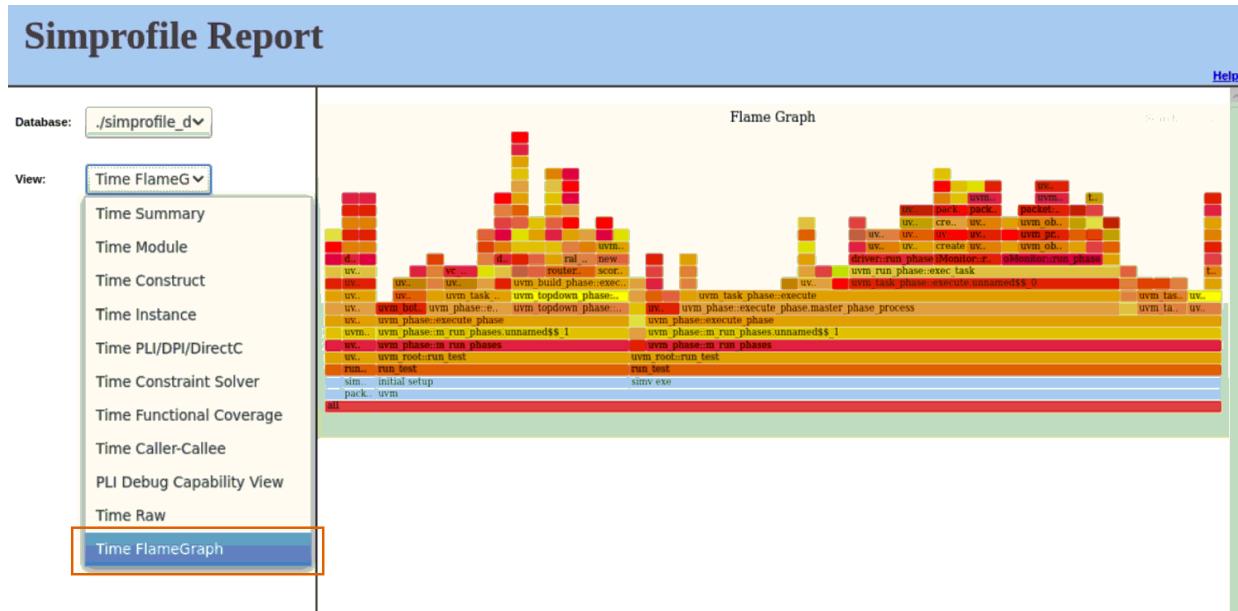
```
- uvm:
    priority: 1
    type: path
    condition: ['/uvm-1.1/']
- filename:
    priority: 2
    type: path
    condition: ['/virtual_reg/']
    label_type: basename
- initial setup:
    priority: 3
    type: time
    condition: ["0", "4-10"]
- simv exe:
    priority: 3
    type: time
    condition: ["3-7", "9-"]
- Capturing Group example: # this example rule is equivalent to basename
    priority: 4
    type: time
    condition: ["([^\n]*)$"]
    label_type: regex
...
...
```

After simulation, you can open the HTML profile report on the Firefox web browser using the following command:

```
$VCS_HOME/bin/profrpt -firefox profileReport.html
```

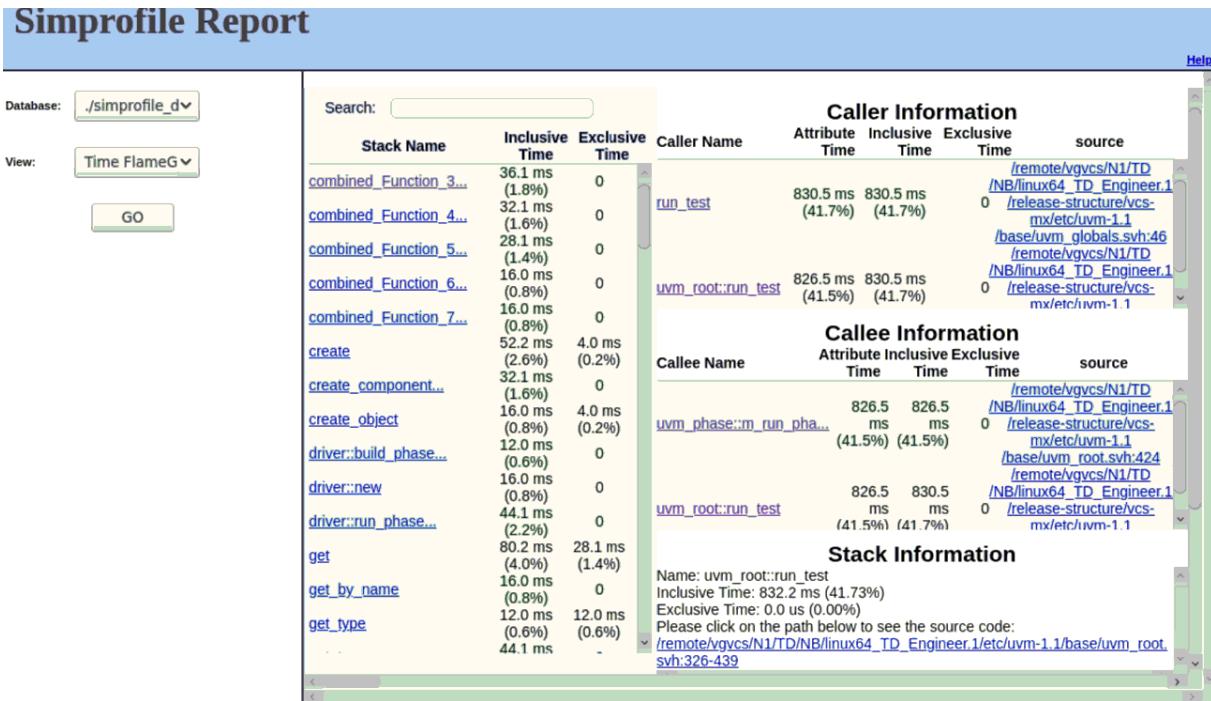
In the *View* field, select *Time FlameGraph* to open the flame graph view of tasks/functions.

The following figure displays the *Time FlameGraph* view in the HTML report:



You can perform the following operations on the **Time FlameGraph** view:

- Press the `Ctrl+f` key to search a function name.
- Click a function to zoom-in.
- To view the original flame graph, click *Reset Zoom* in the Zoom-in status.
- When you press the `ctrl` key and click a frame with the actual function name, you are redirected to the *Caller/Callee* view as shown in the following figure. Clicking frame with “all” or “frames generated by custom YAML configuration” has no impact.



JSON-Based String

JSON-Based String: The call stack related information is printed in the JSON format.

Use Model

The following is the use model:

```
$VCS_HOME/bin/profrpt -get-stacktrace [-start begin_timeframe] [-stop end_timeframe] <profile_database>
```

where

`[-start begin_timeframe]` is an optional field. The default value is 0.

`[-stop end_timeframe]` is an optional field. The default value is 18446744073709551615.

The timeframe is based on the time precision of the time scale. For example, if the timescale is 1ns/100ps, it means that the timeframe 0 is (0 ps, 100 ps] which includes 0 ps but excludes 100 ps. The supported timeframe range is from 0 to 18446744073709551615.

The timeframe supports the following two formats:

- Integer: The value can be equal to or smaller than $2^{32} - 1$.
- <higher 32bit int>+<lower 32bit int>: The value can be equal to or greater than 2^{32} . For example, 1+2 means $1*(2^{32}) + 2$.

<profile_database>: This is a required field. You must provide the path of the profile database.

When you run the following command and provide the simulation time unit either as a specific time, such as 1, or as a time range, such as 1-100, the output is generated in the *JSON Format string*:

```
$VCS_HOME/bin/profrpt -get-stacktrace
```

The JSON Format string can have the stack trace format and the overview format.

Stack trace format

The following is the stack trace format:

```
[  
  ["func/module1", fileid1 , line1],  
  ["func/module2", fileid2 , line2],  
  ["func/module3", fileid3 , line3],  
]
```

- A stack trace contains one or multiple frames by order. A frame contains the information about the function/module name, source file id, and the line number.
- In this stack trace, the call order is func/module1 calling func/module2 calling func/module3.
- When the fileid is -1, it means an unknown source, otherwise, you can use files[fileid] to access the file path. Refer to the Overview Format.

Overview format

The following is the overview format:

```
{  
  "<Simulation timeframe 1>": [  
    [Stacktrace1],  
    [Stacktrace2]  
,  
  "<Simulation timeframe 2>": [  
    [Stacktrace3]  
,
```

```

    "files": [
        "filepath1",
        "filepath2",
        "filepath3"
    ]
}

```

<Simulation timeframe> is based on the time precision of the timescale. A simulation timeframe can have multiple sampling data.

For example,

```

bin/profrpt -get-stacktrace1
{
    "1": [[1, [[{"ubus_tb_top": 0, 67}]]],
    "files": [
        "dut_dummy.v"
    ]
}
bin/profrpt -get-stacktrace 1-3
{
    "1": [[1, [[{"ubus_tb_top": 0, 67}]]],
    "2": [
        [
            2,
            [
                [
                    ["ubus_tb_top", 0, 68],
                    ["ubus_tb_mid", 0, 89],
                ]
            ]
        ],
        "3": [
            [
                3,
                [
                    [
                        ["ubus_tb_top", 0, 68],
                        ["ubus_tb_mid", 0, 90],
                        ["ubus_tb_bottom", 0, 120]
                    ]
                ]
            ],
            [
                4,
                [
                    [
                        ["ubus_tb_top", 0, 68],
                        ["ubus_tb_mid", 0, 90],
                        ["ubus_tb_bottom", 0, 122]
                    ]
                ]
            ]
        ]
    ]
}

```

You can also use the call stack information for the custom report generation. For example, if you want to sort the list of functions based on how frequently they are called, you can create the following API (a python script):

```
File: count.py
#!/bin/env python3
import sys
import json
import pprint

count = {}

data = json.load(sys.stdin)
for timeframe in data:
    # this field is for resolve file id, skip
    if timeframe == "files":
        continue
    for callstacks in data[timeframe]:
        for callstack in callstacks:
            if callstack[-1][0] not in count:
                count[callstack[-1][0]] = 1
            else:
                count[callstack[-1][0]] += 1

pprint.pprint(sorted(count.items(), key=lambda x:x[1]))
```

Use the following command line to execute the script:

```
$VCS_HOME/bin/profrpt simprofile_dir -get-stacktrace | ./count.py
```

It generates the following output:

```
[('uvm_phase::execute_phase.unnamed$$_0', 1),
 ('uvm_transaction::do_begin_tr', 5),
 ('uvm_sequencer_base::wait_for_sequences', 23),
 ('uvm_copy_map::set', 54),
 ('uvm_object::new', 122),
 ('uvm_sequence_item::get_full_name', 282),
 ('combined_Function', 566),
 ('bss_uvc_aurora_tx_driver::drive_packet.unnamed$$_0', 2651),
 ('bss_uvc_aurora_tx_monitor::collect_bss_uvc_aurora_packets', 5309)]
```

HTML Profiler Reports

The HTML profiler report provides various views as shown in the following figures:

Figure 9 *Different Views Available in Time Profiling*

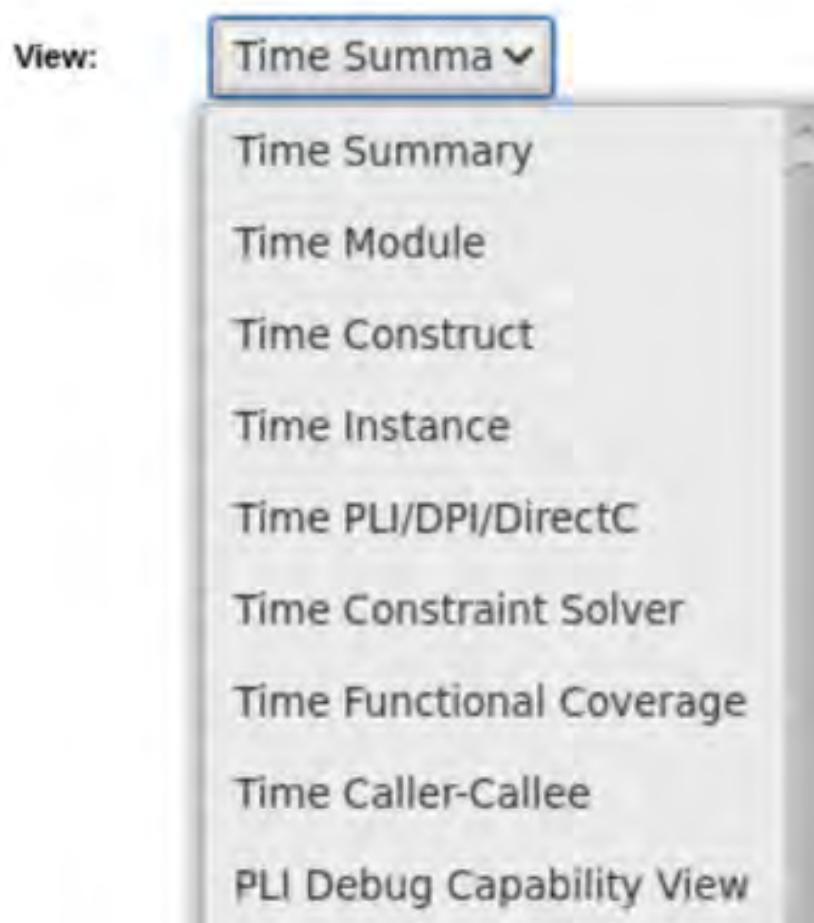
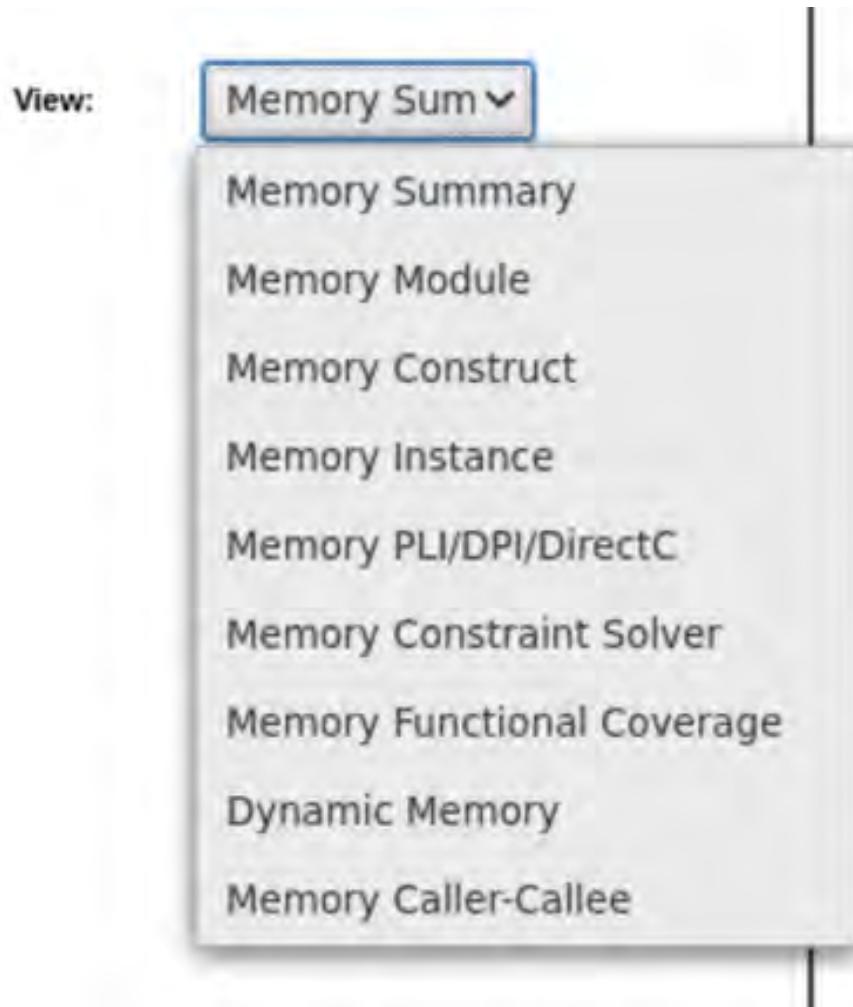
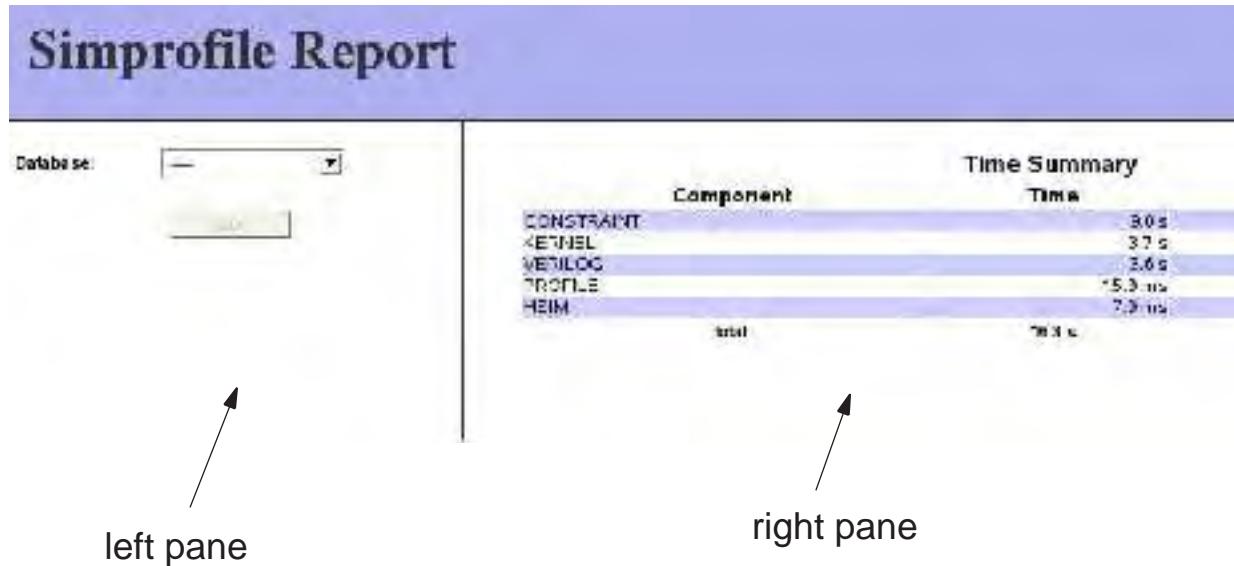


Figure 10 *Different Views Available in Memory Profiling*



The following figure displays the GUI of the HTML Profile Report:

Figure 11 The `profileReport.html` File for HTML Profile Report Information



The `profileReport.html` file contains two panes:

- The left pane is for specifying the profile database and the view you want to see. You can select a database from the **Database** list.
- The right pane is for displaying the profile information.

Figure 12 The Left Pane of the `simprofileReport.html` file



Time Summary View

If you select the Time Summary view in the View field in the left pane, then click the GO button. The right pane changes to show the following view:

Figure 13 The Left Pane of the simprofileReport.html file



The following figure displays an example of the time summary view:

Figure 14 Time Summary View

Time Summary View		
Component	Time	Percentage
VERILOG	326.47s	75.58 %
Module	262.79s	60.84 %
Package	55.72s	12.90 %
Interface	7.40s	1.71 %
Functional Coverage	479.54ms	0.11 %
Function Coverage Kernel	79.92ms	0.02 %
KERNEL	57.42s	13.29 %
Garbage Collection (101 times)	20.87s	4.83 %
Scheduler	8.22s	1.90 %
vhdl-kernel	2.94s	0.68 %
Module Path Delay Computing	1.14s	0.26 %
Hsim_Interpreter	419.60ms	0.10 %
Hsim_Elab	119.89ms	0.03 %
Design_Elab	69.93ms	0.02 %
Profiling	27.67s	6.41 %
DProf	27.67s	6.41 %
CONSTRANT	8.56s	1.98 %
VHDL	7.38s	1.71 %
process	7.20s	1.67 %
PLI/DPI/DirectC	4.47s	1.03 %
total	431.97s	100%

The following are the components displayed in the time summary view:

CONSTRAINT

The CPU time needed to solve and simulate the SystemVerilog constraint blocks.

Also the CPU time used for calls to the `randomize()` method, like in this example, are included in this component. These calls to `randomize()` are taking most of the CPU time reported for this component, and in turn this component used most of the CPU time.

KERNEL

The CPU time needed by the VCS kernel. This CPU time is separate from the CPU time needed to simulate your Verilog or SystemVerilog, VHDL, SystemC, or C or C++ code for your design and testbench.

VERILOG

The CPU time needed by VCS to simulate this example's SystemVerilog code, which is a program block. For Verilog and SystemVerilog there are sub-components. In this example view, there is only one sub-component named Program.

This example consists of a SystemVerilog program block that used 22.03% of the CPU time.

Possible other sub-components are Module, Interface, UDP, and Assertion, for the CPU time used by Verilog and SystemVerilog definitions for `module`, `interface`, user-defined primitive, package and assertion.

Other possible components are as follows:

DEBUG

The CPU time needed by VCS to simulate this example with the debugging capabilities of Verdi and the UCLI or to write a simulation history VCD or FSDB file.

Value Change Dumping

The CPU time needed by VCS to write a simulation history VCD or VPD file. This component is always accompanied by the DEBUG component. This component has the following sub-components:

- **VPD**

The CPU time needed by VCS to write a VPD file.

- **VCD**

The CPU time needed by VCS to write a VCD file.

VHDL

For VCS only, the CPU time needed to simulate the VHDL code design.

PLI/DPI/DirectC

The CPU time needed by VCS to simulate the C/C++ in a PLI, DPI, or DirectC application.

HSIM (Hybrid Simulation)

This is about the CPU time used by HSOPT (Hybrid Simulation Optimization). The HSIM bucket indicates the CPU time consumption of design constructs that are optimized by HSOPT. It has become prominent in GLS design/RTL.

The HSIM cost is more with GLS design because most constructs are optimized by HSOPT. But it cannot be zero because there are some global HSIM activities.

COVERAGE

The CPU time needed for functional coverage (testbench and assertion coverage). Code coverage is not part of this component.

SystemC

The CPU time needed for SystemC simulation.

Note:

If the following categories report a high percentage of time, contact the Synopsys support team (vcs_support@synopsys.com):

- Kernel
- HSIM

Memory Summary Report

If you select the Memory Summary view in the *View* field in the left pane, then click the **GO** button.

The following figure displays an example of the memory summary view:

Figure 15 Memory Summary View

Memory Summary View (Simulation time:376676447)		
Component	Size	Percentage
CONSTRAINT	39.09 MB	6.78 %
KERNEL	14.75 MB	2.56 %
vhdl-kernel	83.01 KB	0.01 %
Design_Elab	1.55 KB	0.00 %
Hsim_Elab	972 B	0.00 %
Garbage Collection	347 B	0.00 %
VHDL	6.93 MB	1.20 %
HSIM	4.84 MB	0.84 %
PLI/DPI/DirectC	3.00 MB	0.52 %
VERILOG	44.70 MB	7.75 %
Module	41.91 MB	7.26 %
Package	2.57 MB	0.45 %
Functional Coverage	189.26 KB	0.03 %
Function Coverage Kernel	90.80 KB	0.02 %
Interface	29.81 KB	0.01 %
Program	500 B	0.00 %
Profiling	4.27 KB	0.00 %
DProf	4.27 KB	0.00 %
PLI/DPI	72 B	0.00 %
ASSERTION_KERNEL	8 B	0.00 %
Anonymous	26.22 MB	4.54 %
SNPS Memory Pool	188.25 MB	32.63 %
Dynamic Memory Pool	138.00 MB	23.92 %

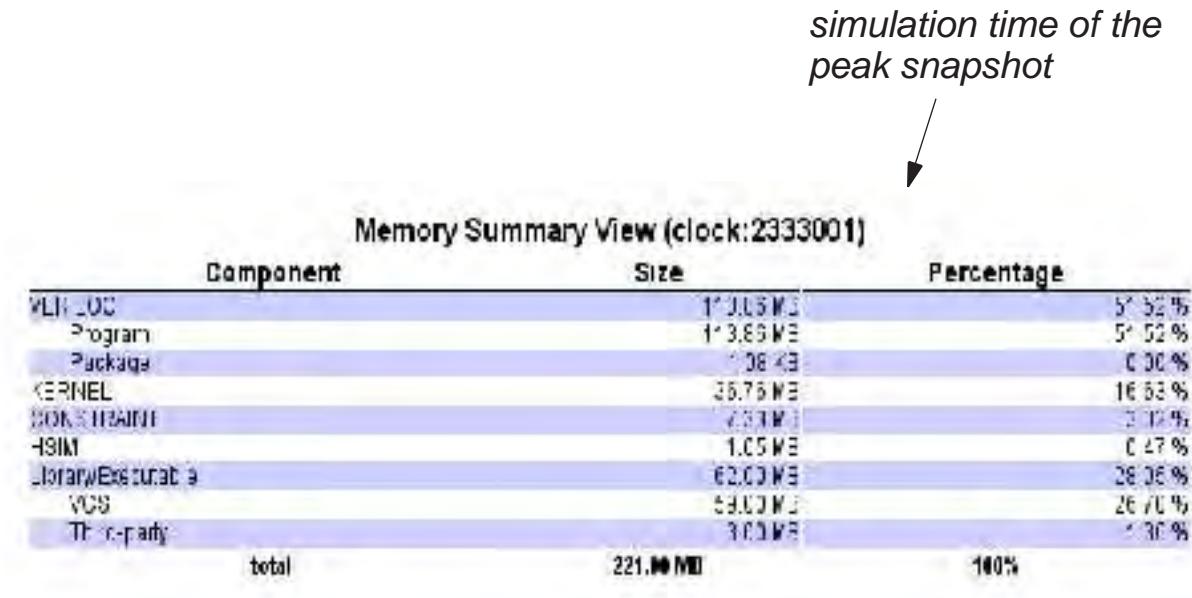
The following are the components displayed in the memory summary view:

In the left pane, in the *Database* field, select the simprofile_dir profile database directory again. This adds the following fields to the left pane:

- the *View* field that is at the default selection of the Memory Summary view
- the *Snapshot* field that is at the default selection of the peak machine memory snapshot. The example shows the 33rd snapshot at simulation time 2000000.

Then click the **GO** button.

Figure 16 The Machine Memory Summary View for the Peak Snapshot



Components, in this view, are consumers of machine memory during simulation. This view reports the amount of machine memory used by each component and their percentage of the total machine memory used in the snapshot. In this example view, this is the peak snapshot.

In this snapshot, the preponderance of the machine memory is used by the VERILOG and KERNEL components.

The components in this view are as follows:

VERILOG

The machine memory needed by VCS to simulate this example's SystemVerilog code, which is a program block at the peak snapshot. The following are the sub-components:

- **Program**

The machine memory needed to simulate the SystemVerilog program block in the code example.

- **Package**

Usually the machine memory needed to simulate a SystemVerilog package.

In this case this is an anomaly; reporting a small amount of machine memory for a package when there is no package in the code example. You can ignore these anomalies.

The other possible sub-components are Module, Interface, UDP, and Assertion.

KERNEL

The machine memory used by the VCS kernel. This is separate machine memory from the machine memory needed to simulate the code in the code example.

CONSTRAINT

The machine memory needed to solve and simulate the SystemVerilog constraint blocks, but also counted in this component are calls to the `randomize()` method.

HSIM

This is about the machine memory used by Hybrid Simulation Optimizations (HSOPT). Memory profiling is not impacted much because dynamic memory allocation is not expected in HSIM except during the initialization.

Library/Executable

This is the sum of the VCS and Third-party sub-components.

- **VCS**

Memory consumed by VCS executable and library. It consists of `simv` and all the libraries provided by VCS. Most of these libraries are located in `$VCS_HOME/lib`.

- **Third-party**

Memory consumed by user-provided libraries and global libraries, such as `libc.so`. This includes all other libraries.

COVERAGE

This component is for functional coverage. A small percentage of machine memory is reported here even though there is no functional coverage code in the design. This is the machine memory needed for functional coverage enabling optimizations, which are default optimizations.

Code coverage is not reported in this component. The machine memory used for code coverage is in the VERILOG (or VHDL) components.

Other possible components, if you change the example source code and entered different options, are as follows:

DEBUG

This component is for the machine memory needed by VCS to simulate this example with the debugging capabilities of Verdi and the UCLI or to write a simulation history VCD or FSDB file.

PLI/DPI/DirectC

This component is for the machine memory needed by VCS to simulate the C/C++ code in a design.

SystemC

The machine memory needed for SystemC simulation.

For VHDL and mixed-HDL designs, there is an additional possible component:

VHDL

The machine memory needed to simulate VHDL code of a design.

SNPS Memory Pool

SNPS Memory Pool has a sub category, Dynamic Memory Pool, which is a machine memory consumed by the dynamic objects in your design. The memory under SNPS Memory Pool apart from Dynamic Memory pool is the memory consumed for the tool implementation.

You can reach out to the Synopsys Support center (vcs_support@synopsys.com) if the memory reported under SNPS Memory Pool minus the memory reported under Dynamic Memory Pool is huge.

Anonymous

Memory consumed by the third-party libraries. If this category consumes huge percentage of memory, it is recommended to review DPI/PLI/DirectC or other similar code which might have a third-party library usage.

Note:

If the following categories report a high percentage of memory, contact the Synopsys support team (vcs_support@synopsys.com):

- Kernel
- HSIM
- SNPS Memory Pool

The following are the examples of these reports based on the SystemVerilog code as shown in [Example 29](#):

Example 29 Profiler SystemVerilog Code Example

```
program tb_top;
    logic [255:0]          Squeue_data_info[$];
    logic [255:0]          temp;

    class PACKET;
        rand reg [255:0] packet_val;
```

```
endclass

initial
begin

    for(int y = 0 ; y < 1000 ; y++)
        begin
            PACKET packet_inst;

            packet_inst = new();
            packet_inst.randomize();
            #1;

            Squeue_data_info.push_back(packet_inst.packet_val);
            #1;

        end

    repeat(10)
        $display("DEBUG==> Pushed 1000");

    for(int y = 0 ; y < 500 ; y++)
        begin

            #1;
            temp = Squeue_data_info.pop_front();
            #1;

        end
    repeat(10)
        $display("DEBUG==> Popped 500");

    for(int y = 0 ; y < 10000 ; y++)
        begin
            PACKET packet_inst;

            packet_inst = new();
            packet_inst.randomize();
            #1;

            Squeue_data_info.push_back(packet_inst.packet_val);
            #1;

        end

    repeat(10)
        $display("DEBUG==> Pushed 10000");
```

```
for(int y = 0 ; y < 5000 ; y++)
begin
    PACKET packet_inst_2;

#1;
temp = Squeue_data_info.pop_front();
#1;

end
repeat(10)
$display("DEBUG==> Popped 5000");

for(int y = 0 ; y < 100000 ; y++)
begin
    PACKET packet_inst;

packet_inst = new();
packet_inst.randomize();
#1;

Squeue_data_info.push_back(packet_inst.packet_val);
#1;

end

repeat(10)
$display("DEBUG==> Pushed 100000");

for(int y = 0 ; y < 50000 ; y++)
begin
    PACKET packet_inst_2;

#1;
temp = Squeue_data_info.pop_front();
#1;

end
repeat(10)
$display("DEBUG==> Popped 50000");

for(int y = 0 ; y < 1000000 ; y++)
begin
    PACKET packet_inst;

packet_inst = new();
packet_inst.randomize();
#1;

Squeue_data_info.push_back(packet_inst.packet_val);
#1;
```

```
    end

repeat(10)
$display("DEBUG==> Pushed 1000000");

for(int y = 0 ; y < 500000 ; y++)
begin
    PACKET packet_inst_2;

    #1;
    temp = Squeue_data_info.pop_front();
    #1;

end
repeat(10)
$display("DEBUG==> Popped 500000");

$finish;

end
endprogram
```

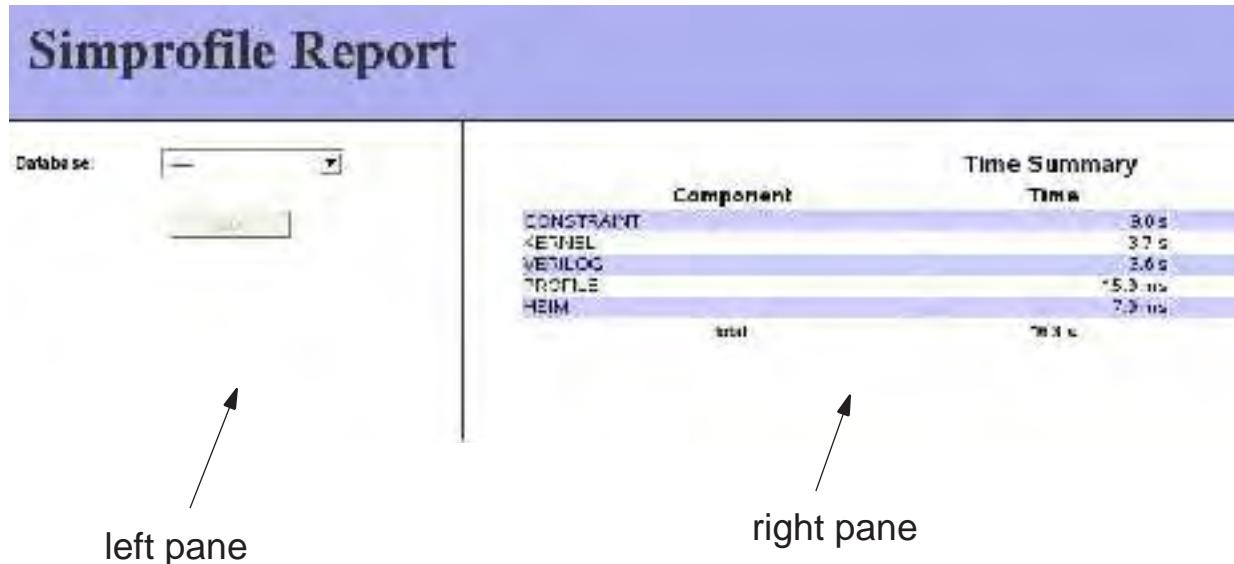
This code was compiled and simulated for CPU time profile information with the following command lines:

```
% vcs smart_queue.v -simprofile -sverilog
% simv -simprofile time
```

The profrpt command line is as follows:

```
profrpt simprofile_dir -view time_all -timeline ALL
```

Figure 17 The profileReport.html File for CPU Time Profile Information



You can select the only database in this example so far, the `simprofile_dir` directory. Doing so adds the `View` field to the left pane and the default view, which in this case is the Time Summary view. Then, click the GO button.

Figure 18 The Right Pane of the simprofileReport.html file for CPU Time Summary Information

Time Summary View		
Component	Time	Percentage
CONSTRAINT	9.03 s	55.40 %
KERNEL	3.65 s	22.42 %
VERILOG	3.59 s	22.03 %
Program	3.59 s	22.03 %
total	16.30 s	100%

If you select the Time Module view in the View field in the left pane, then click the **GO** button again. The right pane changes to show the following view:

Figure 19 The CPU Time Module View

Time Module View					
Module	Inclusive Time	Percentage	Exclusive Time	Percentage	
*tb_top	1.90 s	24.54 %	1.90 s	24.54 %	
NoName	1.90 s	24.54 %	1.90 s	24.54 %	
total	1.90 s	24.54 %	1.90 s	24.54 %	

Page: 1

click here

As explained earlier, modules not only include Verilog and SystemVerilog modules, but can also include SystemVerilog programs and interfaces, and for VHDL, it can also include entity/architectures.

Figure 20 The Expanded CPU Time Module View

Time Module View		
Module	Time	Percentage
*tb_top	3.59 s	22.03 %
NoName	3.45 s	21.16 %
total	3.59 s	22.03 %

Page: 1

In this example view, program block `tb_top` used 3.59 seconds of CPU time, which was 22.03% of the CPU time used by the simulation.

The program name is a hypertext link to expand the display in this view. If you click the hypertext link, you can see the scopes inside the program block.

The scopes inside the program block are `begin-end` blocks that are not named. Thus, `profrpt` calls them all `NoName`. These `begin-end` blocks use most of the CPU time used by the program block. In this example view, `NoName` is not a hypertext link.

Other possible scopes inside a module are fork-join blocks and user-defined tasks and functions.

If you select the Time Construct view in the View field in the left pane, then click the GO button again. The right pane changes to show the following view:

Figure 21 The CPU Time Construct View

Time Construct View		
Name	Time	Percentage
▶Initial	3.45 s	21.16 %
total	3.51 s	21.55 %

Page: 1 click here



In this example view, the only construct is an initial block. This initial block uses 21.55% of the CPU time.

For Verilog and SystemVerilog, the constructs in this view can include `initial` procedures, `always` procedures (including the SystemVerilog `always` procedures such as `always_comb`), SystemVerilog `final` procedures, user-defined tasks, and user-defined functions.

For VHDL, the constructs in this view are processes in architectures.

The `initial` keyword is a hypertext link to expand the display in this view. If you click the hypertext link, you can see the scopes inside the `initial` procedure.

Figure 22 The Expanded CPU Time Construct View

Time Construct View		
Name	Time	Percentage
▶Initial	3.45 s	21.16 %
NoName	3.45 s	21.16 %
total	3.51 s	21.55 %

Page: 1

In this example view, the scopes inside the `initial` procedure are begin-end blocks that are not named. Thus, the profiler calls them all `NoName`. These begin-end blocks use most of the CPU time used by the program block. In this example view, `NoName` is not a hypertext link.

If you select the Time Instance view in the View field in the left pane, then click the GO button again. The right pane changes to show the following view:

Figure 23 The CPU Time Instance View

Time Instance View					
Instance	Inclusive Time	Percentage	Exclusive Time	Percentage	
tb_top	3.59 s	22.03 %	3.59 s	22.03 %	
total	3.59 s	22.03 %	3.59 s	22.03 %	

Page: 1

This view shows CPU times and percentages for the instances in the design. These are instances of Verilog and SystemVerilog modules and also instances of SystemVerilog interfaces and VHDL entity/architectures.

This view shows for an instance the inclusive and exclusive time and percentage values.

The inclusive time and percentage is for the percentage of CPU time used by this instance and all instances that are hierarchically under it in the design hierarchy.

The exclusive time and percentage is for the CPU time used by this instance alone, not counting the instances that are hierarchically under this instance.

In the above figure, there is only one instance of program tb_top. Thus, the inclusive and exclusive values are the same, which are 3.59 seconds and 22.03% of the CPU time.

The instance name tb_top is not a hypertext link.

There is no PLI, DPI, or DirectC code. Therefore, there is no information in the Pli/DPI/ DirectC view. There is also no information in the Dynamic Timeline view because this view is for machine memory information and you do not collect machine memory profile information in the profile database.

You can now simulate for machine memory profile information as shown:

```
% simv -simprofile mem
```

The profrpt command line is as follows:

```
% profrpt simprofile_dir -view mem_all -timeline ALL
```

The -timeline option specifies the snapshot reports.

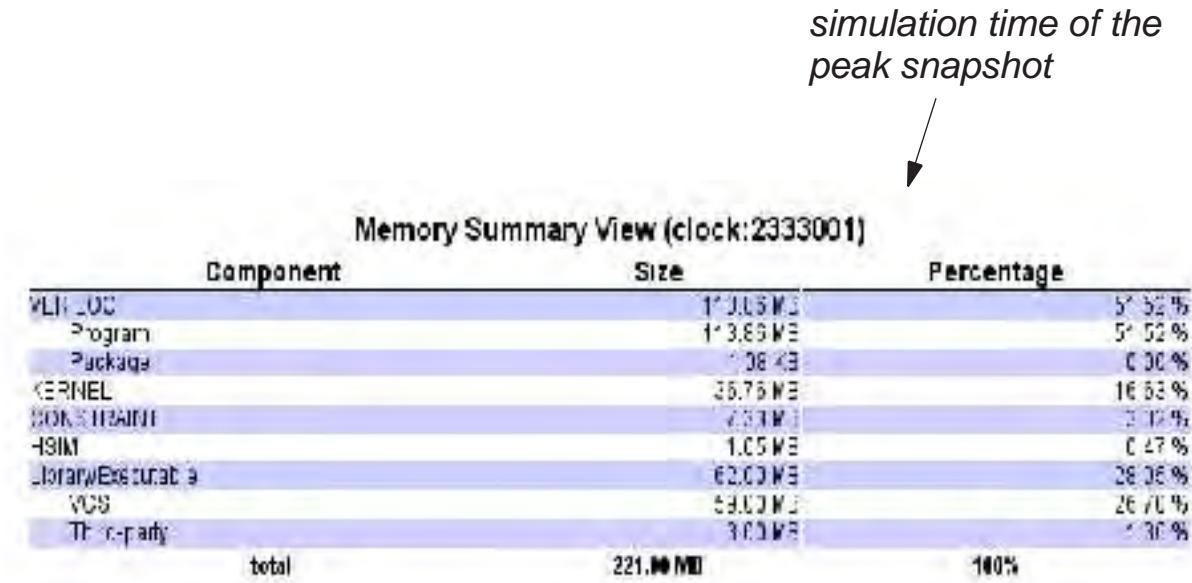
The profile report generator, profrpt, rewrites the profileReport.html file for the machine memory information. Therefore, you should re-open this file.

In the left pane, in the *Database* field, select the simprofile_dir profile database directory again. This adds the following fields to the left pane:

- the *View* field that is at the default selection of the Memory Summary view
- the *Snapshot* field that is at the default selection of the peak machine memory snapshot. The example shows the 33rd snapshot at simulation time 2000000.

Then click the **GO** button.

Figure 24 The Machine Memory Summary View for the Peak Snapshot



Components, in this view, are consumers of machine memory during simulation. This view reports the amount of machine memory used by each component and their percentage of the total machine memory used in the snapshot. In this example view, this is the peak snapshot.

In this snapshot, the preponderance of the machine memory is used by the VERILOG and KERNEL components.

So far you have looked at the machine memory summary view for the peak snapshot. There is a summary view for other snapshots.

For example, if you select the 10th snapshot, as shown:

Figure 25 Selecting the 10th Snapshot

Database:	<input type="text" value="simprofile_dir"/>
View:	<input type="text" value="Memory Sumr"/>
Snapshot:	<input type="text" value="#10 (clock:193131)"/>
GO	

Then click the **GO** button, the right pane shows the machine memory summary view for this snapshot.

Figure 26 The Machine Memory Summary View for the 10th Snapshot

Memory Summary View (clock:193131)		
Component	Size	Percentage
KERNEL	29.99 MB	67.64 %
VERILOG	11.59 MB	26.14 %
Program	11.57 MB	26.09 %
Package	153.64 KB	0.34 %
HSIM	2.01 MB	4.53 %
CONSTRAINT	768.03 KB	1.69 %
COVERAGE	316 B	0.00 %
total	44.33 MB	100%

This view shows the machine memory used by the various components in the 10th snapshot.

Now, back in the left pane, you can return to the peak snapshot, the 33rd in the *Snapshot* field and select the Memory Module view in the *View* field, then click the **GO** button. The right pane changes to the machine memory module view for the peak snapshot.

Figure 27 The Machine Memory Module View for the Peak Snapshot

Memory Module View (clock:2000000)		
Module	Size	Percentage
►tb_top	116.14 MB	73.92 %
total	116.78 MB	74.33 %
Page: 1		
click here		

As explained earlier, modules not only include Verilog and SystemVerilog modules, but can also include SystemVerilog programs and interfaces, and for VHDL, include entity/architectures.

In this example view, program block `tb_top` used 116.14 MB of machine memory, which is 74.33% of the machine memory used to simulate the peak snapshot.

The program name is a hypertext link to expand the display in this view. If you click the hypertext link, you can see the scopes inside the program block.

Figure 28 The Expanded Machine Memory Module View for the Peak Snapshot

Memory Module View (clock:2000000)		
Module	Size	Percentage
►tb_top	116.14 MB	73.92 %
NoName	116.14 MB	73.92 %
total	116.78 MB	74.33 %
Page: 1		

In this example view, the scope inside the program block is a begin-end block that is not named. Therefore, `profrpt` calls it `NoName`. This begin-end block uses most of the machine memory used by the program block. In this example view, `NoName` is not a hypertext link.

Other possible scopes inside a module are fork-join blocks and user-defined tasks and functions.

There is a machine memory module view for each snapshot.

If in the left pane, you select the Memory Constant view and then click the GO button, the right pane changes to the machine memory construct view for the peak snapshot.

Figure 29 The Machine Memory Construct View for the Peak Snapshot

Memory Construct View (clock:2000000)		
Name	Size	Percentage
▶Initial	116.14 MB	73.92 %
total	116.14 MB	73.92 %
Page: 1		
click here		

In this example view, the only construct is an initial block. This initial block uses, at the peak snapshot, 116.14 MB of machine memory, which is 73.92% of the total machine memory use at the peak snapshot.

For Verilog and SystemVerilog, the constructs in this view can include `initial` procedures, `always` procedures (including the SystemVerilog `always` procedures such as `always_comb`), SystemVerilog `final` procedures, user-defined tasks, and user-defined functions.

For VHDL, the constructs in this view are processes in the architectures.

The `initial` keyword is a hypertext link to expand the display in this view. If you click hypertext link, you can see the scope inside the `initial` procedure.

Figure 30 The Expanded Machine Memory Construct View for the Peak Snapshot

Memory Construct View (clock:2000000)		
Name	Size	Percentage
▶Initial	116.14 MB	73.92 %
NoName	116.14 MB	73.92 %
total	116.14 MB	73.92 %
Page: 1		

In this example view, the scope inside the `initial` procedure is a begin-end block that is unnamed. Therefore, `prof rpt` calls it `NoName`. This begin-end block use all of the machine memory used by the `initial` procedure. In this example view, `NoName` is not a hypertext link.

There is a machine memory construct view for each snapshot.

If you select the Memory Instance view in the *View* field in the left pane, then click the **GO** button again, the right pane changes to show the following view:

Figure 31 The Machine Memory Instance View for the Peak Snapshot

Memory Instance View (clock:2000000)					
Instance	Inclusive Size	Percentage	Exclusive Size	Percentage	
►tb_top	116.14 MB	73.92 %	116.14 MB	73.92 %	
total	116.16 MB	73.93 %	116.16 MB	73.93 %	
Page: 1					

This view shows the machine memory used and percentages for the instances in the design at the peak snapshot. These are instances of Verilog and SystemVerilog modules and also instances of SystemVerilog programs and interfaces and VHDL entity/architectures.

This view shows for an instance the inclusive and exclusive machine memory used and percentage values for the peak snapshot.

The inclusive machine memory amount and percentage is the percentage of machine memory used by this instance and all instances that are hierarchically under it in the design hierarchy.

The exclusive machine memory amount and percentage is the machine memory used by this instance alone, not counting the instances that are hierarchically under this instance.

In this example view, there is only one instance of program `tb_top`. So, the inclusive and exclusive values are the same, which are 116.16 MB and 73.93% of the machine memory.

The instance name `tb_top` is not a hypertext link.

Like the machine memory summary, module, and construct views, there is a machine memory instance view for each snapshot.

In this example view, there is no PLI, DPI, or DirectC code so there is no information in the Pli/DPI/DirectC view.

If you select the Dynamic Memory view in the *View* field in the left pane, the *Snapshot* field automatically changes to snapshot #997.

Figure 32 The Left Pane After Selecting the Dynamic Memory View



Snapshot #997 is the peak snapshot for dynamic objects.

If you click the GO button again, the right pane changes to show this view.

Figure 33 The Dynamic Memory View for the Peak Snapshot

Dynamic Memory View (clock: 1998612)				
Dynamic Object	Instance Number	Memory	Percentage	
►PACKET	31552	29.68 MB	87.12 %	
►SmartQueue	N/A	4.39 MB	12.88 %	
►String	1	48 B	0.00 %	
		34.07 MB	100%	

Page: 1 click here

The peak machine memory dynamic view shows the machine memory that was used by dynamic objects at their peak machine memory consumption. This is not the peak machine memory consumption of the entire design and testbench, just the peak machine memory consumption of their dynamic objects.

The dynamic objects include dynamic and associative arrays and queues.

In this view is a SystemVerilog queue and string.

Smart Queues are a concept in the *OpenVera Language Reference Manual: Testbench*. The `profrpt` profile report generator lists SystemVerilog queues as Smart Queues. In this example view, there is only one SystemVerilog queue. It is declared as follows:

```
logic [255:0] Squeue_data_info[$];
```

Squeue_data_info, in this peak machine memory dynamic view, uses 4.39 MB of machine memory, which is 12.88% of the machine memory used at this peak by this queue.

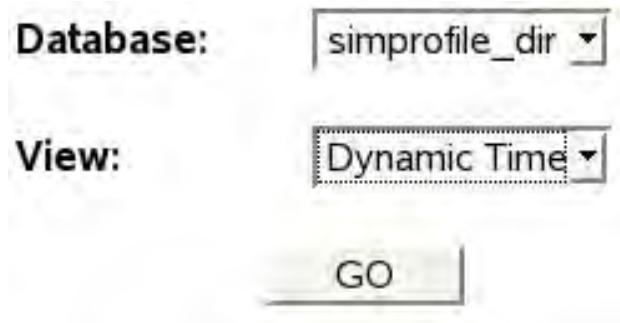
The profrpt profile report generator cannot report the number of instances of this queue.

The string entry is for a small amount of machine memory and can be ignored.

There is a dynamic object machine memory view for each snapshot.

If you select the Dynamic Timeline view in the View: field in the left pane, the Snapshot: field disappears.

Figure 34 The Left Pane After Selecting the Dynamic Timeline View



If you click the GO button again, the right pane changes to show this view.

Figure 35 The Machine Memory Dynamic Timeline View

Column for snapshots

	Clock	Assoc	Aarry	Dynamic Memory Timeline							Display percentage
				Dynamic Array	Smart Queue	Event	Mailbox	String	Class	Total	
#0	0		0 B	0 B	0 B	0 B	0 B	48 B	0 B	48 B	
#1	1		0 B	0 B	288 B	0 B	0 B	48 B	104 B	440 B	
#2	36		0 B	0 B	200 B	0 B	0 B	48 B	1.93 KB	2.17 KB	
#3	1182		0 B	0 B	3.83 KB	0 B	0 B	48 B	60.13 KB	64.01 KB	
#4	1246		0 B	0 B	3.83 KB	0 B	0 B	48 B	63.38 KB	67.26 KB	
#5	1314		0 B	0 B	3.83 KB	0 B	0 B	48 B	66.84 KB	70.71 KB	
#6	1384		0 B	0 B	3.83 KB	0 B	0 B	48 B	70.39 KB	74.27 KB	
#7	1458		0 B	0 B	3.83 KB	0 B	0 B	48 B	74.15 KB	78.02 KB	
#8	1536		0 B	0 B	3.83 KB	0 B	0 B	48 B	78.11 KB	81.98 KB	
#9	1618		0 B	0 B	3.83 KB	0 B	0 B	48 B	82.27 KB	86.15 KB	
#10	1704		0 B	0 B	3.83 KB	0 B	0 B	48 B	86.64 KB	90.52 KB	
#11	1794		0 B	0 B	3.83 KB	0 B	0 B	48 B	91.21 KB	95.09 KB	
#12	1888		0 B	0 B	3.83 KB	0 B	0 B	48 B	95.98 KB	99.86 KB	
#13	1921		0 B	0 B	11.41 KB	0 B	0 B	48 B	97.61 KB	109.06 KB	
#14	2104		0 B	0 B	7.58 KB	0 B	0 B	48 B	106.95 KB	114.58 KB	
#15	2218		0 B	0 B	7.58 KB	0 B	0 B	48 B	112.74 KB	120.37 KB	
#16	2338		0 B	0 B	7.58 KB	0 B	0 B	48 B	118.84 KB	126.46 KB	
#17	2464		0 B	0 B	7.58 KB	0 B	0 B	48 B	125.23 KB	132.86 KB	
#18	2596		0 B	0 B	7.58 KB	0 B	0 B	48 B	131.94 KB	139.56 KB	
#19	2734		0 B	0 B	7.58 KB	0 B	0 B	48 B	138.95 KB	146.57 KB	

Page:

1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28 29 30 31 32 33 34 35 36 37 38 39 40 41 4

hypertext links for page numbers

This view unlike the previous machine memory views is not for a specific snapshot, but for all snapshots in the profile database.

In this example view, there are multiple pages. The page numbers at the bottom of the view are hypertext links to show the different pages. In this view, there are many pages because there are hundreds of snapshots in the database.

Notice that there is a significant increase in the machine memory for the queue in snapshot 13.

You can scroll to the right and click page 50, which includes the dynamic object machine memory peak snapshot, and the right pane changes to show this page.

Figure 36 Page 50 of the Machine Memory Dynamic Timeline View

Dynamic Memory Timeline								Display percentage
Clock	Assoc-Aarry	Dynamic Array	Smart Queue	Event	Mailbox	String	Class	Total
#980 1936896	0 B	0 B	4.39 MB	0 B	0 B	48 B	1.83 MB	6.22 MB
#981 1936896	0 B	0 B	4.39 MB	0 B	0 B	48 B	1.52 MB	5.91 MB
#982 1936896	0 B	0 B	4.39 MB	0 B	0 B	48 B	1.23 MB	5.61 MB
#983 1936896	0 B	0 B	4.39 MB	0 B	0 B	48 B	968.30 KB	5.33 MB
#984 1936896	0 B	0 B	4.39 MB	0 B	0 B	48 B	695.20 KB	5.07 MB
#985 1936896	0 B	0 B	4.39 MB	0 B	0 B	48 B	435.70 KB	4.81 MB
#986 1936896	0 B	0 B	4.39 MB	0 B	0 B	48 B	189.21 KB	4.57 MB
#987 1945230	0 B	0 B	4.39 MB	0 B	0 B	48 B	423.31 KB	4.80 MB
#988 1950072	0 B	0 B	4.39 MB	0 B	0 B	48 B	669.20 KB	5.04 MB
#989 1955156	0 B	0 B	4.39 MB	0 B	0 B	48 B	927.37 KB	5.29 MB
#990 1960494	0 B	0 B	4.39 MB	0 B	0 B	48 B	1.17 MB	5.56 MB
#991 1966098	0 B	0 B	4.39 MB	0 B	0 B	48 B	1.45 MB	5.84 MB
#992 1971982	0 B	0 B	4.39 MB	0 B	0 B	48 B	1.74 MB	6.13 MB
#993 1978160	0 B	0 B	4.39 MB	0 B	0 B	48 B	2.05 MB	6.43 MB
#994 1984648	0 B	0 B	4.39 MB	0 B	0 B	48 B	2.37 MB	6.76 MB
#995 1991460	0 B	0 B	4.39 MB	0 B	0 B	48 B	2.71 MB	7.09 MB
#996 1998612	0 B	0 B	4.39 MB	0 B	0 B	48 B	3.06 MB	7.45 MB
#997 1998612	0 B	0 B	4.39 MB	0 B	0 B	48 B	29.68 MB	34.07 MB

Display of Parameterized Class Functions and Tasks in Profiling Reports

The reports generated by the unified simulation profiler display functions and tasks of parameterized classes that are defined in a package or in the global scope.

For example:

```
class vector #(int size = 1);
  rand bit [size-1:0] a;
  bit [size-1:0] a_arry[];

  constraint num { a > 1; }

  task obj_disp();
    $display("%0d : Object v%0d : %p", $time, size, this);
  endtask

  function void disp_count();
    int i;
    for (i=0; i<1000000; i++) begin
```

```
    this.randomize();
    a_arry[i] = a;
end
endfunction
endclass

program prog;
vector #(2) v2 = new;
vector #(3) v3 = new;
vector #(4) v4 = new;

initial begin
    v2.disp_count();
    v3.disp_count();
    v4.disp_count();
    v2.obj_disp();
    v3.obj_disp();
    v4.obj_disp();
end
endprogram
```

In the above example, the parameterized class `vector` is defined in the global scope. In the profiling report, the instance `_global_` is displayed in the Time Instance View and the class function `disp_count()` is displayed in the Time Module View.

Note:

For objects of the same parameterized class, profiling data for their functions and tasks are combined and displayed as a single entry.

The Display of Parameterized Class Functions and Tasks in the Profiling Reports is the Time Module View.

Figure 37 Time Module View

Time Module View				
Module	Inclusive Time	Percentage	Exclusive Time	Percentage
global	1.23 s	2.65 %	1.23 s	2.65 %
Function	1.23 s	2.65 %	1.23 s	2.65 %
disp_count	1.23 s	2.65 %	1.23 s	2.65 %
<0.50 %	0.00us	0.00 %	0.00us	0.00 %
total	1.23 s	2.65 %	1.23 s	2.65 %

Construct Information

Construct Name:	disp_count
Time	1.23 s
Construct Type:	Function
Parent Module:	global
Source Information:	TESTS\vc1412\key\acct\B439\simprofile\task\mtestv14-20

Hypertext Links to the Source Files

The pathnames of source files in any of the HTML views are hypertext links. Clicking on one of these links opens a new window of the browser to display that source file. This section describes and illustrates this feature.

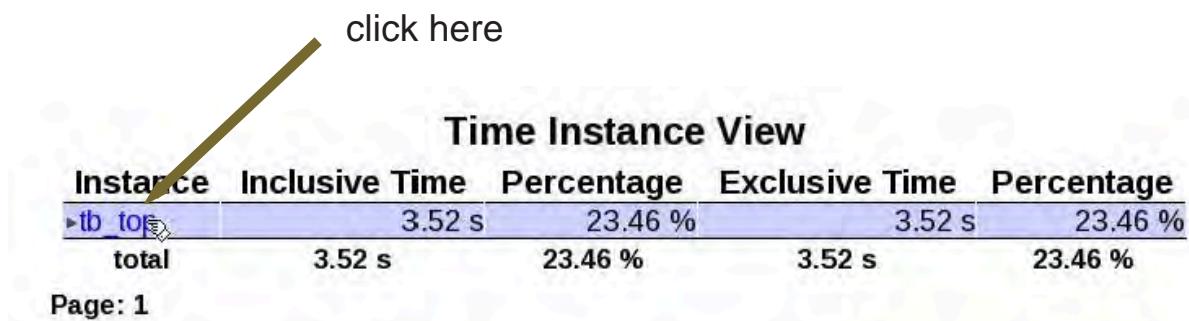
Note:

The hypertext link to the source files feature is not implemented for SystemC/C/C++ source files.

To use this feature, do the following:

1. Compile a design with the `-simprofile` option.
2. Run the simulation with the `-simprofile time/mem/time+mem` option and keyword argument to enable VCS to collect time/memory/time and memory profile information.
3. Run the `profprt` utility to create the HTML views.
4. Open the `profileReport.html` file.
5. Select a profile database in the left pane.
6. Select the Time Instance view.

Figure 38 Time Instance View



A screenshot of a web-based profiler interface showing a table titled "Time Instance View". The table has six columns: "Instance", "Inclusive Time", "Percentage", "Exclusive Time", and "Percentage". There are two rows: the first row is for the top-level module ("tb_top") and the second row is for the total instance. A yellow arrow points from the text "click here" to the first row of the table.

Instance	Inclusive Time	Percentage	Exclusive Time	Percentage
tb_top	3.52 s	23.46 %	3.52 s	23.46 %
total	3.52 s	23.46 %	3.52 s	23.46 %

Page: 1

7. Click an instance in the view.

This adds this information about the instance to the bottom of the HTML page:

Instance Name	a reiteration of the instance name
Exclusive Time	the CPU time used by the instance
Exclusive Percentage	the percentage of the total CPU time that was used by this instance
Inclusive Time	the CPU time used by the instance and all instances under it in the design hierarchy
Inclusive Percentage	the percentage of the total CPU time that was used by this instance and all instances under it in the design hierarchy
Master Module	the name of the top-level module in the design hierarchy
Child Instance Number	the number of instances under this instance in the design hierarchy
Source Information	the path to the source file and line number of the header of the module, interface, or program definition

The *Source Information* is in blue text in this expanded view because it is a hypertext link to the source code, as shown in Figure 39.

Figure 39 Time Instance View Expanded

Instance Information		
Instance Name		tb_top
Exclusive Time		3.52 s
Exclusive Percentage	click here	23.46 %
Inclusive Time		3.52 s
Inclusive Percentage		23.46 %
Master Module		tb_top
Child Instance Number		0
Source Information	/file_system/big_design/VCS_user_files/smart_queue.v:2	

In this example view, source information for the instance is the program definition for instance tb_top in /file_system/big_design/VCS_user_files/smart_queue.v on line 2.

- ▶ Click the blue path name of the source file and line number, this is a hypertext link. The browser opens a new window to display the source file, as shown in Figure 40.

To display the source file in new window, you should open the source file with Firefox 3.* web browser.

Figure 40 New Source File window



```

02 program tb_top;
03
04 logic [255:0] Squeue_data_info[$];
05 logic [255:0] temp;
06
07 class PACKET;
08   rand reg [255:0] packet_val;
09 endclass
10
11
12
13 initial
14 begin
15
16   for(int y = 0 ; y < 1000 ; y++)
17     begin
18       PACKET packet_inst;
19
20       packet_inst = new();
21       packet_inst.randomize();
22       #1;
23
24       Squeue_data_info.push_back(packet_inst.packet_val);
25       #1;
26
27     end
28
29 repeat(10)
30   $display("DEBUG====> Pushed 1000");
31

```

The program header is `program tb_top;`, in line 2, has a lighter background.

The lines in this source file window also shows the line numbers.

SystemC Views

The following views are from a SystemC co-simulation after running the `profprt` profile report generator.

The code examples for these views is in the `$VCS_HOME/doc/examples/systemc/vcs/vcs_profiler`. There is a minor change to one of the files to show the name for a begin-end block in `sv_mod.sv` as follows:

```

module sv_mod(iclk);
  input iclk;
  static int count=0;
  int i;

```

```

always @(posedge iclk)
begin: bel
  count++;
  $display("SV:Executing on pos edge @%d",count);
  for(i=0;i<1000*100000000;i++)
  ;
end

endmodule

```

Figure 41 The Time Summary View

Time Summary View		
Component	Time	Percentage
VERILOG	282.53 s	77.85 %
Module	282.53 s	77.85 %
Package	999.89us	0.00 %
SystemC	79.85 s	22.00 %
KERNEL	505.94 ms	0.14 %
HSIM	12.00 ms	0.00 %
PLI/DPI/DirectC	999.89us	0.00 %
total	362.90 s	100%

As you would expect from reading the SystemVerilog and SystemC files in this example, most of the CPU time was used by the SystemVerilog and SystemC modules.

A small amount of CPU time was used by The VCS kernel.

A small amount of CPU time was reported used by a SystemVerilog package, writing a VPD file, and PLI, DPI, or a DirectC application, even though these are not present in this example. Notice that they all take 0.00% of the CPU time. You can ignore these anomalies.

If our example writes a VPD file or contains a PLI, DPI, or DirectC application, you might see significant values for the CPU times in this view.

Figure 42 The Time Module View

Time Module View		
Module	Time	Percentage
► sv_mod	282.53 s	77.85 %
► sc_mod	79.85 s	22.00 %
► sv_top	2.00 ms	0.00 %
► std	999.89us	0.00 %
► _global_	0.00us	0.00 %
total	282.53 s	77.85 %

Page: 1

This view shows the CPU times and percentages for the main consumers of CPU times, the `sv_mod` SystemVerilog module and the `sc_mod` SystemC module. A small amount of time is used by the top-level module `sv_top`.

The `std` and `_global_` modules are from the internals of VCS and when seen in this view should be ignored.

If you click these module names, the view expands to show scopes inside these module definitions.

Figure 43 The Expanded Time Module View

Time Module View		
Module	Time	Percentage
▼ sv_mod	282.53 s	77.85 %
be1	282.53 s	77.85 %
NoName	0.00us	0.00 %
iclk	0.00us	0.00 %
▼ sc_mod	79.85 s	22.00 %
mythread	79.85 s	22.00 %
▼ sv_top	2.00 ms	0.00 %
NoName	2.00 ms	0.00 %
NoName	0.00us	0.00 %
► std	999.89us	0.00 %
► _global_	0.00us	0.00 %
total	282.53 s	77.85 %

Page: 1

In the `sv_mod` SystemVerilog module:

- The `begin-end` block `be1` consumes all of the CPU time of the module.
- There is an extraneous process call `NoName` that consumes no CPU time and can be ignored.
- The `iclk` input port in `sv_mod` is shown as a process, such as the `begin-end` block of the code. If `sv_mod` have other ports that are not clock signals, `profprt` does not show them as processes.

In the `sc_mod` SystemC module, `mythread()` is the SystemC variant of a named block in Verilog or SystemVerilog and represents the code (like in a SystemVerilog `always` procedure, but is shown in this view rather than in the Time Construct view). The implementation of this function is in the `.cpp` file.

Figure 44 The CPU Time Construct View

Time Construct View

Name	Time	Percentage
► Always	282.53 s	77.85 %
► Initial	2.00 ms	0.00 %
► Task	0.00us	0.00 %
► Function	0.00us	0.00 %
► Port	0.00us	0.00 %
total	282.53 s	77.85 %

Page: 1

The CPU time construct view shows the CPU times and percentages used by the `always` and `initial` procedures in the design and also the port in the design.

In this example view, `Task` and `Function` do not refer to a user-defined task and function, but rather refer to the internals of VCS and do not consume any CPU time. If this example contained user-defined tasks or functions, they are listed as a Task or Function here.

Figure 45 The Time Instance View

Time Instance View				
Instance	Inclusive Time	Percentage	Exclusive Time	Percentage
► sv_top	282.53 s	77.85 %	2.00 ms	0.00 %
► sv_top.sc_mod_inst	79.85 s	22.00 %	79.85 s	22.00 %
► std	999.89us	0.00 %	999.89us	0.00 %
► _global_	0.00us	0.00 %	0.00us	0.00 %
total	282.53 s	77.85 %	282.53 s	77.85 %

Page: 1

In this view, as it initially appears, you see the SystemVerilog top-level instance `sv_top`. You can also see the SystemC instance `sv_top.sc_mod_inst` because it is a SystemC instance in this SystemVerilog.

As in previous views, `std` and `_global_` are from the internals of VCS and can be ignored.

If you click the top-level module `sv_top`, you can see the `sv_mod_inst` instance.

Figure 46 The Expanded CPU Time Instance View

Time Instance View				
Instance	Inclusive Time	Percentage	Exclusive Time	Percentage
▼ sv_top	282.53 s	77.85 %	2.00 ms	0.00 %
► sv_mod_inst	282.53 s	77.85 %	282.53 s	77.85 %
► sv_top.sc_mod_inst	79.85 s	22.00 %	79.85 s	22.00 %
► std	999.89us	0.00 %	999.89us	0.00 %
► _global_	0.00us	0.00 %	0.00us	0.00 %
total	282.53 s	77.85 %	282.53 s	77.85 %

Page: 1

Figure 47 The PLI/DPI/DirectC View

Time PLI/DPI/DirectC View		
Name	Time	Percentage
PLI		
\$sc_mod_init	999.89us	0.00 %
\$vcdplusfilter	0.00us	0.00 %
\$msglog	0.00us	0.00 %
\$lsi_dumpports	0.00us	0.00 %
\$countdrivers	0.00us	0.00 %
\$vcsmemprof	0.00us	0.00 %
\$start_toggle_count	0.00us	0.00 %
\$report_toggle_count	0.00us	0.00 %
\$set_toggle_region	0.00us	0.00 %
\$toggle_start	0.00us	0.00 %
\$toggle_stop	0.00us	0.00 %
\$toggle_reset	0.00us	0.00 %
\$toggle_report	0.00us	0.00 %
\$read_lib_saif	0.00us	0.00 %
\$read_rtl_saif	0.00us	0.00 %
\$set_gate_level_monitoring	0.00us	0.00 %
DPI	0.00us	0.00 %
DirectC	0.00us	0.00 %
total	999.89us	0.00 %

This view for the PLI shows both VCS internal functions and user-written PLI functions.

In this example view, all functions are VCS internal functions. You can look for ones that consume the significant CPU time. The \$vcdplusmsglog system function, not in this example, can consume significant CPU time.

For SystemC, there is an additional CPU time view, the SC (SystemC) OverHead View.

Figure 48 The SC OverHead View

Time SC-OverHead View		
Name	Time	Percentage
SC-Value-OverHead	0.00us	0.00 %
SC-Kernel-OverHead	0.00us	0.00 %
SC-Spawn-OverHead	0.00us	0.00 %
total	0.00us	0.00 %

Page: 1

This data depends on the test case. It can be that kernel overhead becomes an issue and it can be compared against the Verilog kernel overhead.

The `sc-value` overhead is time taken to transfer data from one domain to another, such as to or from SystemC to or from Verilog, or SystemVerilog, or VHDL. This can be expensive when there is large amount of data, such as with a large vector signal or a large multidimensional array. Also spawning of processes can take time and accumulate sc-overhead.

Kernel overhead from SystemC, Verilog, SystemVerilog or VHDL, can become an issue when your code does not consume much CPU time and there is significant overhead to keep the co-simulation running. Usually you want these CPU time values to be low.

Reporting PLI, DPI, and DirectC Function Call Information

Profile information is reported for each PLI, DPI, or DirectC function called by your Verilog or SystemVerilog code. SystemC code is not included in these views.

This profiling capability has the following limitations:

- Details of VHDL code that calls external language functions are not reported.
- The location of the external language call is not reported.
- Text format reports are not supported; only the HTML format is supported.

Compiling and Running the Profiler Example

The following example illustrates the new runtime and memory usage reporting.

If your Verilog code contains user-defined system tasks for PLI functions like those shown in [Example 30](#) for a file named `pli.v`:

Example 30 Verilog System Tasks for PLI Functions

```
module top;
initial
begin
    $foo_200M();
    $foo_400M();
    $foo_200M();
    $foo_400M();
    $foo_200M();
    $foo_400M();
    $foo_200M();
    $foo_400M();
end
endmodule
```

Then, the `pli.tab` file looks like [Example 31](#).

Example 31 PLI Tab File

```
$foo_200M call=foo_200M_func
$foo_400M call=foo_400M_func
```

The C code in example file `pli.c` looks like [Example 32](#).

Example 32 C File

```
#include "stdio.h"
void foo_200M_func()
{
    char *a = NULL;
    int i = 0;
    for (i = 0; i < 100; i++)
        a = (void *) malloc (1024*2048);
}
void foo_400M_func()
{
    char *a = NULL;
    int i = 0;
    for (i = 0; i < 100; i++)
        a = (void *) malloc (2048*2048);
}
```

To compile these example files, use the following command line:

Two-step Flow:

```
% vcs -simprofile -P pli.tab pli.v pli.c
```

Three-step Flow:

```
% vlogan pli.v
% vcs top -simprofile -P pli.tab pli.c
```

Profiling Time Used by Various Parts of the Design

To profile the CPU time used by various parts of the design, use the `-simprofile` time option:

```
% simv -simprofile time
```

VCS creates:

- `simprofile_dir` database directory for the CPU time profile information.
- `profileReport.html` file and `profileReport` directory for the CPU time post-simulation profile report information.

To generate the Time PLI/DPI/DirectC view, use the `profprt` command with the following options:

```
% profprt -view time_pli simprofile_dir
```

To view the Time PLI/DPI/DirectC reports, open the `profileReport.html` file. In the left pane, select the `simprofile_dir` database and the Time PLI/DPI/DirectC view (see Figure 49).

This view shows the following:

- The `$foo_200M()` and `$foo_400M` PLI user-defined system tasks that call the `foo_200M_func()` and `foo_$00M_func()` C functions.
- CPU time that they use.
- Percentage of the total CPU time of the simulation.

Figure 49 Time PLI/DPI/DirectC View

The screenshot shows the 'Simprofile Report' interface. On the left, there are two dropdown menus: 'Database' set to 'simprofile_dir' and 'View' set to 'Time PLI/DPI/'. Below these is a 'GO' button. The main area is titled 'Time PLI/DPI/DirectC View' and contains a table with the following data:

Name	Time	Percentage
PLI		
\$foo_200M	2.97 ms	1.56 %
\$foo_200M /remote/vtqhome11/chandrab/SIMPROFILE/Simprofile_EPL/1.v4	2.97 ms	1.56 %
total	2.97 ms	1.56 %

Page: 1

Profiling Memory Used by Various Parts of the Design

To profile the machine memory used by various parts of the design, simulate using the `-simprofile mem` option:

```
% simv -simprofile mem
```

To generate the Memory PLI/DPI/DirectC view, use the `profrpt` command with the following options:

```
% profrpt -view mem_pli simprofile_dir
```

To view the Memory PLI/DPI/DirectC reports, open the `profileReport.html` file. In the left pane, select the `simprofile_dir` database and the Memory PLI/DPI/DirectC view (see [Figure 50](#)).

This view shows the following:

- The `$foo_200M()` and `$foo_400M` PLI user-defined system tasks that call the `foo_200M_func()` and `foo_$00M_func()` C functions.
- Machine memory that they used.
- Percentage of the total machine memory needed during the simulation.

Figure 50 Memory PLI/DPI/DirectC View

Memory PLI/DPI/DirectC View (clock:10)		
Name	Size	Percentage
PLI		
\$foo_400M	3600.00 MB	99.06 %
\$foo_400M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:5	2400.00 MB	66.04 %
\$foo_400M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:7	400.00 MB	11.01 %
\$foo_400M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:9	400.00 MB	11.01 %
\$foo_400M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:11	400.00 MB	11.01 %
\$foo_400M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:14	400.00 MB	11.01 %
\$foo_400M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:16	400.00 MB	11.01 %
\$foo_200M	1200.00 MB	33.02 %
\$foo_200M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:4	200.00 MB	5.50 %
\$foo_200M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:6	200.00 MB	5.50 %
\$foo_200M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:8	200.00 MB	5.50 %
\$foo_200M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:10	200.00 MB	5.50 %
\$foo_200M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:13	200.00 MB	5.50 %
\$foo_200M /remote/vtghome11/chandrab/SIMPROFILE/Simprofile_EPL1.v:15	200.00 MB	5.50 %
total	3600.00 MB	99.06 %

Page: 1

Here, the clock reference (clock:0) specifies the clock cycle of the peak memory usage (in this example, time 0).

Constraint Profiling Integrated in the Unified Profiler

Constraint profiling is integrated in the unified profiler. This integration adds the following views to the profile reports:

- the Time Constraint Solver view
- the Memory Constraint Solver view

These views tell you in detail the calls to the `randomize()` method that use the most CPU time or the most machine memory. With this information you can consider revising your constraints on the random variables to use less of these resources.

The Time Constraint Solver View

The following is an example of the Time Constraint Solver View.

Figure 51 Example Time Constraint Solver View

Time Constraint Solver View						
Total user time: 11.670seconds						
Total system time: 0.120seconds						
Total randomize time: 0.030seconds						
Total randomize count: 2						
Top randomize calls based on cpu runtime						
File:line@visit	serial#	time (sec)	variables	constraints	cnst blocks	
/env/nvs_atapi env.sv:118@1	1	0.030	27	37	9	
/env/nvs_atapi env.sv:120@1	2	0.000	3	3	1	
Top randomize calls based on cumulative cpu runtime						
File:line	calls	time (sec)				
/env/nvs_atapi env.sv:118	1	0.030				
/env/nvs_atapi env.sv:120	1	0.000				

Figure 52 Constraint Solver Profile

Constraint solver profile	
Solver	Time (sec)
Core Solver (default)	0.030
Core Solver (mode=1)	0.000
Core Solver (FAST)	0.000
Problem Generation	0.000

Top partitions based on BDD size					
File:line@visit	Rand.Partition	peak BDD size	final BDD size	variables	constraints
				cnst	blocks

Parts of this view in [Figure 51](#) are described in detail in [Figure 53](#), [Figure 54](#), [Figure 55](#), [Figure 56](#), [Figure 57](#), and [Figure 58](#).

Figure 53 Introductory information is at the top of the view

Time Constraint Solver View					
Total user time: 11.670seconds					
Total system time: 0.120seconds					
Total randomize time: 0.030seconds					
Total randomize count: 2					
Top randomize calls based on cpu runtime					
File:line@visit	serial#	time (sec)	variables	constraints	cnst blocks
/env/nvs_atapi_env.sv:118@1	1	0.030	27	37	9
/env/nvs_atapi_env.sv:120@1	2	0.000	3	3	1
Top randomize calls based on cumulative cpu runtime					
File:line	calls	time (sec)			
/env/nvs_atapi_env.sv:118	1	0.030			
/env/nvs_atapi_env.sv:120	1	0.000			

Total user time:

Specifies the total CPU time to simulate the design and testbench. In this example view, it is 11.670 seconds.

Total system time:

Specifies the total CPU time used by VCS when not simulating the design or testbench. In this example view, it is 0.12 seconds.

Total randomize time:

Specifies the CPU time that VCS needs to execute the `randomize()` method calls in the design. In this example view, it is 0.03 seconds.

Total randomize count:

Specifies the number of entries of the `randomize()` method in the SystemVerilog source code. In this example view, it is 2.

Figure 54 Top randomize calls based on CPU time

hypertext link Top randomize calls based on cpu runtime

File:line@visit	serial#	time (sec)	variables	constraints	cnst blocks
/env/nvs_atapi_env.sv:118@1	1	0.030	27	37	9
/env/nvs_atapi_env.sv:120@1	2	0.000	3	3	1

This section of the view is for the `randomize()` entries in the source code that use the most CPU time. There is a separate line for each entry. There are two such entries in this code example. So, they are listed here.

The columns in this section are for the following values:

File:line@visit

Specifies the following three things:

File

Specifies the path name for the source file that contains the entry. In this example view, the first line is for a source file with the `/env/nvs_atapi_env.sv` path name.

line

Specifies the line number in the source file that contains the entry. In this example view, the entry is on line 118.

@visit

An execution of the entry. There can be multiple executions of the same entry throughout a simulation. In this example view, the first line is for the first execution or visit of the entry.

If VCS executes the entry in the code three times, the line in this section could begin with:

`/env/nvs_atapi_env.sv:118@3`

Note:

The `File:line@visit` part of a line is in blue because this part is a hypertext link. When you click the link, the browser opens a new window showing the source file with the line specified at the top.

serial#

The series in this column is the order in which VCS executes the calls to the `randomize()` method. In this example view, line 118 contains the first call and line 120 contains the second call.

Note:

This section of the view is for the calls that used the most CPU time. However, these top users are not always the first or second `randomize()` calls that VCS executes.

time (sec)

The amount of CPU time used by the call.

variables

The number of `rand` or `randc` variables randomized by a call. Not all such variables in a class are randomized by a call.

constraints

The number of constraints in the class that are randomized by a call.

cnst blocks

The number of constraint blocks that contain these constraints.

Note:

In the following example, there is one constraint block and four constraints, as shown:

```
constraint reasonable_on_latencies {
    dior_to_data_place_time < 10;
    data_prepare_time       < 10;
    dior_to_data_place_time > 0;
    data_prepare_time       > 0;
} //end constraint reasonable_on_latencies
```

Figure 55 Top randomize calls based on cumulative CPU runtime

hypertext link

Top randomize calls based on cumulative cpu runtime		
File:line	calls	time (sec)
/env/nvs_atapi_env.sv:118	1	0.030
/env/nvs_atapi_env.sv:120	1	0.000

VCS can execute or visit a call to the `randomize()` method in a specific location of the source code more than once. If it does so, VCS keeps track of the cumulative CPU time used by these multiple executed calls and `profrpt` reports this cumulative time in this section.

This section reports:

- The location of the call in a hypertext link that opens a new window displaying the source code.
- The number of calls or visits to this location.
- The cumulative CPU time used by the calls.

Figure 56 Top partitions based on CPU time

Top partitions based on cpu time						
hypertext link	File:line@visit	Rand.Partition	cpu time	variables	constraints	cnst blocks
/env/nvs_atapi_env.sv:118@1		1.1	0.03	2	4	2
/env/nvs_atapi_env.sv:118@1		1.2	0.00	1	1	1
/env/nvs_atapi_env.sv:118@1		1.3	0.00	7	12	3
/env/nvs_atapi_env.sv:118@1		1.4	0.00	5	10	3
/env/nvs_atapi_env.sv:118@1		1.5	0.00	2	1	1
/env/nvs_atapi_env.sv:118@1		1.6	0.00	1	1	1
/env/nvs_atapi_env.sv:118@1		1.7	0.00	1	1	1
/env/nvs_atapi_env.sv:118@1		1.8	0.00	2	1	1
/env/nvs_atapi_env.sv:118@1		1.9	0.00	2	1	1
/env/nvs_atapi_env.sv:118@1		1.10	0.00	2	1	1

VCS has a constraint solver to determine the possible values that conform to your constraints. To solve these problems, the constraint solver divides its work into partitions. This section reports the number of partitions in a problem.

In this example view, this section reports the visit to the `randomize()` method in the example source file at `/env/nvs_atapi_env.sv` on line 118.

The constraint solver divided its work into 10 partitions. The `profrpt` utility generate reports for each partition:

- the CPU time needed to solve the partition
- the number of random variables in the partition
- The number of constraints

- The number of constraint blocks that contained these constraints

Figure 57 Constraint solver profile

Constraint solver profile	
Solver	Time (sec)
Core Solver (default)	0.030
Core Solver (mode=1)	0.000
Core Solver (FAST)	0.000
Problem Generation	0.000

Top partitions based on BDD size				
	peak	final		
File:line@visit Rand.Partition	BDD	BDD variables	constraints	cnst blocks

The total randomize time is further broken down into the different internal solvers and problem generation. This information might indicate where you can revise your constraints and randomize calls to improve the total CPU time.

Figure 58 Top partitions based on BDD size

Constraint solver profile	
Solver	Time (sec)
Core Solver (default)	0.030
Core Solver (mode=1)	0.000
Core Solver (FAST)	0.000
Problem Generation	0.000

Top partitions based on BDD size				
	peak	final		
File:line@visit Rand.Partition	BDD	BDD variables	constraints	cnst blocks

This part of the constraint profile report is empty unless VCS uses the solver (mode=1) in the randomization. When it uses mode=1, this section shows some memory footprint

information of different randomize calls executed under this solver (mode=1). You specify using the mode=1 solver with the `+ntb_solver_mode=1` runtime option and argument.

No information is in this example section because the default solver is doing the constraint solving for this example.

The Memory Constraint Solver View

The following is an example of the memory constraint solver view.

Figure 59 Example Memory Constraint Solver View

Memory Constraint Solver View						
Largest memory increment: 640KB						
Top randomize calls based on memory increment						
File:line@visit	serial#	mem incr (KB)	variables	constraints	cnst blocks	
/env/nvs_atapi_env.sv:118@1	1	640	27	37	9	
/env/nvs_atapi_env.sv:120@1	2	8	3	3	1	

Top recurring randomize calls based on memory increment		
File:line	calls	mem incr (KB)
/env/nvs_atapi_env.sv:118	1	640
/env/nvs_atapi_env.sv:120	1	8

Parts of this view, [Figure 59](#), are described in detail in [Figure 60](#) and [Figure 61](#).

In the beginning, the view shows the size of the largest memory increment during the simulation, as shown:

Figure 60 Largest memory increment

Memory Constraint Solver View						
Largest memory increment: 640KB						
Top randomize calls based on memory increment						
File:line@visit	serial#	mem incr (KB)	variables	constraints	cnst blocks	
/env/nvs_atapi_env.sv:118@1	1	640	27	37	9	
/env/nvs_atapi_env.sv:120@1	2	8	3	3	1	

Top recurring randomize calls based on memory increment		
File:line	calls	mem incr (KB)
/env/nvs_atapi_env.sv:118	1	640
/env/nvs_atapi_env.sv:120	1	8

In this example view, the largest increase in machine memory is an increase of 640 KB.

The view follows with the `randomize()` entries that cause the largest increase in the use of machine memory, as shown:

Figure 61 Top randomize calls based on memory increment

Top randomize calls based on memory increment						
hypertext link	File:line@visit	serial#	mem incr (KB)	variables	constraints	cnst blocks
/env/nvs_atapi_env.sv:118@1	1	640	27	37	9	
/env/nvs_atapi_env.sv:120@1	2	8	3	3	1	

The columns in this section are as follows:

File:line@visit

Specifies the following three things:

File

Specifies the path name for the source file that contains the entry. In this example view, the first line is for a source file with the path name `/env/nvs_atapi_env.sv`.

line

Specifies the line number in the source file that contains the entry. In this example view, the entry is on line 118.

@visit

A visit is an execution of the call. There can be multiple executions of the same call throughout the simulation. In this example view, the first line is for the first execution, or visit of the call.

If VCS executes the entry in the code three times, the line in this section could begin with:

`/env/nvs_atapi_env.sv:118@3`

Note:

The File:line@visit part of a line is in blue because this part is a hypertext link. When you click the link, the browser opens a new window showing the source file with the line specified at the top.

serial#

The series in this column is the order in which VCS executes the calls to the `randomize()` method. In this example view, line 118 contains the first call and line 120 contains the second call.

Note:

This section of the view is for the calls that use the most machine memory and these top users are not always the first or second `randomize()` calls that VCS executes.

mem incr (KB)

Specifies the amount of additional machine memory that VCS needs when it executes the call.

variables

The number of `rand` or `randc` variables randomized by a call. Not all such variables in a class are randomized by a call.

constraints

The number of constraints in the class that are randomized by a call.

cnst blocks

The number of constraint blocks that contain these constraints.

Figure 62 Top Recurring Randomize Calls

hypertext link

Top recurring randomize calls based on memory increment

File:line	calls	mem incr (KB)
/env/nvs atapi env.sv:118	1	640
/env/nvs atapi env.sv:120	1	8

The next section is for the `randomize()` calls that VCS executes the most. There are two `randomize()` entries in this example. Each entry is executed only once. The calls that are executed once are shown in this section because the code example does not contain calls that execute more frequently during the simulation.

This section reports:

- The path to the source file and the line number of the call.
- The number of times VCS executes a call.
- The amount of additional machine memory VCS needs to execute the call.

Performance/Memory Profiling for Coverage Covergroups

It provides the total time/memory taken by each covergroup across all its instantiations and the time/memory taken by individual instances of each covergroup. The data reported for a covergroup or a covergroup instance includes the time/memory spent in instantiating and initializing the covergroup instances and the time/memory spent in sampling the covergroup and the associated processing of the bins.

A covergroup instance is defined as the covergroup instantiation that is uniquely determined by an external reference as defined by the SystemVerilog LRM. This is also the lowest granularity at which time/memory data is reported. If a covergroup is instantiated multiple times on the same line of code, then the time/memory data is gathered for all those instances. Similarly, if a covergroup is instantiated within the same scope in different branches using the same handle, then the time/memory data is gathered for all those instances.

Default Summary View

When a default HTML Simprofile Report is loaded, the Default Summary View is opened.

The coverage component is split into two new components — Functional Coverage and Code Coverage. The Covergroup captures the total time/memory spent in all the instantiated covergroups for the run. The Code Coverage component captures the time/memory spent in segments of code coverage collection to be determined later. The full code coverage data is collected and reported.

Figure 63 Default Summary View

Time Summary View		
Component	Time	Percentage
VERILOG	13.57 s	97.17 %
Functional Coverage	8.68 s	62.14 %
Module	4.16 s	29.80 %
Function Coverage Kernel	730.69 ms	5.23 %
KERNEL	383.31 ms	2.74 %
HSIM	11.98 ms	0.09 %
total	13.97 s	100%

Time/Memory Summary View

To access the Time/Memory Summary View, click *Time/Memory Summary* option in the left pane of the Simprofile Report.

This view is similar to Default Summary View. To view more information, see [Default Summary View](#).

Time/Memory Module View

To access the Time/Memory Module view, click *Time/Memory Module* option in the left pane of the Simprofile Report.

Expanding a module/interface/program/package provides the data for the covergroups instantiated in it. The data for each covergroup captures the total time/memory spent in all instances of that covergroup across all the instances of the scope. As shown in above example, the `my_class1::my_cg` covergroup is instantiated thrice in the `my_mod` module; once as part of `mc1_top`, an object of class `my_class1`, and twice as part of `mc2_top`, an object of class `my_class2`. There are two instances of `my_mod` in the design. The data presented for `my_class1::my_cg` under `my_mod` is the cumulative data from all the six instances of the covergroup.

The covergroups are further expanded to provide data for each cover item (coverpoint or cross) in the covergroup.

Figure 64 Time/Memory Module View

Time Module View				
Module	Inclusive Time	Percentage	Exclusive Time	Percentage
< top	12.84 s	91.94 %	12.84 s	91.94 %
CoverGroup	8.68 s	62.14 %	8.68 s	62.14 %
cg	8.68 s	62.14 %	8.68 s	62.14 %
cpt1_cp	964.27 ms	6.90 %	964.27 ms	6.90 %
cpt4_cp	745.67 ms	5.34 %	745.67 ms	5.34 %
cpt3_cp	730.69 ms	5.23 %	730.69 ms	5.23 %
cpt2_cp	715.72 ms	5.12 %	715.72 ms	5.12 %
cg_cc_0	428.23 ms	3.07 %	428.23 ms	3.07 %
cg_cc	404.28 ms	2.89 %	404.28 ms	2.89 %
Initial	233.58 ms	1.67 %	233.58 ms	1.67 %
▶ NoName	233.58 ms	1.67 %	233.58 ms	1.67 %
<0.50 %	0.00us	0.00 %	0.00us	0.00 %
total	12.84 s	91.94 %	12.84 s	91.94 %

Page: 1

In Time/Memory Module view, click a covergroup to view the details of that covergroup in the *Construct Information* pane. These include the name of the covergroup, the scope in which it is declared (package, module, programs, interface, checker, or class), the total time/memory taken by all the covergroup instances in all the instances of the instantiating scope, and the file and line number for the declaration of the covergroup. Click the source file/line information to get the appropriate file and move the cursor to the appropriate line.

Time/Memory Construct View

To access the Time/Memory Construct View, click *Time/Memory Construct* option in the left pane of the Simprofile Report.

A new covergroup entry is added to the existing constructs. When a covergroup is expanded, it lists all the covergroups declared in the design. The data displayed for each covergroup is the cumulative data across all the instances of that covergroup regardless of where it is instantiated.

The covergroups are further expanded to provide data for each cover item (coverpoint or cross) in the covergroup.

Figure 65 Time/Memory Construct View

Time Construct View		
Name	Time	Percentage
▼ CoverGroup		
cg	8.68 s	62.14 %
▼ CoverPoint	3.16 s	22.59 %
cpt1_cp	964.27 ms	6.90 %
cpt4_cp	745.67 ms	5.34 %
cpt3_cp	730.69 ms	5.23 %
cpt2_cp	715.72 ms	5.12 %
▼ CoverCross	832.51 ms	5.96 %
cg_cc_0	428.23 ms	3.07 %
cg_cc	404.28 ms	2.89 %
▼ Initial	233.58 ms	1.67 %
NoName	233.58 ms	1.67 %
total	12.90 s	92.37 %

Page: 1

In Time/Construct View, click a covergroup to provide the details of the covergroup in the *Construct Information* pane. These include name of the covergroup, scope in which it is declared (package, module, programs, interface, checker, or class), total time/memory taken by all the covergroup instances of this covergroup in the entire design, and file and line number for the declaration of the covergroup. Click the source file/line information to get the appropriate file and move the cursor to the appropriate line.

Time/Memory Covergroup View

To access the Time/Memory Covergroup View, click the *Time/Memory Covergroup* option in the left pane of the Simprofile Report. It provides information for the functional covergroups and the time/memory information at the covergroup definition level. The time/memory data for the covergroup definition includes the time/memory spent in all the instances of that covergroup in the entire design.

Figure 66 Time/Memory Coverage View

Time Coverage View			Source Information
Name	Time	Percentage	
cg	8.69 s	62.24 %	/remote/vtghome11/chandrab/SIMPROFILE/Profiler_Example /Simprofile_Fcov_Support/Test/test.v:4-17
cpt1_cp(CoverPoint)	965.72 ms	6.91 %	/remote/vtghome11/chandrab/SIMPROFILE/Profiler_Example /Simprofile_Fcov_Support/Test/test.v:5-5
cpt4_cp(CoverPoint)	746.79 ms	5.35 %	/remote/vtghome11/chandrab/SIMPROFILE/Profiler_Example /Simprofile_Fcov_Support/Test/test.v:11-11
cpt3_cp(CoverPoint)	731.79 ms	5.24 %	/remote/vtghome11/chandrab/SIMPROFILE/Profiler_Example /Simprofile_Fcov_Support/Test/test.v:10-10
cpt2_cp(CoverPoint)	716.79 ms	5.13 %	/remote/vtghome11/chandrab/SIMPROFILE/Profiler_Example /Simprofile_Fcov_Support/Test/test.v:9-9
cg_cc_0(CoverCross)	428.88 ms	3.07 %	/remote/vtghome11/chandrab/SIMPROFILE/Profiler_Example /Simprofile_Fcov_Support/Test/test.v:16-16
cg_cc(CoverCross)	404.88 ms	2.90 %	/remote/vtghome11/chandrab/SIMPROFILE/Profiler_Example /Simprofile_Fcov_Support/Test/test.v:15-15
total	8.69 s	62.24 %	

Page: 1

Generating Simprofile Dynamic Report

Real Time Profile Monitoring (RPM) is a feature that supports generating memory profile report dynamically when the simulation starts. The RPM feature allows the profile report to be available even when the simulation terminates abruptly.

You can enable RPM using the following two interfaces:

1. Command Line Interface (CLI)

- Use the `-simprofile` runtime sub-option `rpm` to enable RPM. For example,
`simv -simprofile mem+rpm`
- Use the `-rpm` option of `profrpt` to access the alternative view. For example,
`profrpt -rpm <simprofile_dir> ...`

See the [Specifying Views](#) section for more details.

When the above commands are run, a dynamic page URL is displayed on the console, as highlighted in the following figure.

Figure 67 Displaying Dynamic Page URL on Console

```
/slowfs/[REDACTED]/pro [Thu 12:09pm] % profrpt -rpm ciena_MV_LP_AON/simprofile_dir/
Start RPM web server with process ID 9246
RPM web server activities will be logged in
    /slowfs/[REDACTED]/pro/simp_rpm.log
Listening on http://us01odcvde20728:1106
(Press CTRL+C to quit)
```

It is recommended to use Mozilla Firefox browser (version >= 38.0) to view all the functions.

You can invoke a web browser to open the dynamic page URL and view pages. The following page opens on the web browser:

Figure 68 Dynamic Page on Web Browser

Profile data	/slowfs/vgcs23/kavyaa/pro/ciena_MV_LP_AON/simprofile_dir
VCS Build Date	May 11 2022 20:36:04
Compile Version	T-2023.03-Alpha
Runtime Version	T-2023.03-Alpha
Host	us01odcvde20728
User	kavyaa

2. Using Graphical User Interface (GUI)

On the Firefox browser, you can view GUI where all pages show memory consumption of the specified entries. The following views are available on the GUI:

Home

The *Home* page displays the simulation profile and contains the following details:

- Simulation information

The following figure displays simulation information:

Figure 69 Simulation Information

Profile data	/SCRATCH/kavyaa/simprofile/ciena_MV_LP_AON/simprofile_dir
VCS Build Date	Aug 21 2023 21:01:40
Compile Version	V-2024.09-Alpha_Full64
Runtime Version	V-2024.09-Alpha_Full64
Host	odcphy-vg-1217
User	kavyaa
Simulation Start at	Tue Aug 22 21:54:23 2023
Simulation Timescale	1 ps
Simulation Command	./simv.V-2024_09-Alpha_Full64 -simprofile mem+rom -power loconfig z.tcl

- Memory profile information

The following figure displays memory profile information:

Figure 70 Memory Profile Information

The screenshot shows the 'Memory Profile Information' page of the Simprofile tool. The left sidebar has links for Home, Memory Summary, Verilog, Dynamic Memory, Memory Construct, and Memory Search. The main content area is titled 'Memory Profile Information' and contains a table with the following data:

Latest Snapshot Time	2023-08-22 21:54:27
Number of Dynamic Memory Snapshots	0
Number of Heap Memory Snapshots	28
Number of Virtual Memory Snapshots	55
Total Virtual Memory	546 MB
Dynamic Memory Pool	0 MB
Simulation Time	0
Virtual Memory Operation Count	550067

Click any hyperlink available above the Memory Profile Information table to open the respective page.

Note:

Simulation with less dynamic memory operations dumps a small number of snapshots. For example, if dynamic MOC (Memory Operation Count) is less than 10,000, only one snapshot is considered which is not sufficient for generating graphs. Therefore, dynamic memory report of RPM is most suitable for simulation with a large number of SystemVerilog activities.



Click the *Settings* icon,  , to open the *Settings* page, as shown in the following figure:

Figure 71 Settings

The screenshot shows the 'Settings' page. It has a header 'Settings' and a close button 'x'. Below the header are two radio buttons: 'X-axis' (selected) with options 'Simulation time (simtime)' and 'Memory operation count (moc)'. There are two sections for 'No. of result per query': 'on general large-size data' (radio buttons 5 and 10, with 5 selected) and 'on search & construct data' (radio buttons 10 and 20, with 10 selected). At the bottom is a toggle switch for 'Auto refresh'.

On the *Settings* page, you can configure the following fields:

- *X-axis*: You can choose the parameter you want to display on x-axis. By default, it is *Simulation time*.
- *No. of result per query on general large-size data*: You can choose the number of entries or traces per query. By default, it is set to 5.
- *No. of result per query on search & construct data*: You can choose the number of entries to be displayed. By default, it is set to 10.
- *Auto refresh*: You can enable or disable auto refresh feature by clicking the *Auto refresh* button.

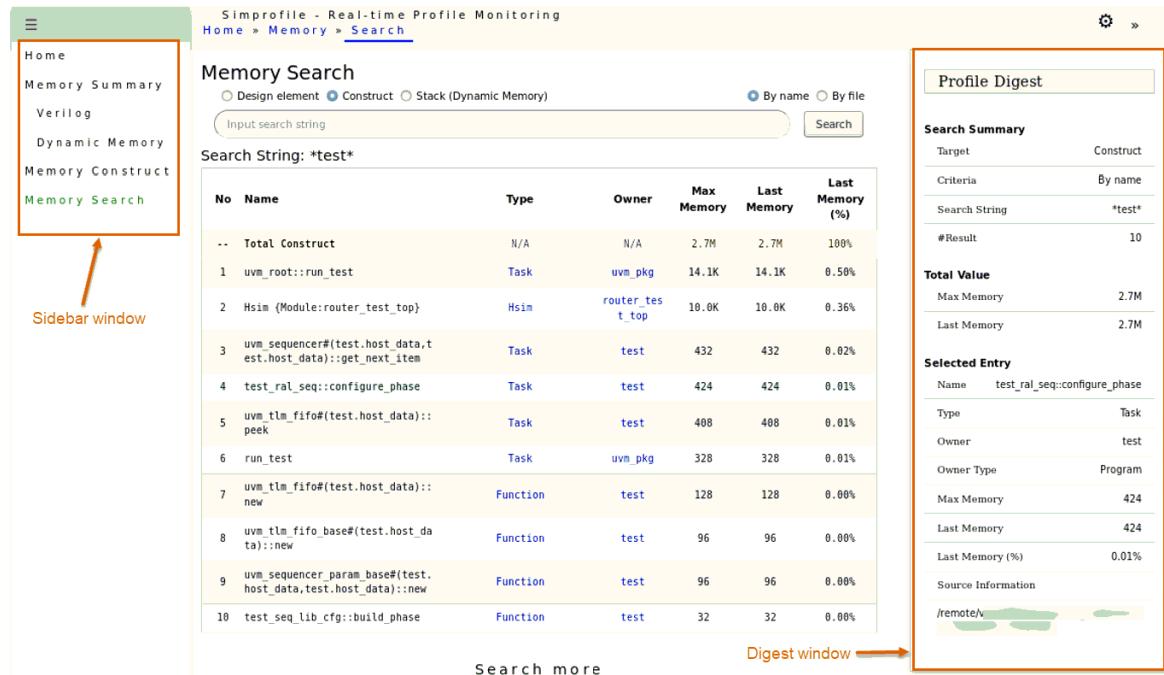
Memory Summary

It displays entries under total virtual memory which consists of the following two types of memories:

- Verilog
- Dynamic Memory

The Memory Summary consists of the following windows:

Figure 72 Memory Summary Windows



- Digest Window

When you hover over or click a row displayed on the Memory Summary page, the related profile information, such as the filename and the line number of the corresponding entry, is displayed in the Digest window available on the right-hand side of the page.

You can show and hide the Digest window using the “**«**” and “**»**” icons respectively. These icons are available at the upper-right corner of the page.

- Sidebar Window

The Sidebar window provides links to the pages and their sub pages. It displays the following:

- Top-level pages: The top-level pages consists of Memory Summary, Memory Construct, and Memory Search.
- Sub pages: The sub pages such as Verilog and Dynamic Memory are available under the top-level page, Memory Summary.

Note:

An entry displaying a corresponding line chart indicates that the entry has values recorded for the time series. VCS supports time-series values only on Memory Summary View, Memory Construct, and some selected entries, such as design elements.

The following figure displays a graph with simulation time (*simtime* →) on x-axis and memory consumption (\uparrow mem) on y-axis for each entry:

Figure 73 *Memory Summary View*

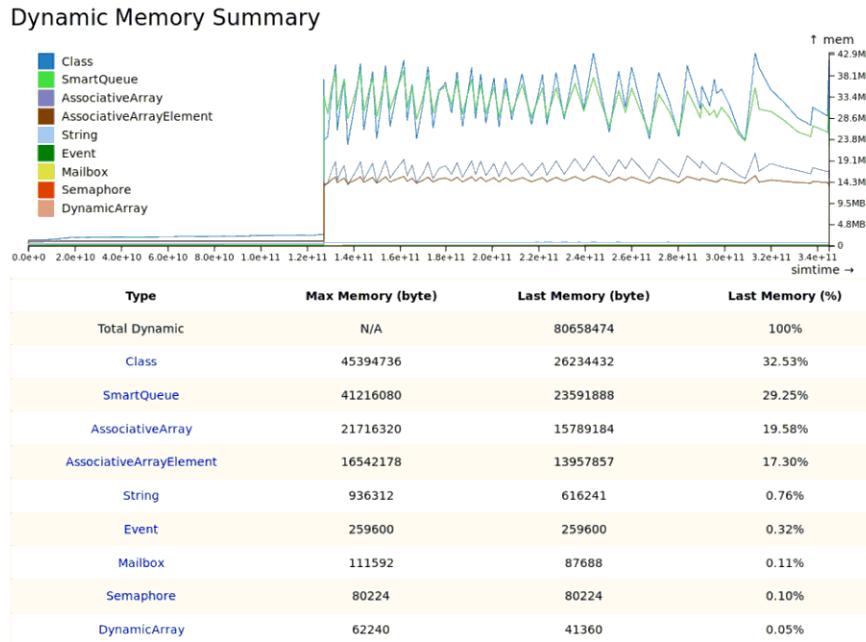


Dynamic Memory

In the Memory Summary view, *Dynamic Memory Pool* is a hyperlink and you can click it for more details.

When you click *Dynamic Memory Pool*, the following Dynamic Memory Summary view opens:

Figure 74 Dynamic Memory Summary View

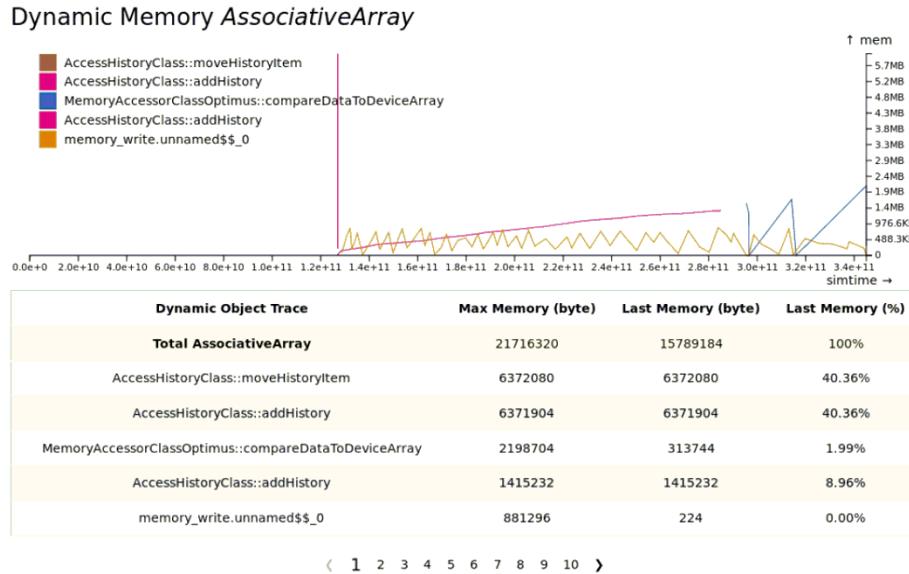


Dynamic Memory Summary displays the memory consumption of the categorized predefined groups, for example, *Class*, *SmartQueue*, *AssociativeArray*, and so on. You can click the hyperlink of the listed predefined groups for more details.

When you click a predefined group, it opens the specific dynamic memory view of the predefined group.

The following view displays entries under the *AssociativeArray* group:

Figure 75 Dynamic Memory View on AssociativeArray



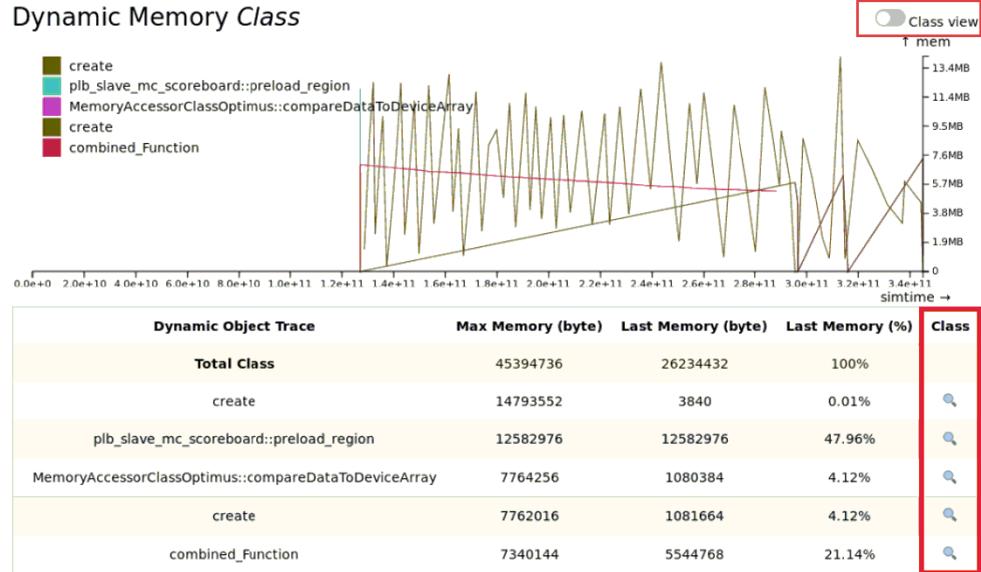
The *Dynamic Memory* view for a predefined group consists of a large number of entries and only a limited number of entries are displayed on the page. You can view rest of the entries by clicking the numbers, < 1 2 3 4 5 6 7 8 9 10 >, available below the table. These numbers show the pagination, and the current page is highlighted with a larger font size.

The entries are sorted based on the memory consumed. The entry that consumes the maximum memory among all snapshots is placed at the topmost position.

When you click the *Class* hyperlink available in the *Dynamic Memory Summary* view, an additional column *Class* is added to link to the class specific view. You can enable or disable Class view using *Class view* button.

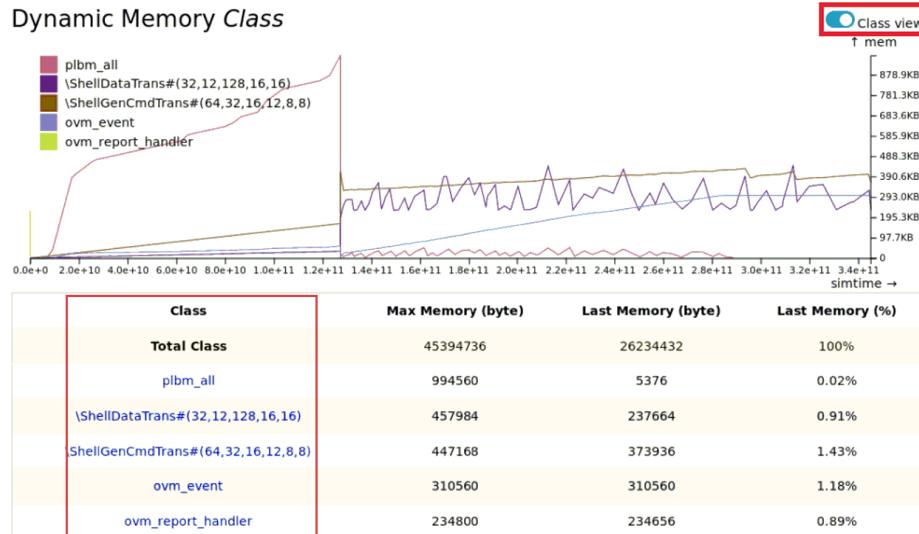
The following figure displays Dynamic Memory view with Class view disabled:

Figure 76 Dynamic Memory View on Class (Class view disabled)



When you enable Class view, the table lists the aggregated entries under Class type of the dynamic memory. The following figure displays the dynamic memory view with Class view enabled:

Figure 77 Dynamic Memory View on Class (Class view enabled)

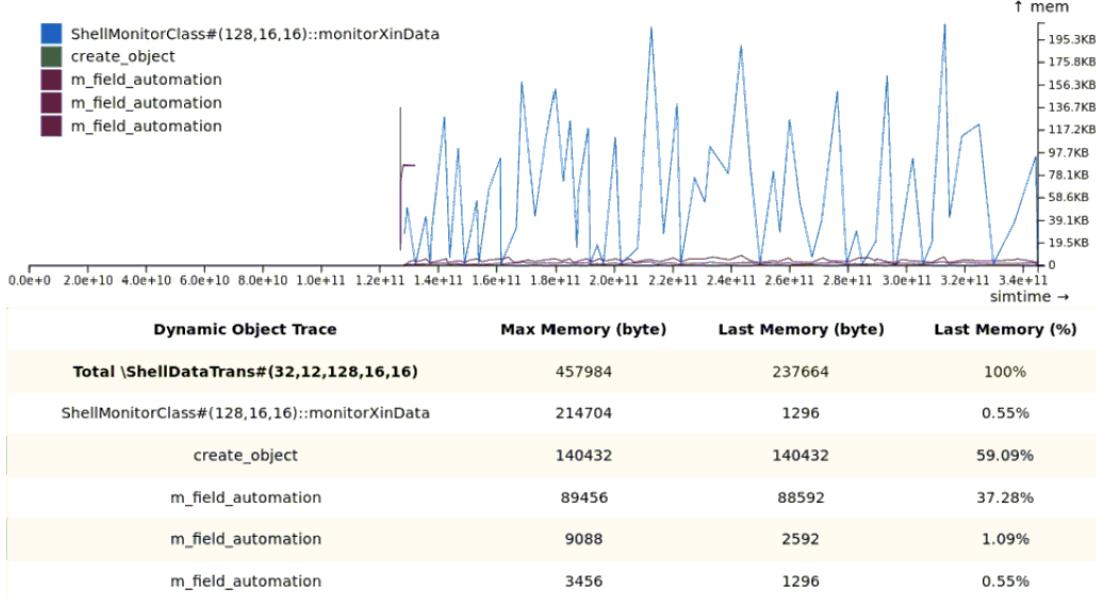


To learn more about each entry listed in the *Class* column, click the specific entry. It displays the underlying object of the specific class type.

For example, when you click `ShellDataTrans#(32, 12, 128, 16, 16)`, it displays information shown in the following figure:

Figure 78 Dynamic Memory View on `ShellDataTrans#(32,12,128,16,16)`

↳ Dynamic Memory `\$ShellDataTrans#(32,12,128,16,16)`



Verilog

In the Memory Summary view, you can find non-dynamic entries like VERILOG, Kernel, Value Change Dumping, and Profiling which can be expanded to show sub-entries. The following figure displays the sub-entries of VERILOG:

Figure 79 Memory Summary View with VERILOG Expanded

Total Memory	379.0M	379.0M	100.00%
▶ Library/Executable	261.0M	261.0M	68.87%
▶ Value Change Dumping	42.0M	42.0M	11.07%
▼ SNPS Memory Pool	35.6M	33.3M	8.79%
Dynamic Memory Pool	5.0M	5.0M	1.32%
▶ KERNEL	10.3M	10.2M	2.68%
HSIM	4.3M	2.3M	0.60%
▼ VERILOG	3.6M	3.6M	0.95%
Package	3.1M	3.1M	0.81%
Program	254.5K	254.5K	0.07%
Module	251.7K	198.3K	0.05%
Interface	81.2K	81.2K	0.02%
Function Coverage Kernel	600	600	0.00%
CONSTRAINT	288.8K	288.8K	0.07%
PLI/DPI/DirectC	231.1K	184.1K	0.05%
▶ Profiling	4.0K	4.0K	0.00%
Anonymous	N/A	26.2M	6.92%

In the *Memory Summary* view, entries displayed in blue text are hyperlink. When you click such entry, you are redirected to a page with a detailed information about that entry.

For example, when you click VERILOG, you are redirected to a page displaying memories from Verilog, grouped under design elements (top-level elements) of the Verilog language, like `uvm_pkg`, `test`, and `router_test_top` as shown in the following figure:

Figure 80 Memory Verilog View with uvm_pkg Expanded

Name	Type	Max Memory	Last Memory	Last Memory (%)
Total Verilog	N/A	4.2M	4.2M	100%
▼ uvm_pkg	Package	2.5M	2.5M	60.29%
Function		1.4M	1.4M	34.29%
Task		839.5K	839.5K	19.69%
CombineConstruct		11.0K	11.0K	0.26%
Hsim		48	48	0.00%
▶ test	Program	367.9K	367.9K	8.63%
▶ router_test_top	Module	131.0K	131.0K	3.07%
std	Package	94.8K	94.8K	2.22%
▶ router_io	Interface	69.2K	69.2K	1.62%

Each design element when expanded provides a list of construct types. You can click the construct type, a hyperlink, to go to the page specific to the clicked construct type of the selected design element.

For example, VCS takes you to the following page when you click the *Function* construct type under the *uvm_pkg* design element:

Figure 81 Memory *uvm_pkg* (*Function*) View

Name	Max Memory	Last Memory	Last Memory (%)
Total Function	1.4M	1.4M	100%
uvm_task_phase::execute	628.5K	620.0K	42.41%
exists	459.9K	459.9K	31.46%
uvm_phase::drop_objection	219.8K	219.8K	15.03%
uvm_resource_base::set_scope	33.5K	33.5K	2.29%
get	25.0K	25.0K	1.71%
verdi_set_verbosity	24.1K	24.1K	1.65%
uvm_sequencer_base::start_phase_sequence	18.6K	18.6K	1.27%
uvm_reg_field::configure	18.2K	18.2K	1.25%
uvm_event::trigger	15.8K	15.8K	1.08%
uvm_report_server::f_display	8.0K	8.0K	0.55%

Memory Construct Summary View

The *Memory Construct Summary* view provides summary of the construct types as shown in the following figure:

Figure 82 Memory Construct Summary View

Name	Max Memory	Last Memory	Last Memory (%)
Total Construct	2.7M	2.7M	100%
Function	1.4M	1.4M	52.69%
Task	1.1M	1.1M	42.16%
PortConnections	71.7K	71.7K	2.58%
CombineConstruct	32.8K	32.8K	1.18%
Hsim	19.4K	19.4K	0.70%

You can click each construct type to display the memory details on another page. For example, the following figure displays the *Memory Task* page that opens when you click the *Task* construct type:

Figure 83 Memory Task View

↳ Memory Task

Name	Owner	Max Memory	Last Memory	Last Memory (%)
Total Task		1.1M	1.1M	100%
uvm_phase::execute_phase	uvm_pkg	451.7K	451.7K	38.49%
iMonitor::get_packet	test	120.3K	120.3K	10.25%
uvm_objection::m_execute_scheduled_forks	uvm_pkg	105.4K	105.4K	8.98%
reset_sequence::reset_dut	test	95.4K	95.4K	8.13%
uvm_sequence_base::start	uvm_pkg	51.5K	51.5K	4.39%
host_driver::post_reset_phase	test	26.7K	26.7K	2.27%
execute	uvm_pkg	25.6K	25.6K	2.18%
uvm_run_phase::exec_task	uvm_pkg	23.0K	23.0K	1.96%
body	uvm_pkg	17.4K	17.4K	1.48%
driver::send_address	test	16.6K	16.6K	1.42%

Memory Search View

The *Memory Search* view displays entries matched with the string entered in the search text box for any of the following targets:

- Design element
- Construct
- Stack (Dynamic memory)

Select one of the following two search criteria:

- By name: Searches by the text entered in the Search box.
- By file: Searches the file where the entered search text is defined.

Note:

The Memory Search view does not support the Auto refresh feature available in the Settings page.

Figure 84 *Memory Search Target and Criteria*



You can use the wildcard character “*” to perform a wildcard search. Wildcard can be used along with other characters to form a flexible pattern which are valid in Verilog. For example, when you type *_my_test in the Search box, it displays results such as foo_my_test, abc_my_test, and so on.

By default, ten matched results are displayed, and you can click *Search more* to display more matched entries.

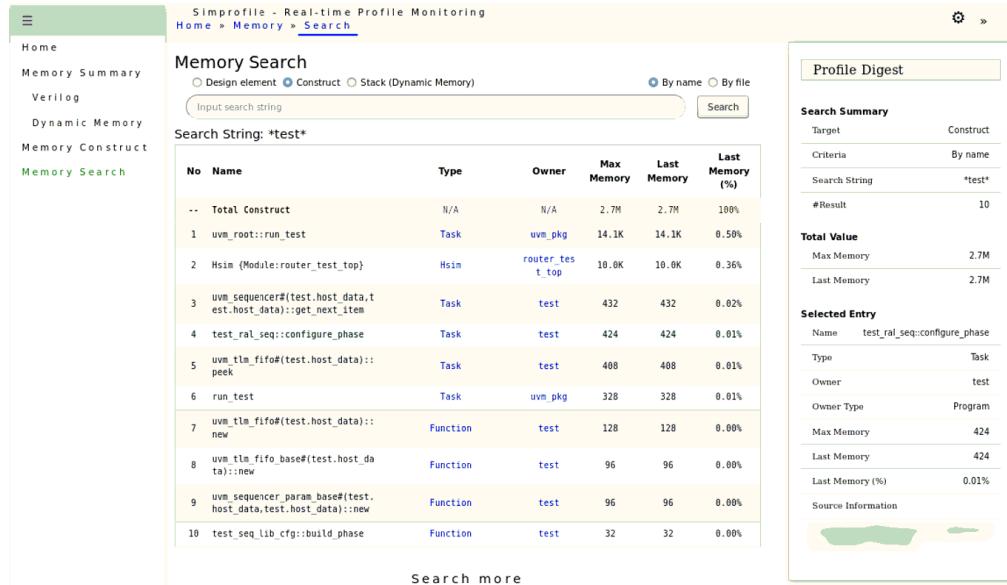
For example, when you click the *Memory Search* link available in the Sidebar window, the *Memory Search* page opens. Type "*test*" as the search string in the Search text box, and click *Search*. A list of ten matched results are displayed on the Memory Search page.

When you select any matched result (highlighted in the following figure), the Profile Digest displays the following details:

- Search Summary
- Total Value

- Selected Entry

Figure 85 Memory Search Matched Results



Advanced Simprofile Usages

This section describes the following features of Simprofile:

- [The Accumulative Views](#)
- [The Comparative View](#)
- [Reporting Debug Capabilities for Each Module](#)
- [Simulation Time Slice Based Profiler](#)

The Accumulative Views

The accumulative views displaying the accumulated CPU time or the machine memory profile information from two or more databases have been expanded to more accumulative views. These views are:

- the accumulative summary view
- the accumulative module view
- the accumulative instance view
- the accumulative construct view

To generate a report showing the accumulated results from more than one profile database, follow these steps:

1. Write a file that lists the profile databases.
2. Run the `profrpt` profile report generator without entering a profile database on the command line. Instead, enter the `-f` option specifying the file that lists the databases.

```
% profrpt -view time_all -f time_db_list \
-output accum_time
```

In this example, the `-f` option specifies the file `time_db_list`, which contains a list of databases:

```
simprofile_dir
simprofile_dir.1
simprofile_dir.2
```

The `profrpt` profile report writing utility writes `accum_time.html`.

The `profrpt` utility only writes this accumulative view in HTML format. The text file format is not supported.

Figure 86 The Right Pane of the Accumulated CPU Time View in accum_time.html

Time Summary		
Component	Time	Percentage
KERNEL	779.6 ms	51.97%
VERILOG	700.0 ms	46.67%
HSIM	19.8 ms	1.32%
PROFILE	622.0 us	0.04%
total	1.5 s	100%

This view shows the accumulated CPU times.

Figure 87 The Left Pane of the Accumulated CPU Time Views in *accum_time.html*



The *Database* drop-down menu contains:

- selections for each of the accumulated databases
- selections for the module, instance and construct points of view.

Figure 88 The Database Drop-Down Menu in the Left Pane



The following selections display the accumulated information from different points of view:

module summary

Contains a section for each module definition.

instance summary

Contains a section for each module instance.

construct summary

Contains a section for each type of construct, such as initial and always blocks.

Also from the drop-down menu, you can select the accumulated simulation time profile database. If you select one of these databases, the view changes to the time summary view for that database.

This example shows how to generate accumulative views of the CPU time profile data. The same can be done for machine memory profile data.

Example 33 Code Example for the Accumulative View

```
// test.v
module dut (input reg in[0:3], output reg out);
wire c1, c2;
assign c1 = in[0] & in[1];
assign c2 = in[2] & in[3];
or o1 (out, c1, c2);
endmodule

//tb.v
module tb1;
reg in[0:3], out;

dut d1 (in, out);
initial begin
in[0]=1; in[1]=0; in[2]=1; in[3]=0;
end
always #5 in[0] = ~in[0];
always #6 in[1] = ~in[1];
always #7 in[2] = ~in[2];
always #8 in[3] = ~in[3];
initial begin
$monitor($time,"in[0]=%b, in[1]=%b, in[2]=%b, in[3]=%b, out=%b\n", in[0],
in[1], in[2], in[3], out);
end
endmodule
```

Also for this example you have the following -i UCLI command files:

```
// run1
run 10000
quit

// run2
run 100000
quit
```

To run this example, enter the following command lines:

```
% vlogan -nc -sverilog test.v tb1.v
```

```
% vcs tb1 -sverilog -nc -simprofile
```

```
% simv -simprofile time -ucli -i run1
```

```
% simv -simprofile time -ucli -i run2
```

```
% profrpt -view time_all -f file
```

VCS writes the first profile database

VCS writes the second profile database

The `time_all` argument specifies, among other views, the accumulative time view.

Open the `profileReport.html` file to see the accumulative time view

Figure 89 The Accumulative Summary View

Simprofile Report			
Database:	accumulative	Time Summary	
	<input type="button" value="GO"/>	Component	Time Percentage
KERNEL		1.5 s	94.57%
VERILOG		53.4 ms	3.34%
HSIM		17.7 ms	1.11%
DEBUG		13.9 ms	0.87%
Value Change Dumping		1.8 ms	0.11%
		total 16 s	100%

Figure 90 The Accumulative Module View

Simprofile Report			
Database:	accumulative	Module Time Summary	
	<input type="button" value="GO"/>	Name Time Percentage	
		tb1 47.2 ms 88.4%	
		dut 6.2 ms 11.6%	
		total 53.4 ms	
Page: 1			

Figure 91 The Accumulative Instance View

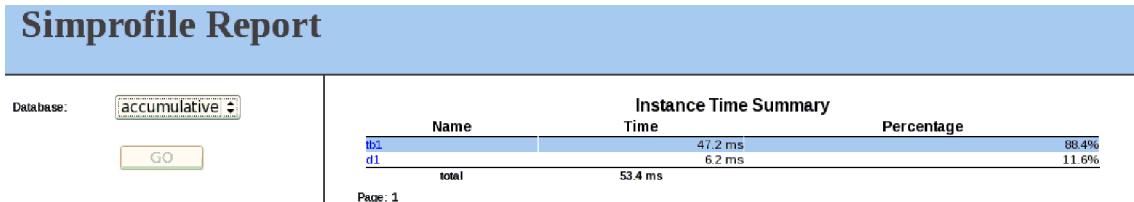
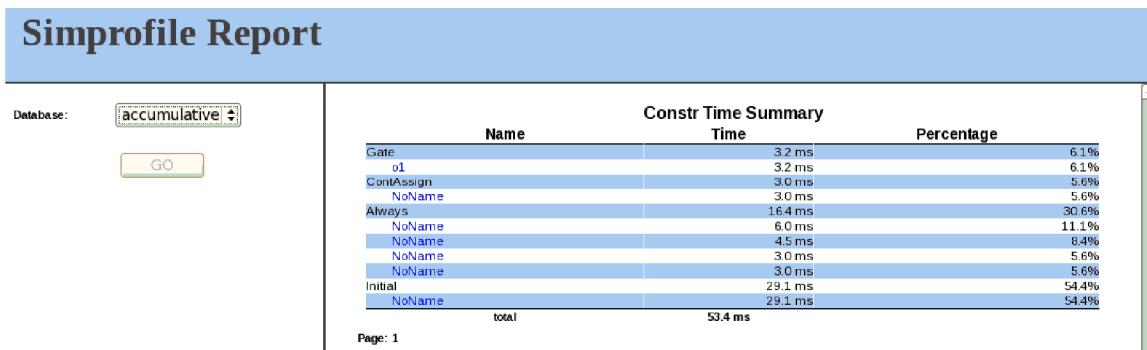


Figure 92 The Accumulative Construct View



The accumulative views are only available in HTML format.

The Comparative View

To generate a report that compares the results of two profile databases from two different simulations, include the `-diff` option and enter both databases on the `profrpt` command line.

Of the two specified databases, the first is the target database and the second is the reference database.

For example:

```
% profrpt -view ALL -diff simprofile_dir simprofile_dir.1 -output diff
```

The `profrpt` profile report writing utility in this example writes the `diff.html` file to compare the two profile databases.

Figure 93 The Comparative View in diff.html

Time Summary			
Component	Time	Reference	Gap
VER LOG	52.6 s	81.6 s	17.3 ms
PL/DPI/C reIC	32.2 s	36.2 s	-30 s
KERNEL	572.4 ms	101.2 ms	731.2 ms
HSIM	39.9 ms	23.9 ms	15.9 ms
Value Change Dumping	16.0 ms	0 ms	16.0 ms
total	134.4 s	127.1 s	-7.3 s

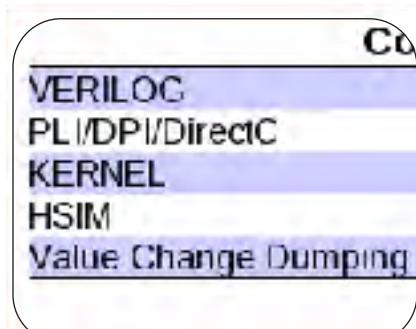
Memory Summary			
Component	Memory	Reference	Gap
PL/DPI/C reIC	512.3 MB	511.5 MB	0 B
KERNEL	35.8 MB	35.6 MB	0 B
Value Change Dumping	8.3 MB	6.3 MB	0 B
VLLOC	5.8 MB	5.6 MB	0 B
HSIM	2.1 MB	2.1 MB	0 B
total	601.1 MB	561.1 MB	0 B

Dynamic Summary			
Component	Dynamic	Reference	Gap
String	4.1 MB	4.2 MB	0 B
Class	60.0 KB	63.0 KB	0 B
SmartQueue	1.1 KB	0.0 KB	0 B
total	4.7 MB	4.7 MB	0 B

In this example view, the target database is the newer `simprofile_dir` directory and the reference database is the older `simprofile_dir.1` directory.

The significant differences are in the CPU times. Figure 94 shows magnifications of the CPU time as part of the comparative view.

Figure 94 Magnifications of the Values in the Comparative View



Time	Reference	Gap
81.6 s	81.6 s	17.9 ms
32.2 s	35.2 s	-3.0 s
574.4 ms	335.2 ms	239.2 ms
39.9 ms	23.9 ms	15.9 ms
16.0 ms	4.0 ms	12.0 ms
114.4 s	117.1 s	-2.7 s

As you can see in Figure 94, the reference database needed a full 3.0 seconds more of CPU time to execute its PLI application, but in the other components the target database needed more CPU time. Green negative gap values indicate that the reference database values exceed the target database. Red values indicate that the target database exceeds the reference database.

Limitations

- The comparative view only compares the information in the summary views.
- The comparative view is only supported in HTML format.

Reporting Debug Capabilities for Each Module

VCS integrates the profiler report with debug capabilities profile that shows the debug capacities enabled and used by each module. The debug capabilities that you enable are collected at the compile time. At runtime, the capabilities actually used by modules are recorded. Simprofile automatically analyzes the database generated on runtime as well as the compile-time data and shows the profile in the final report.

Use Model

The following is the use model for reporting debug capabilities:

```
%vcs -simprofile  
%vcs -simprofile <time/mem>
```

To generate debug aware profile report, invoke `profrpt` using one of the following commands:

```
%profrpt -view plilearn <other profrpt options>
```

or

```
%profrpt -view ALL <other profrpt options>
```

Separate file `PliLearn.txt` is generated for PLI debug view. The switch at `profrpt` is `profrpt -view plilearn plilearn+mem_mod`, which considers both time and memory to generate report.

HTML Reports

The debug capability profile presents a separate view. This is called as the ACC capacity view in the final HTML report. This view consists of two sets of data:

- Capacities Statistics (PLI Debug Capability View)

For each capability, the following three sets of statistics data are generated:

- Enabled modules
- Enabled and used modules
- Enabled but not used modules

The HTML reports are shown in [Figure 95](#):

Figure 95 PLI Debug Capability View in HTML Report

Click this blue colored hyperlink to display the time taken by the module below HTML report.

Stat.	read	read_write	callback	callback_all	force	static
enabled module no.	9	6	3	2	3	7
enabled	100.00 %	56.57 %	88.83 %	22.22 %	23.33 %	77.78 %
enabled module time	7.17 s	20.92 ms	7.15 s	0.00 s	16.23 ms	7.15 s
percentage	30.82 %	2.26 %	89.08 %	0.02 %	0.24 %	89.39 %
used module no.	3	3	2	0	0	0
percentage	0.00 %	0.00 %	22.22 %	0.02 %	33.33 %	0.00 %
used module time	3.30 ms	3.30 ms	0.00 ms	0.00 ms	18.53 ms	3.30 ms
percentage	0.00 %	0.00 %	0.02 %	0.02 %	0.24 %	0.00 %
disabled module no.	3	6	5	2	0	7
percentage	100.00 %	36.51 %	65.67 %	22.22 %	0.00 %	17.78 %
Unused module time	7.17 s	20.92 ms	7.15 s	0.00 s	0.00 s	7.15 s
Percentage	100.00 %	11.56 %	29.01 %	0.00 %	0.00 %	88.03 %
Capabilities enabled but no module uses: read, read_write, callback_all, static						
Capabilities enabled and used by all modules: force						
show full module list						

Percentage = enabled module number / Total number of modules enabled. For example, 9/9 for read, 6/9 for read_write, and so on.

Click this hyperlink to display the full module view with each capability enabled.

Each data set includes module number, the percentage in total module number, modules exclusive time, and the percentage in total execution time.

As shown in the figure, percentage is calculated as follows:

$$\text{Percentage} = \frac{\text{enabled module number}}{\text{Total number of modules enabled}}$$

To display the full module view with each capability enabled, click the module number, as shown in [Figure 96](#):

VCS also provides the following aggregated statistics:

- Capabilities enabled but no module uses. This means that the capabilities are enabled by some modules but are not used in any of the modules.
- Capabilities enabled and used by all modules. This means that the capabilities are enabled by some modules and are used in all these modules.
- Module level debug capacities (Full Module View)

The reports indicate which module needs which capacity. Capacities that are used by all modules or by no modules are extracted for clarity.

The slot labels are:

- *Enabled*: Enabled but not used
- *Used*: Enabled and used
- *Empty slot*: Not enabled

Figure 96 Full Module List View in HTML Report

Full Module View							
Module	Exclusive Time	read	read_write	callback	callback_all	force	static
TbTop4	7.29 s	Enabled		Enabled			Enabled
DutTop2	19.27 ms	Enabled	Enabled				Used
Top	9.63 ms	Enabled	Enabled	Enabled			Enabled
TbTop	0.00us	Enabled		Enabled			Enabled
mid1	0.00us	Enabled	Enabled	Used	Enabled		
bot1	0.00us	Enabled	Enabled	Used	Enabled		Enabled
TbTop1	0.00us	Enabled	Enabled	Enabled		Used	Enabled
TbTop2	0.00us	Enabled		Enabled			Enabled
TbTop3	0.00us	Enabled	Enabled	Enabled		Used	Enabled

Text Reports

There are large number of modules in a design. Therefore, the module level data do not fit into the size of the text report. So, the text report includes only the capacities statistics data as shown in [Figure 97](#).

Figure 97 Text Report

PLI Debug Capability View						
Stat.	read	read_write	callback	callback_all	force	static
enabled module no.	9	6	8	2	3	7
enabled module percentage	100.00 %	66.67 %	88.89 %	22.22 %	33.33 %	77.78 %
enabled module time	7.32 s	28.90 ms	7.30 s	0.00 us	19.27 ms	7.30 s
enabled module percentage	89.62 %	0.35 %	89.39 %	0.00 %	0.24 %	89.39 %
used module no.	0	0	2	0	3	0
used module percentage	0.00 %	0.00 %	22.22 %	0.00 %	33.33 %	0.00 %
used module time	0.00 us	0.00 us	0.00 us	0.00 us	19.27 ms	0.00 us
used module percentage	0.00 %	0.00 %	0.00 %	0.00 %	0.24 %	0.00 %
unused module no.	9	6	6	2	0	7
unused module percentage	100.00 %	66.67 %	66.67 %	22.22 %	0.00 %	77.78 %
unused module time	7.32 s	28.90 ms	7.30 s	0.00 us	0.00 us	7.30 s
unused module percentage	89.62 %	0.35 %	89.39 %	0.00 %	0.00 %	89.39 %

Limitations

The following are the limitations with this feature:

- Module level data is available only for text reports.
- No detail data, such as source code is available in module list view in HTML or text reports.
- Only support capabilities, such as `read`, `read_write`, `callback`, `callback_all`, `force`, `static` are reported. Other debug capabilities like `line callback` are not reported.
- Only the original module is shown for parameterized modules.

Simulation Time Slice Based Profiler

VCS allows you to generate profile report for a specific time period and helps you to limit the size of the database.

For example, you can generate a profile report for the time period when the simulation is very slow (before reset) or you can generate a profile report for the time period when the simulation is occupying huge memory.

Use Model

The following is the use model for reporting debug capabilities:

```
%vcs -simprofile
%simv -simprofile -simprofile_start <t+ht> -simprofile_stop <t+ht>
```

where,

```
-simprofile_start <t+ht>
```

Turns on the simulation profile dumping at simulation time t . ht is the high 32 bits. If ht is 0, then it can be omitted.

```
simprofile_stop <t+ht>
```

Turns off the simulation profile dumping at simulation time t . ht is the high 32 bits. If ht is 0, then it can be omitted.

Example 1

```
% simv -simprofile time -simprofile_start 1+50 -simprofile_stop 1+60
```

Here,

```
start time is è 1 * 2 ^32 + 50 = 4294967346ns
stop time is è 1 * 2 ^32 + 60 = 4294967356ns
```

Example 2

```
% simv -simprofile time -simprofile_start 50 -simprofile_stop 60
```

Here,

start time is è 50 = 50ns
stop time will be è 60 = 60ns

To generate time slice profiler report, invoke profrpt as follows:

```
% profrpt -start <start-time> -stop<stop-time> <other profrpt options>
```

where,

-start <time>

Specifies the starting time (in simulation units) when the report generation should begin.
By default, the start time is 0.

-stop <time>

Specifies the stopping time (in simulation units) when the report generation should end.
By default, the report generation stops at the latest time available in the simulation profile database.

Figure 98 HTML Report - Time/Memory Controlled During Simulation

Current Database Information	
Build Date	April 10, 2015 03:41:57
Compiler	3-2015.05 Alpha
Verifier	J 2015.06 Alpha
Simulator	Alpha 8.0.27-78-SN-0015
Version	290000
Config	
Date	
Profile Start	290000
Profile Stop	29271423

All individual profiler reports contain the profile start and stop duration and time consumed is during this specified time only.

```
%simv -simprofile time -simprofile_start 290000 -simprofile_stop 29271423
```

Figure 99 HTML Report - Time/Memory Controlled During Profrpt

VCS build date	Apr 06 2015 20:41:37
Compiler version	3-2015.09-Alpha
Runtime version	3-2015.09-Alpha
Machine Name	vapintab6
Profile runner	memprof
Profile type	/cosmic/vtaholic12/nccor/5IMPROF_E/timing_line_based_profiler/SIMPROF_ME_DMA_SVD/simprof/runner
Creation date	Fri Apr 3 09:07:25 2015
Profile start	291000
Profile stop	29270000

Profile start time and stop time is profrpt time. All individual reports are for this duration only.

```
%simv -simprofile time -simprofile_start 290000 -simprofile_stop
29271423
%profrpt -start 291000 -stop 29270000 <all other options>
```

Diagnostics

Diagnostic messages indicate when the simulation profiling is turned on and turned off in the simulation log.

Note-[ON-SIMPROF] Simprofile is turned on
simulation profile is turned on at simulation time <time>

For example, simulation profiling is turned on at simulation time 290000.

Note-[OFF-SIMPROF] Simprofile is turned off
simulation profile is turned off at simulation time <time>

For example, simulation profiling is turned off at simulation time 2971423.

Limitations

The following are the limitations with this feature:

- Multiple start and stop on the same command line is not supported during profrpt stage.
- Multiple start and stop on the same command line is not supported during simulation stage.

Line-Based CPU Time Profiler

You can generate line-based profile report. This helps you to generate more accurate profile report and also helps you to efficiently identify the line in a module or construct that has taken most of the time.

Use Model

The use model remains the same as the use model of the reporting debug capabilities for each module. For more details, refer to the [Reporting Debug Capabilities for Each Module](#) section.

Line-based profile is enabled automatically along with the simprofile. The line-based profile report is generated in the source information window that is displayed when you click a module or a construct.

If you select the *Time Module View* or the *Time Construct View* in the *View* field in the left pane and then click the *GO* button, the right pane changes to show the *Time Module View* or the *Time Construct View* respectively as shown in [Figure 100](#) and [Figure 101](#).

Figure 100 The CPU Time Module View

Time Module View					
Module	Inclusive Time	Percentage	Exclusive Time	Percentage	
W3_600	1.90 s	24.64 %	1.90 s	24.64 %	
▶ Initial	1.90 s	24.64 %	1.90 s	24.64 %	
<0.00 %	0.00 us	0.00 %	0.00 us	0.00 %	
Total	1.90 s	24.64 %	1.90 s	24.64 %	

Page: 1

Figure 101 The CPU Time Construct View

Time Construct View			
Name	Time	Percentage	
▶ Initial	3.45 s	21.16 %	
total	3.51 s	21.55 %	

Page: 1

In *Time Module View* or *Time Construct View*, click the hyperlink of the source information in the *Construct Information* pane as shown in [Figure 102](#):

[Figure 102 Construct Information Pane](#)



The line-based profile report is displayed in a new browser as shown in [Figure 103](#):

[Figure 103 Line-Based Profile Report](#)

```

01 class Packet;
02
03 bit[100000:0] b;
04
0.13s 5.7% 05 function new();
0.01s 0.4% 06 b = 0;
0     0     07 endfunction
08
09 endclass
10

```

The first column displays the CPU time consumed by that line and the second column displays the percentage. The CPU time of a construct is not reported in the source view. The CPU time of a construct is equal to the total CPU time of all the lines in a construct.

Limitations

The following are the limitations with this feature:

- Supports only the CPU time profile.
- Supports only the HTML form for the line-based profile.

Isolating the Cost of Garbage Collection

VCS isolates the CPU time consumed by the garbage collection.

The cost of garbage collection is reported in the *Time Summary View* report. It is displayed as a sub-category under the KERNEL category.

Use Model

The use model remains the same as the simulation time slice based profiler. For details, refer to the [Simulation Time Slice Based Profiler](#) section.

Figure 104 The Time Summary View

Time Summary View	
Component	Percentage
VERILOG	90.45%
Module	90.45%
KERNEL	8.82%
Garbage Collection	1.08%
HSIM	0.73%
TOTAL	100.00%

Time consumption for garbage collection under KERNEL.



Isolating the Cost of Loading Design Database

In some large designs, loading the design database consumes lot of time and memory.

VCS isolates the CPU time and memory consumed for loading the design database.

The cost of loading the design database is reported in the *Time PLI/DPI/DirectC View* and *Peak Memory PLI/DPI/DirectC View* reports. It is displayed as a sub-category under the *PLI* category.

Use Model

The use model remains the same as the simulation time slice based profiler. For details, refer to the [Simulation Time Slice Based Profiler](#) section.

Figure 105 Time PLI/DPI/DirectC View

Time PLI/DPI/DirectC View	
Name	Percentage
PLI	5.91 %
\$countdrivers	5.91 %
\$countdrivers /remote/vgr9/REGRUN/TD	4.72 %
/unit_RUNTIME/simprofile/RTVIR_profile/	
test1.v:9885	
Load Design Database	1.57 %
\$countdrivers /remote/vgr9/REGRUN/TD	1.18 %
/unit_RUNTIME/simprofile/RTVIR_profile/	
test1.v:9760	
Load Design Database	0.79 %
Total:	5.91 %

Cost of loading the design database.

Figure 106 Peak Memory PLI/DPI/DirectC View

Peak Memory PLI/DPI/DirectC View		
Name	Percentage	Memory
PLI	30.10 %	4.19 M
\$countdrivers	30.10 %	4.19 M
\$countdrivers /remote/vgr9/REGRUN/TD	28.82 %	3.95 M
/unit_RUNTIME/simprofile/RTVIR_profile/		
test1.v:9885		
Load Design Database	4.34 %	618.50 K
\$countdrivers /remote/vgr9/REGRUN/TD	1.75 %	249.40 K
/unit_RUNTIME/simprofile/RTVIR_profile/		
test1.v:9760		
Load Design Database	1.02 %	144.83 K
Total:	30.10 %	4.19 M

Cost of loading the design database.

Third-Party Shared Library Profiler Report

Simprofile report displays the detailed information of the memory cost of the individual shared library besides the total memory cost of all third-party shared libraries.

Use Model

The use model remains the same as that of the simulation time slice based profiler. For details, refer to the [Simulation Time Slice Based Profiler](#) section.

Figure 107 Memory Size of the Individual Third-Party Shared Library

Peak Memory Summary View		
Component	Memory	Percentage
HSIM	5.90 M	4.84%
KERNEL	3.31 M	2.72%
VERILOG	576	0.00%
Module	436	0.00%
Package	140	0.00%
Anonymous	27.78 M	22.77%
Library/Executable	85.00 M	69.67%
VCS	82.00 M	67.21%
Third-party	3.00 M	2.46%
libc-2.5	1.34 M	1.10%
libstdc++	960.00 K	0.73%
libncurses	288.00 K	0.23%
libn-2.5	164.00 K	0.13%
libpthread-2.5	92.00 K	0.07%
libnss_files-2.5	48.00 K	0.04%
libgcc_s-4.1.2-20080825	48.00 K	0.04%
libdl-2.5	20.00 K	0.02%
TOTAL	122.00 M	100.00%

Profile report provide the detailed information for each individual library.

VHDL Unified Simulation Profiler Report

VCS offers module view and construct view under called process to the VHDL component. There is a category called VHDL-Kernel under the Kernel component.

The following illustration shows the time construct view:

Figure 108 Time Construct View

Time Construct View		
Name	Time	Percentage
►VHDL Process	234.67 ms	26.67 %
SIM_MDL	222.93 ms	25.33 %
►TOP_CLOCK_P1	11.73 ms	1.33 %
►Port	11.73 ms	1.33 %
total	246.40 ms	28.00 %

The construct view allows you to view the VHDL processes that consumed the most simulation time.

Limitations

The feature has the following limitations:

- VHDL 2008 constructs are not supported.

The HSIM View With the Simulation Profiler Report

In the simprofile summary view, the HSIM component consumes a significant part of gate-level simulation (GLS) and RTL designs.

VCS offers better visibility for designs with a high HSIM usage using the following methods:

- Maps the HSIM cost back to regular Verilog constructs.
- Breaks down the global HSIM bucket into small HSIM buckets inside each module or instance.

Figure 109 Time Summary View

Time Summary		
Component	Time	Percentage
VERILOG	921.6 ms	64.00%
KERNEL	451.2 ms	31.33%
HSIM	67.2 ms	4.57%
total	1.4 s	100%

Benefits

- In the *Time Summary* view, the cost of global HSIM bucket is decreased.
- In *Time Module* view, time cost on each module is increased. As the HSIM cost is mapped back to regular constructs, more regular constructs are visible under each module. When the HSIM cost cannot be mapped back to a regular construct, a new HSIM construct is added to the module.

Figure 110 Time Module View with HSIM component

▼ IQ_OP_REG	100.00 ms	7.25 %	100.00 ms	7.25 %
▼ Hsim	60.00 ms	4.35 %	60.00 ms	4.35 %
► Hsim {Module:IQ_OP_REG}	60.00 ms	4.35 %	60.00 ms	4.35 %
▼ Always	40.00 ms	2.90 %	40.00 ms	2.90 %
► NoName {line:27606}	20.00 ms	1.45 %	20.00 ms	1.45 %
► NoName {line:27702}	10.00 ms	0.72 %	10.00 ms	0.72 %
► NoName {line:27811}	10.00 ms	0.72 %	10.00 ms	0.72 %

Limitations

The following technologies are not supported in the unified profiler:

- The behavior becomes unpredictable if you fork child processes or threads in your C code, which might be called through PLI/DPI/DirectC interfaces.
- Incremental compilation is not yet supported for the unified profiler.
- OpenVera is not officially supported, VCS provides some information for reference but the name of the programs and constructs might be a bit different from the original one.
- Code coverage is not yet supported. The time and memory used by code coverage is counted to the corresponding HDL code.
- The accumulative views are available only in HTML format.
- The caller-callee views are available only in HTML format.
- No break down information is available for analog simulations. The information is available only in the summary form.
- No instance information is available for SystemVerilog Assertions (SVAs). Only time module view is displayed that helps you to determine which caller consume the most memory or time.

7

Diagnostics

This chapter covers various diagnostic tools and provides instructions on how to use these tools.

The following tasks are covered in this chapter:

- [Using Diagnostics](#)
- [Compile-time Diagnostics](#)
- [Runtime Diagnostics](#)
- [Post-Processing Diagnostics](#)
- [Sparse Memory Diagnostics](#)
- [Event Order Diagnostics](#)

You can divide the configuration of `onfail` into multiple configuration commands.

You can use the `config onfail disable` configuration command to disable this feature.

Example

The following command enables you to catch system faults, DT.* errors, and NOA errors:

```
config onfail enable sysfault {error DT.*} {error NOA}
```

You can also specify the above command as three different configuration commands:

```
config onfail enable sysfault
config onfail enable {error DT.*}
config onfail enable {error NOA}
```

► Use the following UCLI command to get a UCLI prompt when a runtime error occurs:

```
% simv -ucli -i file_name.tcl
```

or

```
% simv -ucli
```

```
ucli% do file_name.tcl
```

Where `file_name.tcl` is the Tcl file that contains the `config onfail enable` command and run script (see [Example 41](#)).

Note:

You must run the simulation using the `run` command by specifying it in a Tcl file. You can also specify the `config onfail enable` command in the same Tcl file, but instead, if you use `simv -ucli` at the UNIX prompt to run the simulation, then UCLI exits when there is a failure.

You can use all the above options in conjunction. For example, to print diagnostics for all the memories above 100MB, you can use the `-Xkeyopt=sparseDiag+elabThres+100` option.

Using Diagnostics

This section describes the following topic:

- [Using -diag Option](#)

Using -diag Option

Use the `-diag` option to enable the libconfig/timescale diagnostic messages at compile-time and VPI/VHPI diagnostic messages at runtime. The `-diag` option supports compile-time diagnostics on the `vcs` command-line and runtime diagnostics on the `simv` command-line.

Syntax

Following is the syntax of the `-diag` option:

```
-diag [,diag_arg] [,diag_arg]..
```

Where, `diag_arg` is a diagnostic argument. [Table 14](#) lists the supported diagnostic arguments.

Table 14 Supported Diagnostic Arguments

Argument	Use Model	Description
libconfig	<code>vcs -diag libconfig</code>	Enables the library binding diagnostics. For more information, see Libconfig Diagnostics .
timescale	<code>vcs -diag timescale</code>	Enables timescale diagnostics. For more information, see Timescale Diagnostics .

Table 14 Supported Diagnostic Arguments (Continued)

Argument	Use Model	Description
vpi	simv -diag vpi	Enables VPI diagnostics. For more information, see Diagnostics for VPI/VHPI PLI Applications .
vhpi	simv -diag vhpi	Enables VHPI diagnostics. For more information, see Diagnostics for VPI/VHPI PLI Applications .
all	vcs -diag all	Enables the libconfig and timescale diagnostics.
	simv -diag all	Enables the vpi and vhpi diagnostics.
help	vcs -diag help simv -diag help	Displays the following help message: Usage for -diag flag: -diag <option>,<option>,... Options: all Enable all diagnostics help Display this message libconfig Library binding diagnostics (compile time) timescale Timescale diagnostics (compile time) vpi VPI diagnostics (simulation time) vhpi VHPI diagnostics (simulation time)

Compile-time Diagnostics

This section describes the following topics:

- [Libconfig Diagnostics](#)
- [Timescale Diagnostics](#)
- [Generating Information on Unused Libraries at vlogan](#)
- [Generating Information on Unused Libraries at VCS](#)
- [Obtaining Statistics on Package Utilization](#)

Libconfig Diagnostics

You can use the libconfig option, as shown below, to enable libconfig diagnostics:

```
% vcs -diag libconfig
```

This option provides the library binding diagnostics at compile-time. It generates physical mappings of user-defined libraries and the default work library specified by VCS.

For each VHDL/Verilog instance, this option generates the instance name, location, binding rule, and entity-architecture pair/module to which it is bound.

Example

Consider the following test case:

```
leaf.vhd
=====
entity leaf is
end entity leaf;

architecture behv of leaf is
begin
end architecture;

mid.vhd
=====
entity mid is
end entity mid;

architecture behv of mid is
  component leaf
    end component leaf;
begin
  a0: leaf;
end architecture;

top.v
=====
module top();
  mid inst1 ();
endmodule
```

Perform the following commands:

```
% vhdlan leaf.vhd -work lib1
% vhdlan mid.vhd -work lib1
% vlogan top.v -work lib2
% vcs top -diag libconfig -l log
```

Following is the output:

```
Setup library mapping:
  DEFAULT : /remote/vtghome13/diag./work/
  LIB1 : /remote/vtghome13/diag./lib/
  LIB2 : /remote/vtghome13/diag./lib/
Work logical library name set to 'DEFAULT'.
Default library search order:
```

```

DEFAULT
LIB1

instance: LIB1.top
          "/remote/vtghome13/diag/top.v", 1
rule: Top Module
module: LIB1.top
          "/remote/vtghome13/diag/top.v", 1

Top Level Modules:
top
instance: top.inst1
          "/remote/vtghome13/diag/top.v", 3
rule: Direct Instantiation
entity: LIB1.MID
          "/remote/vtghome13/diag/mid.vhd", 3
architecture: BEHV
          "/remote/vtghome13/diag/mid.vhd", 6

instance: top.inst1.A0
          "/remote/vtghome13/diag/mid.vhd", 10
rule: Default Binding
entity: LIB1.LEAF
          "/remote/vtghome13/diag/leaf.vhd", 4
architecture: BEHV
          "/remote/vtghome13/diag/leaf.vhd", 7

```

Note:

- If the `-l` option is specified, the output is dumped into the corresponding text file.
- If the `-sml` option is specified, smartlog output is also dumped into the corresponding smartlog file.

Timescale Diagnostics

You can use the `timescale` option, as shown below, to enable timescale diagnostics:

```
% vcs -diag timescale
```

This option generates timescale diagnostic message for each module during VCS elaboration phase. This allows you to understand how VCS has scaled delays in its design, and helps you to quickly identify, localize and fix the timescale issues.

Note:

- The output is printed on the `STDOUT` by default.
- If the `-l` option is specified, the output is dumped into the corresponding text file.
- If the `-sml` option is specified, smartlog output is also dumped into the corresponding smartlog file.

Example

Example 1: Module has `timescale

Consider the following test case `test.v`, which contains module `test` with ``timescale as 1ns/1ns`:

```
`timescale 1ns/1ns
module test;
  initial
    $printtimescale;
endmodule
```

Enabling timescale diagnostics at elaboration time using `-diag timescale`:

```
% vcs test.v -diag timescale
```

Following is the output:

```
Parsing design file 'test.v'
Top Level Modules:
  test
TimeScale is 1ns/1ns
module 'test' gets time unit '1ns' from source code
  '/remote/vgscratch7/timescale_diag/tests/cft/sva_bind/11_svb/Source/test
.v', 1
module 'test' gets time precision '1ns' from source code
  '/remote/vgscratch7/timescale_diag/tests/cft/sva_bind/11_svb/Source/test
.v', 1
Starting vcs inline pass...
1 module and 0 UDP read.
recompiling module test
if [ -x ..//simv ]; then chmod -x ..//simv; fi
g++ -o ..//simv -melf_i386 -m32      -Wl,-whole-archive
  -Wl,-no-whole-archive _vcsobj_1_1.o  5NRI_d.o
...
..//simv up to date
```

From the above output, you can figure out which module gets what timescale at elaboration, and also the reason why and from where the module got that timescale.

```
module 'test' gets time unit '1ns' from source code
  '/remote/vgscratch7/timescale_diag/tests/cft/sva_bind/l1_svb/Source/test
.v', 1
module 'test' gets time precision '1ns' from source code
  '/remote/vgscratch7/timescale_diag/tests/cft/sva_bind/l1_svb/Source/test
.v', 1
```

In the above example, as mentioned `timescale 1ns/1ns on line# 1, the module gets the time unit of 1ns and time precision of 1ns.

Example 2: Passing -timescale from vcs command-line

Consider the following testcase test.v:

```
module test;
  initial
    $printtimescale;
endmodule
```

Perform the following command:

```
% vcs test.v -diag timescale -timescale=1ns/1ns
```

Following is the output:

```
Parsing design file test.v
Top Level Modules:
  test
TimeScale is 1ns/1ns
module 'test' gets time unit '1ns' from vcs command option
module 'test' gets time precision '1ns' from vcs command option
Starting vcs inline pass...
1 module and 0 UDP read.
recompiling module test
if [ -x ../simv ]; then chmod -x ../simv; fi
g++ -o ../simv -melf_i386 -m32      -Wl,-whole-archive
     -Wl,-no-whole-archive _vcsobj_1_1.o 5NrI_d.o
...
../simv up to date
```

In the following command, timescale is passed at elaboration using the -timescale option.

```
% vcs test.v -diag timescale -timescale=1ns/1ns
```

The diagnostics message printed on the output is as follows:

```
module 'test' gets time unit '1ns' from vcs command option
module 'test' gets time precision '1ns' from vcs command option
```

Generating Information on Unused Libraries at vlogan

VCS allows you to generate information on unused libraries at vlogan stage for three-step flow. VCS displays a lint message if there are more libraries specified than required with the `-liblist` option at vlogan stage.

Use Model

You can use the `-diag libdepends` option at vlogan stage, as shown, to generate information on unused libraries at vlogan stage:

```
% vlogan -diag libdepends
```

VCS displays a lint message `ULILL-L` for unused libraries and prints the information on required libraries in `vlogan_statistics.out` file in the library path.

Usage Example

Consider the following examples:

Example 34 Generating Unused Libraries at vlogan stage

```
tb.v
=====
`define SUB d.u.s
package p1;
int option;
wire w1;
endpackage

module tb;
import p1::*;
`ifdef INIT initial
$display("TB1 is runing"); `endif
dut d();
bot bot1();
endmodule

module bot();
import p1::*;
class B;
endclass
B b = new;
endmodule
```

Perform the following command:

```
% vlogan -sverilog tb.v -diag libdepends -liblist temp2 -liblist temp3
```

VCS generates the following lint message at compile time and shares the list of unused libraries passed at vlogan step using the `-liblist` option:

```
Lint-[ULILL-L] Unused library in liblist
Current analysis step does not require the following libraries:
temp2
temp3
```

VCS also prints the following information of the used libraries in a file. This file is generated as `<library_folder>/vlogan_statistics.out` file.

```
Module: tb
Required Packages:
    p1
Required Libraries:
    DEFAULT
Module: bot
Required Packages:
    p1
Required Libraries:
    DEFAULT
```

Generating Information on Unused Libraries at VCS

VCS generates information on unused libraries passed at VCS stage. VCS displays a lint message that provides the information on the unused libraries mentioned in the `synopsys_sim.setup` file. The lint message also shows information on the amount of disk space consumed by each of the unused libraries.

Use Model

You can use the `-diag libusage` option at VCS stage, as shown, to generate the information on unused libraries at VCS stage:

```
% vcs -diag libusage
```

VCS displays a lint message `LINU-L` for unused libraries and prints the information on required libraries in `elaboration_statistics.out` file in the library path.

Example 35 Generating Unused Libraries at VCS stage

```
tb.v
=====
`define SUB d.u.s
package p1;
int option;
wire w1;
endpackage

module tb;
import p1::*;


```

```

`ifdef INIT initial
  $display("TB1 is runing"); `endif
dut d();
bot bot1();
endmodule

module bot();
import p1::*;
class B;
endclass
B b = new;
endmodule

synopsys_sim.setup
=====
WORK > DEFAULT
DEFAULT:./work
lib3:./lib3

```

Perform the following commands:

```
% vlogan -sverilog tb.v
% vcs tb -diag libusage
```

VCS generates the following lint message at compile time and shares the list of unused libraries passed at vcs step using the `-libusage` option

```
Lint-[LINU-L] Library included but not used
None of the cells from the following libraries has been used:
lib3 (disk size: 4593 bytes)
```

VCS also prints the following information of the used libraries in a file. This file is generated as `<library_folder>/elaboration_statistics.out` file.

```
Library Cell Usage:
DEFAULT: (7/8)
```

As shown in the output, 7 out of 8 cells are used. To get details on the used cells, use the `-diag libdepends=verbose` option.

Obtaining Statistics on Package Utilization

VCS obtains statistics on the consumption of packages in the design. The statistics show information on the number of classes, tasks, and functions from the package used in the design.

Use Model

You can use the `-diag pkgusage` compile-time option to obtain statistics on package utilization.

VCS reports the package utilization information in the `elaboration_statistics.out` file. This file is generated in the elaboration directory.

Example 36 Obtaining Statistics on Package Utilization

```
tb.v
=====
`define SUB d.u.s
package p1;
int option;
wire w1;
endpackage

package p12;
import p1::*;

function f();
    $display(option);
endfunction
endpackage
module tb;
    import p12::*;
    `ifdef INIT initial $display("TB1 is running"); `endif
    dut d();
initial begin
    f(); //XMR
end
endmodule
```

Perform the following commands:

```
% vlogan -sverilog tb.v
% vcs tb -diag pkgusage
```

VCS prints the following information of the used libraries in the `elaboration_statistics.out` file.

```
Package usage diagnostics:std: Classes: 0/6, Functions: 0/0, Tasks: 0/0
p1: Empty

Package usage diagnostics:std: Classes: 0/6, Functions: 0/0, Tasks: 0/0
p1: Empty
p12: Classes: 0/0, Functions: 1/1, Tasks: 0/0
```

Runtime Diagnostics

This section describes the following topics:

- [Diagnostics for VPI/VHPI PLI Applications](#)
 - [Keeping the UCLI/Verdi Prompt Active After a Runtime Error](#)
 - [Diagnosing Quickthread Issues](#)
-

Diagnostics for VPI/VHPI PLI Applications

As per LRM, VPI/VHPI remain silent when an error occurs. The application checks for error status to report an error. If error detection mechanisms are not in place, the C code of the application must be modified and recompiled. In addition, you may need to recompile the HDL code, if required.

However, you can use the following new runtime diagnostics options to make the PLI application to report errors without code modification:

- `-diag vpi`
- `-diag vhpi`

Furthermore, reporting provides you the information related to the HDL code context, wherever applicable, to help fix problems with a faster turnaround time.

Note:

- If the `-l` option is specified, the output is dumped into the corresponding text file.
- If the `-sml` option is specified, smartlog output is also dumped into the corresponding smartlog file.

For example, consider the following test case `tokens.v` and files `value.tab` and `value.c`.

Example 37 tokens.v

```
module top;
    reg r;

    initial begin
        #5;
        $putValue("sys_top.rst", 1'b1);

        #1 $finish;
    end
endmodule
```

```

module sys_top;
  wire rst;

  assign db.A = rst;
endmodule

module db;
  wire Y;
  wire A;

  my_buf b1(Y, A);

  initial begin
    end
endmodule

module my_buf(Y, A);
  output Y;
  input A;

  buf #5 (Y, A);
endmodule

```

Example 38 value.tab

```
$putValue call=put_value acc=rw:top
```

Example 39 value.c

```

#include <string.h>
#include <stdlib.h>
#include <assert.h>
#include "sv_vpi_user.h"

void put_value() {
  vpiHandle sysTfH, argI, objH, valueH;
  s_vpi_value value;
  s_vpi_time time_s;
  int format;
  p_vpi_value value_p;
  p_vpi_time time_p;

  sysTfH = vpi_handle(vpiSysTfCall, 0x0);

  argI = vpi_iterate(vpiArgument, sysTfH);

  objH = vpi_scan(argI);
  valueH = vpi_scan(argI);

```

```

if (vpi_get(vpiType, objH) == vpiConstant) {
    value.format = vpiStringVal;
    vpi_get_value(objH, &value);
    vpi_free_object(objH);
    if(strcmp(value.value.str, "null")) {
        objH = vpi_handle_by_name(value.value.str, 0x0);
    } else {
        objH = 0x0;
    }
}

time_p = 0x0;
value.format = vpiIntVal;
vpi_get_value(valueH, &value);
value_p = &value;

vpi_put_value(objH, value_p, time_p, vpiNoDelay);
}

```

Compile and run the `tokens.v` code as follows:

Two-step Flow:

```
% vcs -sverilog +vpi -P value.tab value.c tokens.v
% simv -diag vpi
```

Three-step Flow:

```
% vlogan tokens.v
% vcs top -sverilog +vpi -P value.tab value.c
% simv -diag vpi
```

Here, the user application tries to write a value on the `sys_top.rst` signal, but there is no write permission enabled on `sys_top`. So VPI generates an error message and prints the HDL information, as follows:

```
Error-[VPI-WPNEN] VPI put value error
At time 5, in PLI routine called from tokens.v, 6
  In vpi_put_value call, write permission not enabled.
  Please add capability 'wn' to signal 'sys_top.rst' of module 'sys_top'.
  Please refer to the VCS User Guide, Section 'Specifying ACC Capabilities
  PLI functions' in the chapter 'Using PLI' for further details.

At time 5, in the PLI application '$putValue' called from tokens.v, 6:
  vpiSeverity - vpiError
  PLI Routine - vpi_put_value
  Reference Object - rst
  Reference Scope - sys_top
  Reference vpiType - vpiNet
  Path - /remote/us01home17/Downloads/12-09/VPI_EM/tokens.v, 14
  Delay Propagation Method - 1
```

Keeping the UCLI/Verdi Prompt Active After a Runtime Error

VCS allows you to debug an unexpected error condition by not exiting and keeping the UCLI or Verdi command prompt active for debugging commands.

Verdi or UCLI command prompt remains active when there is an error condition, allowing you to examine the current simulation state (the simulation stack, variable values, and so on) so you can debug the error condition.

UCLI Use Model

If simv is executed from UCLI, perform the following steps to enable this feature:

1. Specify the following UCLI configuration command in a Tcl file (see [Example 41](#)) or in `$HOME/.synopsys_ucli_prefs.tcl` file:

```
config onfail enable [failure_type]
```

Where `failure_type` is optional. It allows you to specify the failure type. The following table lists the types of failures which are normally observed during an unexpected runtime error.

Table 15 Types of Failures

Failure Type	Failure Description
sysfault	Assertion or signal (including segfault)
{error <regex>}	Error for which the tag matches regex. The tag of an error can be seen in the error message (Error-[TAG]).
fatal	Fatal error for which VCS currently dumps a stack trace.
all	All failures (default)

2. You can divide the configuration of `onfail` into multiple configuration commands.

You can use the `config onfail disable` configuration command to disable this feature.

Example

The following command enables you to catch system faults, DT.* errors, and NOA errors:

```
config onfail enable sysfault {error DT.*} {error NOA}
```

You can also specify the above command as three different configuration commands:

```
config onfail enable sysfault
config onfail enable {error DT.*}
config onfail enable {error NOA}
```

- Use the following UCLI command to get a UCLI prompt when a runtime error occurs:

```
% simv -ucli -i file_name.tcl
```

or

```
% simv -ucli
ucli% do file_name.tcl
```

Where `file_name.tcl` is the Tcl file that contains the `config onfail enable` command and run script (see [Example 41](#)).

Note:

You must run the simulation using the `run` command by specifying it in a Tcl file. You can also specify the `config onfail enable` command in the same Tcl file, but instead, if you use `simv -ucli` at the UNIX prompt to run the simulation, then UCLI exits when there is a failure.

Automating User Actions on Failure

You can create the `onfail` routine to automate some actions (like printing specific message, collecting data into a file, and so on) when an unexpected crash happens during runtime. You can create this routine in your script or in the `.synopsys_ucli_prefs.tcl` file.

If you declare this routine, and the `onfail` configuration is enabled, then `simv` calls the `onfail` routine before going into the UCLI prompt. If you do not want to go into the UCLI prompt, you can call the UCLI `exit` command from the routine.

Verdi Use Model

By default, Verdi enables the `onfail` configuration on all types of failures. Verdi systematically enables the `onfail` configuration on all error types.

When you enable the `onfail` configuration, `simv` stays active and continue to respond to Verdi queries. Also, Verdi shows the location of the error with the simulation pointer (pink arrow in the source view), and the Stack pane shows the current HDL stack. You can use value annotation to obtain signal values in order to debug the issue.

UCLI Usage Example

Consider the following test case `test.v`. This code causes simv to exit during simulation:

Example 40 UCLI Prompt on Error Test Case (test.v)

```
module test;
    class Packet;
        int _a;

        function void set (int a);
            _a = a;
        endfunction
    endclass

    initial begin
        Packet pkt;
        reg a;
        pkt.set(a);
    end
endmodule
```

Compile the `test.v` file, as shown below:

Two-step Flow:

```
% vcs -sverilog -debug_access+all test.v
```

Three-step Flow:

```
% vlogan test.v
```

```
% vcs test -sverilog -debug_access+all
```

If you run the above test case using the `simv -ucli` command, VCS generates the following NOA error message:

Figure 111 NOA Error Message

```
Error-[NOA] Null object access
test.v, 13
The object is being used before it was constructed/allocated.
Please make sure that the object is newed before using it.

#0 in unnamed$$_0 at test.v:13
#1 in test

V C S   S i m u l a t i o n   R e p o r t
Time: 0
CPU Time:      0.790 seconds;          Data structure size:  0.0Mb
Mon Jan 23 02:59:51 2012
```

Create the following Tcl file to catch the above error and analyze it inside an onfail routine:

Example 41 Tcl File (test.tcl)

```
onfail {
    set err_msg "Stopped in "
    append err_msg [scope]
    puts $err_msg
}
config onfail enable {error NOA}
run
```

Run the `test.tcl` file using the following command to keep the UCLI prompt active after the NOA error, as shown in [Figure 112](#):

```
% simv -ucli -i test.tcl
```

Figure 112 Viewing the UCLI Prompt After Failure

```
ucli% config -onfail enable {error NOA}
ucli% run

Error-[NOA] Null object access
test.v, 13
The object is being used before it was constructed/allocated.
Please make sure that the object is newed before using it.

#0 in unnamed$$_0 at test.v:13
#1 in test

file test.tcl, line 7: System Fault
Stopped in test

Pause in file test.tcl, line 7
pause%
```

The onfail routine is executed after the NOA error is generated.

Limitations

- You cannot specify an onfail routine to be executed on error in Verdi.

Diagnosing Quickthread Issues

VCS is now equipped with a better mechanism to report VCS runtime crashes caused by certain problems with quickthreads used during VCS runtime. You will get clear feedback as to what went wrong and which thread is causing the crash thereby enabling you to take specific action to circumvent the issue.

Diagnosing Quickthread Issues in DPI

While calling an import DPI routine that either calls any SystemVerilog blocking/time consuming task or is declared as a context task, VCS creates quickthread with a runtime memory of 256 KB (by default) for the call chain that originated from the import DPI routine. Such import DPI routine is also called a heavy weight DPI routine. If the call chain from the import DPI routine uses more memory than the pre-allocated buffer (due to large local variables and/or a deep call chain), it causes a segmentation fault and the following runtime error message is generated:

```
Note-[VCS-QTHREAD-OVERRUN] Stack of quickthread maybe too small
The simulation received a fatal segmentation violation signal SEGV and
will end because it accessed protected stack guard memory. This memory
belongs to the thread 'top'. It is likely, but not certain that a stack
overflow in this thread caused the segmentation violation (SEGV). It may
also be caused by a different, unknown problem and the quickthread is
not related.
```

The suspected quickthread belongs to the DPI domain.
 Its stack has a size of 4194300.96 K bytes and is located from address '0x9eb43dc' to '0x9eb5000'.
 Its redzone has a size of 4.00 K bytes and is located from address '0x9eb5000' to '0x9eb4000'.
 The SEGV happened at address '0x9eb4390' which is 3184 bytes into the redzone.
 Increase the stack size for this thread and check whether this solves the problem. See the VCS user guide for more information.

You should give a conservative (less restrictive) estimate about the DPI quickthread runtime memory based on the import DPI routine code. If the default runtime memory of 256 KB is too restrictive, the DPI quickthread runtime memory size can be set with the environment variable `DPI_QSTACK_SIZE` in the following ways:

- Before running the simulation as follows:

```
% setenv DPI_QSTACK_SIZE <number>
```

Or

- From the DPI application using the `setenv` function as follows:

```
setenv ("DPI_QSTACK_SIZE", "<number>", 1);
```

Here, the `<number>` should be provided in bytes.

For example, to set the limit of 8 KB, the value should be $8 * 1024 = 8192$.

If the default DPI quickthread runtime memory size is overwritten and a heavy weight DPI routine is invoked, VCS issues the following message at runtime:

```
Note-[DPI-RTMBSS] DPI runtime memory buffer size set
The size of the runtime memory buffer for invoking time consuming DPI
task(s) has been overwritten from default 256KB to 8192B (8KB) by
environment variable DPI_QSTACK_SIZE.
```

Diagnosing Quickthread Issues in SystemC

VCS reports runtime crashes in the following two scenarios:

- A quickthread overruns its allocated stack
- Simulation runs out of memory due to quickthread stacks

Following are the recommended guidelines to diagnose a stack overrun, in case you use the runtime options mentioned in the following section:

- If you suspect that the simulation crashes, because one or more SC threads overrun their stacks, then first try to increase the stack to a large value, for example, 100 MB.

```
% simv -sysc=stacksize:100M
```

- If the crash goes away, then there is a chance that a stack overrun has occurred before. If so, then leave the stack at its previous size (which is too small), but increase the stack guard size to a large value (for example, 200 MB).

This increases the chance for the simulation to abort with an SEGV error on the first time, when a stack overrun occurs. Compile all SystemC source code with debug information, and start the simulation from a debugger such as gdb or from Verdi/CBug.

```
syscan -cflags -g file1.cpp file2.cpp ...
...
gdb simv
(gdb) run -sysc=stackguardsize:200M
... SEGV occurred in file1.cpp line 123 ...
```

Identify the SC thread that used more memory and increase its stack size by calling `set_stack_size()` in the constructor. For more information on `set_stack_size()`, see the SystemC LRM.

The default stack size of a SystemC thread (either `SC_THREAD` or `SC_CTHREAD`) is 1MB and the default stackguard size is 16KB.

If a quickthread overruns its allocated stack, then it will probably try to read/write into its redzone. This causes an SEGV with the diagnostic message. Here is an example:

```
Error-[SC-VCS-QTHREAD-OVERRUN] Stack of quickthread maybe too small
The simulation received a fatal segmentation violation signal SEGV and
will end, because it accessed protected stack guard memory. This memory
belongs to the thread 'top.ref_model_0.cpu.ALU'. It is likely, but not
certain that a stack overflow in this thread caused the segmentation
violation (SEGV). It may also be caused by a different, unknown problem
and the quickthread is not related.
```

The suspected quickthread belongs to SystemC domain.

Its stack has a size of 60 K bytes and is located from address
'0x800a00000' to '0x800a0efff'.

Its redzone has a size of 4 K bytes and is located from address
'0x800a0f000' to '0x800a0ffff'.

The SEGV happened at address '0x800a0f004' which is 5 bytes into the
redzone.

Increase the stack size for this thread and check whether this
solves the problem. This can be done by calling the `stack_size()`
method within the `SC_CTOR`. Alternatively, start the simulation with
'simv -sysc=stacksize:10M'. See the VCS SystemC user guide for more
information.

Limitations

The `SC-VCS-QTHREAD-OVERRUN` diagnostic applies only to quickthreads. It is not available if you use POSIX threads in SystemC by defining environment `SYSC_USE_PTHREADS`.

Simulation Runs Out of Memory Due to Quickthread Stacks

Each quickthread allocates memory for its stack. Simv may run out of memory due to this. When allocation of memory for a SystemC stack of a quickthread fails, a message like the following is printed:

```
Error-[SC-VCS-QTHREAD-ALLOC] Thread memory allocation failed
The creation of thread 'top.sc_thread_04' in the SystemC domain failed
because its stack of 64MB could not be allocated. Currently, 149MB
stack
memory are allocated by 95 threads.

Details about stack allocation:
(sorted by size in decreasing order)
32MB total (31.9MB stack + 19.9KB guard) in SystemC:top.sc_thread_05
16MB total (15.9MB stack + 19.9KB guard) in SystemC:top.sc_thread_06
8.01MB total (7.99MB stack + 19.9KB guard) in SystemC:top.sc_thread_07
(~50 lines removed, we show approx. 50..60 stack frames , ordered by
size, largest first)
...(truncated)...
Total: 149MB qthread stack memory used in 95 threads.
```

If this was a 32 bit simulation, consider a 64 bit simulation. You can also decrease the stack size for other threads. This can be done by calling the `stack_size()` method within the `SC_CTOR`. Alternatively, start the simulation with e.g. '`simv -sysc=stacksize:500k`'. See the VCS user guide, chapter Using SystemC for more information.

Reducing or Turning Off Redzones

You can decrease the number of redzones or turn them off altogether if the number of quickthreads you are using is exceedingly large. For instance, if the quickthreads are reaching the limit set in your OS, then some of the operations may fail. To avoid such a situation, you may want to decrease the number of the redzones or turn them off completely. Though the diagnostic is not supported when a particular thread overruns its stack, you would still increase the chances of running your simulation without any issues.

You can use the following environment variable to either decrease the number of redzones or turn them off completely. To decrease the number of redzones, you must set the following environment variable to a value greater than 2000 and less than 30000. For example:

```
setenv SNPS_VCS_SYSC_RESERVED_MAP_COUNT 10000
```

Setting the above environment variable to a value higher than 30000 will turn off the redzones completely.

Post-Processing Diagnostics

This section describes the following topic:

- [Using the vpdutil Utility to Generate Statistics](#)

Using the vpdutil Utility to Generate Statistics

The `vpdutil` utility generates statistics of the data in the VPD file. This utility takes a single VPD file as an input. You can specify options to this utility to query at design, module, instance, and node levels.

This utility supports time ranges and input lists for query on more than one object. Output is in ASCII to stdout with option to redirect to an output file.

The `vpdutil` Utility Syntax

The syntax of the `vpdutil` utility is as follows:

```
vpdutil <input_vpd_file>
    [-help]
    [-vc_info]
    [-tree [-lvl <level>] [-source]]
    [-vc_info_detail]
    [-info]
    [-design]
    [-find_forces]
    [-start <Time> -end <Time>]
    [-find_glitches]
    [output_file_name]
```

Options

`-h/help`

Displays the options to be used with the `vpdutil` application.

`output_file_name`

Writes the output of the `vpdutil` application to a file instead of stdout.

Options for VPD File Information

`-info`

Prints the basic information present in the header of the VPD file.

Options for Design Information

`-design`

Prints statistics about static design hierarchy in the VPD file.

`-tree`

Prints the full hierarchy tree in the VCD-like (not vcd compatible) format.

`-lvl <level>`

Prints the tree with the hierarchy depth=level.

`-source`

Prints source file/line data to tree.

Options for Value Change Information

`-vc_info`

Displays value change information with the number of dump off events, force events, glitch events, and repeat count events.

`-vc_info_detail`

Prints the detailed value change summary statistics about the given VPD file.

`-find_forces`

Displays forces on node and the times when forces occurred.

`-start <Time> -end <Time>`

Enables the collection of value change data between start time to end time.

`-find_glitches`

Prints the list of nodes with glitches and the time when glitches occurred, if the glitch capturing is enabled during the simulation.

Sparse Memory Diagnostics

You can use the `-Xkeyopt=sparseDiag` option at compile time to enable sparse memory diagnostics. This option prints detailed diagnostics about the memories that are inferred as sparse and the memories that are not inferred as sparse. This option is also useful for debugging and determining opportunities for sparse memory inferencing. This section describes the following:

- [Compile Time Options](#)
- [Runtime Options](#)
- [Sparse Disable Options](#)

Compile Time Options

The compile time diagnostics options are as follows:

`-Xkeyopt=sparseDiag`

This is a basic option. All the other options are used along with this option.

This option prints all the memories above the default threshold of 512 MB and mention sparse inferred/non-sparse. For the memories which are not inferred as sparse, it prints the offending construct which inhibits the memory from being inferred as sparse.

Table 16 Options with -Xkeyopt=sparseDiag

Options with <code>-Xkeyopt=sparseDiag+<..></code>	Description
print	This option prints information about memory above threshold, whether they are inferred sparse or not. Default threshold value is 512MB.
printDetails	This option prints information about memory above threshold, whether they are inferred sparse or not. For the memories which are not inferred as sparse, it prints the offending construct which inhibits the memory from being inferred as sparse. The default threshold value is 512MB.
elabThres+<int>	This option specifies the threshold above which all memories are considered under <code>sparseDiag</code> . However, if <code>elabThres</code> is not specified, 512MB is considered the default threshold. The size must be specified in MB. This option can be specified alone or at the right end of the option. You can compute the threshold (in MB) using the <i>Elaboration Threshold in MB</i> = $P * U * f * (Y? 2: 1) / (8 * 1024 * 1024)$ formula. Here, P = Packed Dim, U = Unpacked Dim, F = Flat Memory Count and Y = 4State
force	This option enables memories above a specified threshold to be sparse. The default threshold value is 512MB.
noDynUnsparse	This option stops dynamic unsparsing for memories at runtime.

Table 16 Options with -Xkeyopt=sparseDiag (Continued)

Options with -Xkeyopt=sparseDiag+<...>	Description
logFile+<FileName>	This option prints all sparse memory diagnostics in specified file name. If this option is not specified, it prints in <code>snpS_SparseMemComp<name>.log</code> file which is available at the present working directory. For PC flow, this logfile is created in child partition directories. Here the name is generated using PID, date combination so that the log file name is unique for every compilation. This option can be specified alone or at the right end of the option combination.
noDesignNames	This option does not print module/node names used in a design.

Note:

You can use all the above options in conjunction. For example, to print diagnostics for all the memories above 100MB, you can use the `-Xkeyopt=sparseDiag+elabThres+100` option.

Example

Consider the following testcase:

Example 42 test.v

```
module tb;

    modA A1();
    modA A2();

endmodule

module modA();

    reg [40:0] mda1[40000000:0];
    bit [40:0] mda2[400:10000000];

    initial begin
        mda1[0][1] = 1'b0;
        mda2[500][1] = 1'b1;
        $finish;
    end

endmodule
```

Run the example using the following command:

```
% vcs -sverilog -Xkeyopt=sparseDiag test.v
```

The output generated is as follows:

```
SPMEM_ELAB_THRES_DIAG:: Scope "modA", Node "mda1" is Inferred Sparse
BitSizes=41 ArrSize=40000001 FlatInstCnt=2 State=4STATE
```

In this example, if you want to find the threshold, you can use the *Elaboration Threshold in MB = P * U * f * (Y? 2: 1) / (8 * 1024 * 1024)* formula. Here, P= 41, U = 40000001, f = 2, Y= 2. Therefore,

Elab Threshold in MB = $41 * 40000001 * 2 * 2 / (8 * 1024 * 1024) = 782.012$ MB

Runtime Options

The runtime diagnostic options are as follows:

```
-sparse+logFile[+<FileName>] -sparse+stats
```

This option prints all the sparse memory diagnostics in specified file name. If this option is not specified, it prints in `snps_SparseMemRun<name>.log` available at the present working directory. Here, the name is generated using PID, date combination so that the log file name is unique for every compilation. This option can be specified alone or at the right end of the option combination.

For example, consider the example [Example 42](#). To print the sparse memory diagnostics in file `abc`, use the following commands:

```
% vcs -sverilog -Xkeyopt=sparseDiag test.v
```

```
% simv -sparse+logFile+abc -sparse+stats
```

A file `abc` is created in your working directory. The following is the format of the `abc` log file.

Figure 113 abc logfile

```
-----SPARSE MEMORY SUMMARY-----
test.v:10 :: mda1
Scope : tb.A1
WordSize: 41, ArraySize: 40000001 LaneSize: 64
TotalLanes: 625001      LanesWritten: 1 Occupancy:
0.000160%
4-state, STD/BYTE LAYOUT, vcs-memory:12 units
Entries:1,
test.v:11 :: mda2
Scope : tb.A1
WordSize: 41, ArraySize: 9999601 LaneSize: 64
TotalLanes: 156244      LanesWritten: 1 Occupancy:
0.000640%
2-state, NON-STD/WORD LAYOUT, vcs-memory:8 units
Entries:64,
test.v:10 :: mda1
Scope : tb.A2
WordSize: 41, ArraySize: 40000001 LaneSize: 64
TotalLanes: 625001      LanesWritten: 0 Occupancy:
0.000000%
4-state, STD/BYTE LAYOUT, vcs-memory:12 units
Entries:0,
test.v:11 :: mda2
Scope : tb.A2
WordSize: 41, ArraySize: 9999601 LaneSize: 64
TotalLanes: 156244      LanesWritten: 0 Occupancy:
0.000000%
2-state, NON-STD/WORD LAYOUT, vcs-memory:8 units
Entries:0,
-----DONE-----
```

-sparse+noDynUnsparse

This option stops dynamic unsparsing of all the design memories.

-sparse+stats

This option reports sparse memory stats at the end of simulation, that is, module name, node name, dynamically unsparsed or not, unpacked dimension, packed dimension and count of memory words written till end of simulation.

Consider the example [Example 42](#). Run the example using the following commands:

- % vcs -sverilog -Xkeyopt=sparseDiag test.v
- % simv -sparse+stats

The sparse memory summary report is generated as follows:

Figure 114 Sparse Memory Summary Report

```
-----SPARSE MEMORY SUMMARY-----
test,v;10 :: mda1
Scope : tb.A1
WordSize: 41, ArraySize: 40000001      LaneSize: 64      TotalLanes: 625001      LanesWritten: 1      Occupancy: 0.000160%
4-state, STD/BYTE LAYOUT, vcs-memory:12 units
Entries:1,
test,v;11 :: mda2
Scope : tb.A1
WordSize: 41, ArraySize: 9999601      LaneSize: 64      TotalLanes: 156244      LanesWritten: 1      Occupancy: 0.000640%
2-state, NON-STD/WORD LAYOUT, vcs-memory:8 units
Entries:64,
test,v;10 :: mda1
Scope : tb.A2
WordSize: 41, ArraySize: 40000001      LaneSize: 64      TotalLanes: 625001      LanesWritten: 0      Occupancy: 0.000000%
4-state, STD/BYTE LAYOUT, vcs-memory:12 units
Entries:0,
test,v;11 :: mda2
Scope : tb.A2
WordSize: 41, ArraySize: 9999601      LaneSize: 64      TotalLanes: 156244      LanesWritten: 0      Occupancy: 0.000000%
2-state, NON-STD/WORD LAYOUT, vcs-memory:8 units
Entries:0,
-----DONE-----
```

-sparse+unsparselThresPercent+<Int>

This option specifies the threshold Integer percentage beyond which the memory is dynamically unsparsed. By default, the threshold is 20%. This option can be specified alone or at the right end of the option.

Sparse Disable Options

The following are the sparse disable options:

-Xkeyopt=sparseDisable

This option disables sparse inferencing. It disables the auto inferencing and unsparsel pragma specified by you and config file based sparse memories.

-Xkeyopt=sparseDisable+autoInfer

This option disables auto inferencing of sparse memories. The memories that are marked as sparse using either pragma or config file is considered for sparsing.

Event Order Diagnostics

The event order diagnostics feature helps you achieve consistent simulation by eliminating race scenarios. It helps you identify and notify where the simulation is deviating from the expected simulation behavior. You can run the feature on a passing or failing simulation and modify the design based on the diagnostics report. This feature is targeted for Device Under Test (DUT) races.

This feature provides the following diagnostics.

- Clock Diagnostics
 - Done on the clock root (not per flop)
- EOD-CLOCK_NBA : @ 2 : Module: bot1 Instance: top.c
 Line: 150 File: eod.sv Signal: clk
- Data Diagnostics
 - EOD-DATA_NBA : @ 5 : Module: bot Instance: top.b0
 Line: 143 File: eod.sv Signal: d
- Glitches on Trigger Signals
 - Multiple changes in the same NBA delta
 - Triggers of non-combo processes

Can be flagged at a per process level

EOD-EVENT_GLITCH : @ 5 : Instance: top

Line: 58 File: eod.sv Signal(s): b

- \$random Identification
 - Usage of \$random might cause instability

Flop Data Race Rules

The event order diagnostics feature works in accordance with the following flop data race rules:

- Clocks should settle before the first Non-Blocking Assignment (NBA) region.
- Data should change after the first NBA region.

Use Model

You can use the `-event_order_diag` compile-time option to enable event order diagnostics.

The diagnostic reports are dumped in the current working directory in the following files:

- `eventOrderDiag.txt`: Detailed diagnostic report with instance-level information.
- `eventOrderDiag_module.txt`: Diagnostic report unqualified based on module and signal name.

Diagnostic Report Details

The following sample report detects four types of violations. Resolving these violations help to maintain the consistent simulation behavior.

```
+-----+-----+
---+
| EOD | Description |
|-----+-----+
--+
| EOD-CLOCK_NBA | Clock signal changed after NBA region start |
| EOD-DATA_BA | Data signal changed before NBA region start |
| EOD-EVENT_GLITCH | Glitch on the event control of a non-combo process|
| EOD-RANDOM_USAGE | $random usage might cause instability |
+-----+-----+
---+
```

- **EOD-CLOCK_NBA**: According to flop data race rules followed for this diagnostic, clock should settle before the first NBA region. If clock in the test is changing after the first NBA region, this is reported as a violation.
- **EOD-DATA_BA**: According to flop data race rules followed for this diagnostic, data should settle after the first NBA region. If data changes before the first NBA region, this is reported as a violation.
- **EOD-EVENT_GLITCH**: Same-time stamp value changes on the events of non-combo processes are reported as glitches.
- **EOD-RANDOM_USAGE**: \$random generates random stimulus and its usage might result in random and inconsistent behavior. \$random usage in the design is reported to sensitize you.

Limitations

The feature has the following limitations:

- Data and clock violations are reported only at non-zero time. Time zero violations for `CLOCK_NBA` and `DATA_BA` are not reported to avoid reporting multiple messages due to time zero initialization.
- In some scenarios, the feature can report some internal signal name.

8

VPD, VCD, and EVCD Utilities

This chapter describes the following:

- [Advantages of VPD](#)
- [Dumping a VPD File](#)
- [Dump Multi-Dimensional Arrays and Memories](#)
- [Dumping an EVCD File](#)
- [Post-processing Utilities](#)

VCS allows you to save your simulation history in the following formats:

- Value Change Dumping (VCD)

VCD is the IEEE Standard for Verilog designs. You can save your simulation history in VCD format by using the `$dumpvars` Verilog system task.

- VCDPlus Dumping (VPD)

VPD is a Synopsys proprietary dumping technology. VPD has many advantages over the standard VCD ASCII format. See [Advantages of VPD](#) for more information. To dump a VPD file, use the `$vcdpluson` Verilog system task. See [Dumping a VPD File](#) for more information.

- Extended VCD (EVCD)

EVCD dumps only the port information of your design. See [Dumping an EVCD File](#) for more information.

VCS also provides several post-processing utilities to:

- Convert VPD to VCD
- Convert VCD to VPD
- Merge VPD Files

Advantages of VPD

VPD offers the following advantages over the standard VCD ASCII format:

- Provides a compressed binary format that dramatically reduces the file size as compared to VCD and other proprietary file formats.
- The VPD compressed binary format dramatically reduces the signal load time.
- Allows data collection for signals or scopes to be turned on and off during a simulation run, thereby dramatically improving simulation runtime and file size.
- Can save source statement execution data.

To optimize VCS performance and VPD file size, consider the size of the design, the RAM memory capacity of your workstation, swap space, disk storage limits, and the methodology used in the project.

Dumping a VPD File

You can save your simulation history in VPD format in the following ways:

- [Using System Tasks](#) - For Verilog designs.
 - [Using UCLI](#) - For VHDL, Verilog, and mixed designs.
-

Using System Tasks

VCS provides Verilog system tasks to:

- [Enable and Disable Dumping](#)
- [Override the VPD Filename](#)
- [Dump Multi-Dimensional Arrays and Memories](#)
- [Capture Delta Cycle Information](#)

Enable and Disable Dumping

You can use the `$vcdpluson` and `$vcdplusoff` Verilog system tasks to enable and disable the dumping of the simulation history in VPD format.

Note:

The default VPD filename is `vcdplus.vpd`. However, you can use `$vcdplusfile` to override the default filename. For more information, see [Override the VPD Filename](#).

\$vcndlpluson

The following is the syntax of the \$vcndlpluson system task:

```
$vcndlpluson (level|"LVL=integer_variable",scope*,signal*);
```

Usage:

level | "LVL=integer_variable"

Specifies the number of hierarchy scope levels to descend to record signal value changes (a zero value records all scope instances to the end of the hierarchy. The default value is zero).

You can also specify the number of hierarchy scope levels using "LVL=integer_variable". Where, integer_variable specifies the level to descend to record signal value changes.

scope

Specifies the name of the scope in which to record signal value changes (the default is all).

signal

Specifies the name of the signal in which to record signal value changes (the default is all).

Note:

In the syntax, * indicates that the argument can have a list of more than one value (for scopes or signals).

Example 1: Record all signal value changes

```
'timescale 1ns/1ns
module test ();
...
initial
$vcndlpluson;
...
endmodule
```

When you simulate the above example, VCS saves the simulation history of the whole design in vcdplus.vpd. For information on the use model to simulate the design, see “Basic Usage Model” .

Example 2: Record signal value changes for scope test.risc1.alureg and all levels below it

```
'timescale 1ns/1ns
module test ();
```

```

...
risc1 risc(...);

initial
$vcpluspluson(test.risc1.alureg);

...
endmodule

```

When you simulate this example, VCS saves the simulation history of the instance `alureg`, and all instances below `alureg` in `vcplusplus.vpd`.

\$vcplusplusoff

The `$vcplusplusoff` system task stops recording the signal value changes for the specified scopes or signals.

The following is the syntax of the `vcplusplusoff` system task:

```
$vcplusplusoff (level|"LVL=integer",scope*,signal*);
```

Example 1: Turn recording off

```

'timescale 1ns/1ns
module test ();
...
initial
begin
  $vcpluspluson; // Enable Dumping
  #5 $vcplusplusoff; //Disable Dumping after 5ns
  ...
end
...
endmodule

```

This example enables dumping at 0ns and disables dumping after 5ns.

Example 2: Stop recording signal value changes for scope `test.risc1.alu1`.

```

'timescale 1ns/1ns
module test ();
...
initial
begin
  $vcpluspluson; // Enable Dumping
  $vcplusplusoff(test.risc1.alu1); //Does not dump signal value
                                  //changes in test.risc1.alu1
  ...
end
...
endmodule

```

This example enables dumping of the entire design. However, `$vcdplusoff` disables the dumping of the instance `alu1` and instances below `alu1`.

Note:

If multiple `$vcdpluson` commands cause a given signal to be saved, the signal continues to be saved until an equivalent number of `$vcdplusoff` commands are applied to the signal.

Override the VPD Filename

By default, `$vcdpluson` writes the simulation history in the `vcdplus.vpd` file. However, you can override the default filename by using the `$vcdplusfile` system task as follows:

```
$vcdplusfile ("filename.vpd");
$vcdpluson();
```

Note:

You must use `$vcdpluson` after specifying `$vcdplusfile`, as shown above, to override the default filename.

Example:

```
'timescale 1ns/1ns
module test ();
...
initial
begin
    $vcdplusfile("my.vpd"); //Dumps signal value changes
                           //in my.vpd
    $vcdpluson; // Enable Dumping
...
end
...
endmodule
```

The above example writes the signal value changes of the whole design in `my.vpd`.

Dump Multi-Dimensional Arrays and Memories

This section describes system tasks and functions that provide visibility into the multi-dimensional arrays (MDAs).

Following are the two ways to view MDA data:

- The first method, which uses the `$vcdplusmemon` and `$vcdplusmemoff` system tasks, records data each time an MDA has a data change.
- The second method, which uses the `$vcdplusmemorydump` system task, stores data only when the task is called.

Syntax for Specifying MDAs

Use the following syntax to specify MDAs using the `$vcplusmemon`, `$vcplusmemoff`, and `$vcplusmemorydump` system tasks:

```
system_task(Mda);
```

Where,

`system_task`

Name of the system task (required). It can be `$vcplusmemon`, `$vcplusmemoff`, or `$vcplusmemorydump`.

`Mda`

Name of the MDA to be recorded.

Example

This section provides example and graphical representation of MDA and memory declaration using the `$vcplusmemon` system task.

Consider the following example code:

```
module tb();
...
reg [3:0] addr1L, addr1R, addr2L, addr2R, addr3L, addr3R;
reg [7:0] mem01 [1:3] [4:6] [7:9];
...
endmodule
```

In this example, `mem01` is a three-dimensional array. It has 3x3x3 (27) locations; each location is 8 bits in length, as shown in [Figure 115](#).

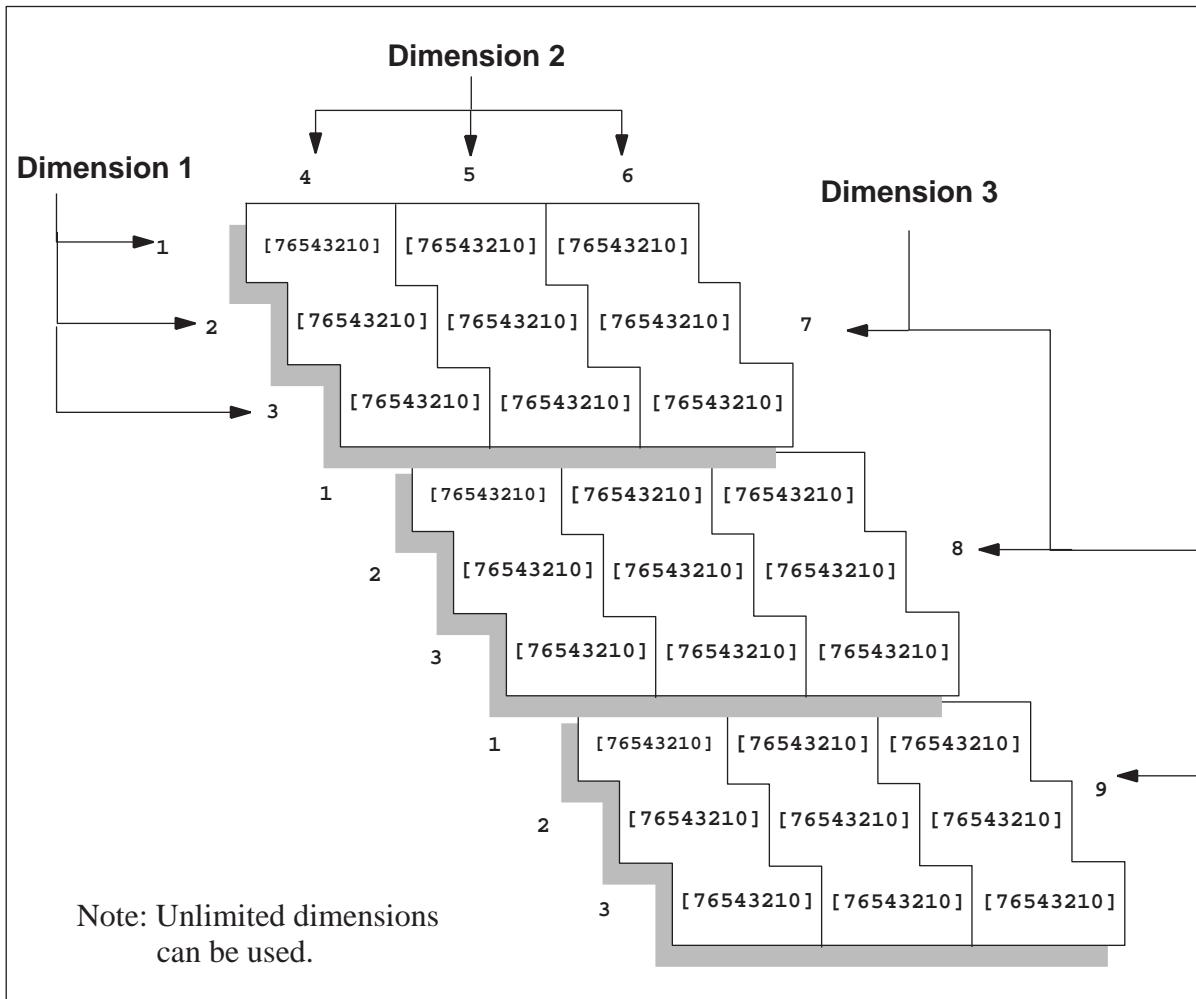
Example: To dump all elements to the VPD File

Consider the following example code:

```
module test();
...
initial
$vcplusmemon( mem01 );
    // Records all elements of mem01 to a VPD file.
...
endmodule
```

In this example, `$vcplusmemon` dumps the entire `mem01` MDA.

Figure 115 reg [7:0] mem01 [1:3] [4:6] [7:9]



Using \$vcdplusmemorydump System Task

The `$vcdplusmemorydump` system task dumps a snapshot of memory locations. When the function is called, the current contents of the specified range of memory locations are recorded (dumped).

You can specify to dump the complete set of multi-dimensional array elements only once. You can specify multiple element subsets of an array using multiple `$vcdplusmemorydump` commands, but they must occur in the same simulation time. In subsequent simulation times, `$vcdplusmemorydump` commands must use the initial set of array elements or a subset of those elements. Dumping elements outside the initial specifications result in a warning message.

Capture Delta Cycle Information

You can use the following VPD system tasks to capture and display delta cycle information in the Wave View

\$vcddplusdeltacycleon

The `$vcddplusdeltacycleon` system task enables reporting of delta cycle information from the Verilog source code. It must be followed by the appropriate `$vcddpluson`/`$vcddplusoff` task.

Glitch detection is automatically turned on when VCS executes `$vcddplusdeltacycleon` unless you have previously used `$vcddplusglitchon/off`. Once you use `$vcddplusglitchon/off`.

Syntax:

```
$vcddplusdeltacycleon;
```

Note:

Delta cycle dumping can start only at the beginning of a time sample. The `$vcddplusdeltacycleon` task must precede the `$vcddpluson` command to ensure that delta cycle collection starts at the beginning of the time sample.

\$vcddplusdeltacycleoff

The `$vcddplusdeltacycleoff` system task turns off reporting of delta cycle information starting at the next sample time.

Glitch detection is automatically turned off when VCS executes `$vcddplusdeltacycleoff` unless you have previously used `$vcddplusglitchon/off`. Once you use `$vcddplusglitchon/off`.

Syntax:

```
$vcddplusdeltacycleoff;
```

Dumping an EVCD File

EVCD dumps the signal value changes and the direction of the ports at the specified module instance. You can dump an EVCD file using the following methods:

- [Using \\$dumpports System Task](#)
- [Analyzing Direction Only for inout Ports](#)
- [Dumping EVCD File for Mixed Designs Using UCLI dump Command](#)
- [Limitations](#)

Using \$dumpports System Task

The `$dumpports` system task creates an EVCD file as specified in IEEE Standard 1364-2001. The EVCD file records the transition times and values of the ports in a module instance. The EVCD file contains more information than the VCD file specified by the `$dumpvars` system task. It includes strength levels and information on whether the test bench or the Device Under Test (DUT) is driving the signal's value.

Syntax:

```
$dumpports(module_instance, [module_instance,] "filename");
```

Example:

```
$dumpports(top.middle1, "dumpports.evcd");
```

Analyzing Direction Only for inout Ports

When you use the `+dumpports+portdir` runtime option, the direction specified in the Verilog module is considered as golden for input and output ports, and the direction is analyzed only for inout ports. For example,

```
% simv +dumpports+portdir
```

Dumping EVCD File for Mixed Designs Using UCLI dump Command

You can either use the `$dumpports` system task or the UCLI `dump` command to dump an Extended Value Change Dump (EVCD) file for mixed designs. However, due to the XMR restriction in VHDL, you may not be able to use `$dumpports` for all mixed design flows.

For pure Verilog design flow, it is recommended to use the `$dumpports` system task to dump an EVCD file, as it does not require any changes at compile time. Also, `$dumpports` allows you to dump multiple EVCD files, which is not possible with UCLI.

For mixed design flows, it is recommended to use the UCLI `dump` command along with the configuration file to dump the EVCD file, as described in the following use model.

Use Model

To dump an EVCD file for mixed design flows, it is recommended to use the configuration file with the `+optconfigfile` compile-time option to specify all the instances for which the UCLI `dump` command may be used to dump EVCD.

```
% vcs +optconfigfile+file_name.cfg -debug_access+r+cbk+drivers
file_name.v
```

Where, `file_name.cfg` is the configuration file which allows you to specify the instances that needs to be dumped at compile time. Following is the syntax of the configuration file:

```
instance {list_of_instance_hierarchical_names} {enable_evcd};
```

For example,

```
instance {top.dut} {enable_evcd};
```

Note:

- The configuration file only enables EVCD dumping, it does not dump EVCD. To dump an EVCD file, you must use the UCLI `dump` command at runtime, as follows:

```
ucli% dump -file test.evcd -type EVCD
ucli% dump -add {top.dut}
```

- If the configuration file is not specified, then a warning message is issued for the cases where ports are connected to the bidirectional switches, and EVCD results may not be accurate.
- All forces are considered as TB regardless of where the force is applied from (TB, DUT, or UCLI).

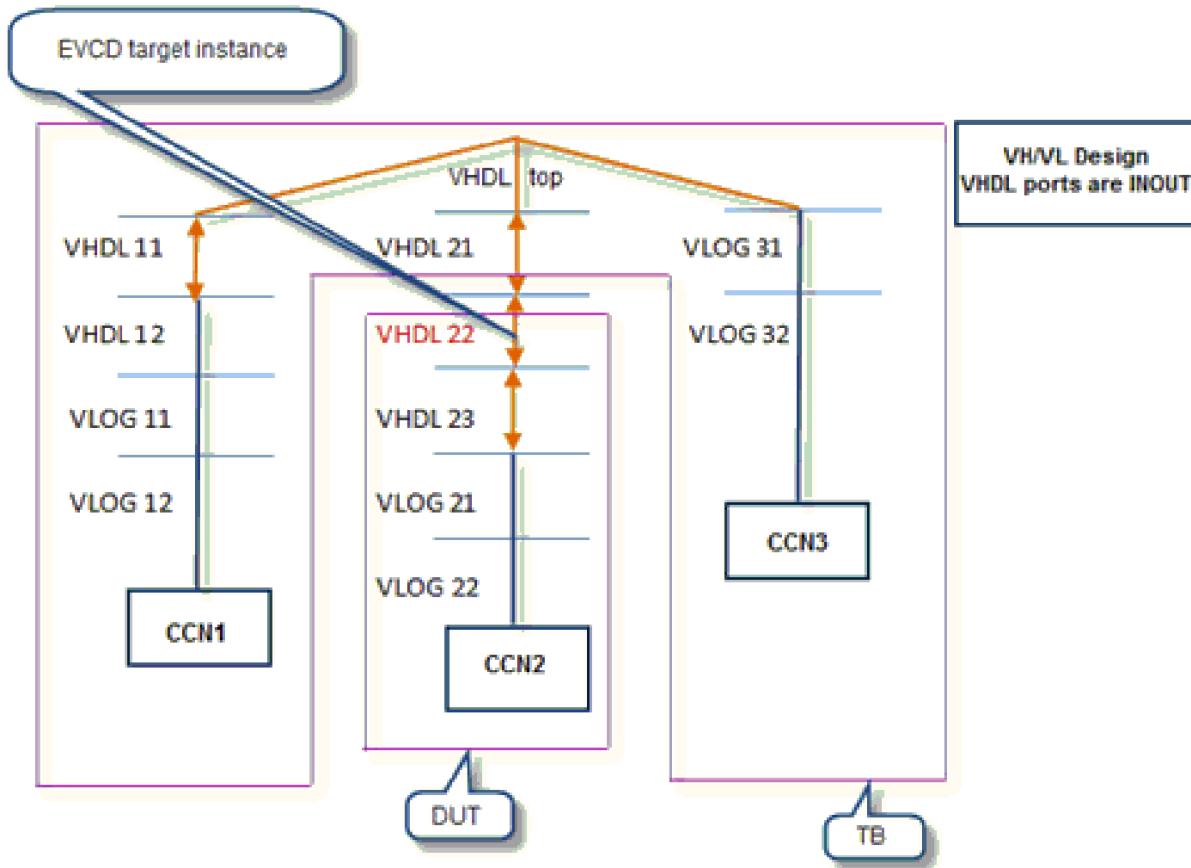
Use Model for Dumping CCN Driver Through INOUT

EVCD file contains the CCN driver when the CCN is connected through INPUT or OUTPUT ports in Verilog-VHDL or VHDL-Verilog mixed designs. However, if a target VHDL instance lies inside a VHDL connected through INOUT ports, you must use the `+dumpports+mxccn` option at compile time to dump CCN drivers.

```
% vhdlan <design files>
% vlogan <design file>
% vcs +optconfigfile+file_name.cfg -debug_access+r+cbk+drivers
  top_module +dumpports+mxccn
```

Following is a sample VHDL-Verilog design which requires `+dumpports+mxccn`:

Both testbench and DUT have VHDL and the design is mixed (VHDL and Verilog). The Verilog design has CCN which is connected to VHDL through the INOUT ports.



Limitations

Following are the limitations of the EVCD dumping using `$dumpports` or UCLI command `dump -type EVCD`:

Unsupported Port Types

- For Verilog DUT:
 - Ports can only be of type Verilog-2001. SystemVerilog type ports are not allowed. VCS generates a warning message, if it finds any unsupported port type.
 - SystemVerilog complex types (including MDAs, dynamic arrays, associative arrays, queues, and so on) are not supported, and not legal in LRM. Interface or virtual interface is not supported.

- For VHDL DUT:
 - Ports can only be of type STD_LOGIC, STD_ULOGIC, STD_LOGIC_VECTOR, STD_ULOGIC_VECTOR, BIT, BIT_VECTOR, BOOLEAN. Any user-defined type or subtype of the above types is supported.
 - Complex types like aggregates, MDA, or enums are not allowed as port or port drivers. A warning message is generated if such constructs are found.
 - Ports having type with user-defined resolution functions in VHDL are not supported.

Unsupported DUT Types

- DUT cannot be SV program, interface, SystemC, Spice, or Verilog-AMS.

SystemC Support

- Each SystemC module is treated like a Verilog shell, and multiple drivers cannot be detected inside SystemC.
- SystemC is not supported as a DUT.

Post-processing Utilities

VCS provides you with the following utilities to process VCD and VPD files. You can use these utilities to perform the following conversions:

- VPD file to a VCD file
- VCD file to a VPD file
- Merge a VPD file

Note:

All utilities are available in \$VCS_HOME/bin.

This section describes these utilities in the following sections:

- [The vcdpost Utility](#)
- [The vcdiff Utility](#)
- [The vcat Utility](#)
- [The vcsplit Utility](#)
- [The vcd2vpd Utility](#)
- [The vpd2vcd Utility](#)

- [The vpdmerge Utility](#)
 - [The vpduutil Utility](#)
-

The vcdpost Utility

You can use the `vcdpost` utility to generate an alternative VCD file that has the following characteristics:

- Contains value change and transition times for each bit of a vector net or register, recorded as a separate signal. This is called “scalarizing” the vector signals in the VCD file.
- Avoids sharing the same VCD identifier code with more than one net or register. This is called “uniquifying” the identifier codes.

Scalarizing the Vector Signals

The VCD format does not support a mechanism to dump part of a vector. For this reason, if you specify a bit select or a part select for a net or register as an argument to the `$dumpvars` system task, VCS records value changes and transition times for the entire net or register in the VCD file. For example, if you specify the following in your source code:

```
$dumpvars(1,mid1.out1[0]);
```

Where, `mid1.out1[0]` is a bit select of a signal (because you need to examine the transition times and value changes of this bit). VCS however writes a VCD file that contains the following:

```
$var wire 8 ! out1 [7:0] $end
```

Therefore, all the value changes and simulation times for the signal `out1` are for the entire signal, not just for `0` bit.

The `vcdpost` utility can create an alternative VCD file that defines a separate `$var` section for each bit of the vector signal. The results are as follows:

```
$var wire 8 ! out1 [7] $end
$var wire 8 " out1 [6] $end
$var wire 8 # out1 [5] $end
$var wire 8 $ out1 [4] $end
$var wire 8 % out1 [3] $end
$var wire 8 & out1 [2] $end
$var wire 8 ' out1 [1] $end
$var wire 8 ( out1 [0] $end
```

What this means is that the new VCD file contains value changes and simulation times for each bit.

Uniquifying the Identifier Codes

In certain circumstances, to enable better performance, VCS assigns the same VCD file identifier code to more than one net or register, if these nets or registers have the same value throughout the simulation. Consider the following example:

```
$var wire 1 ! ramsel_0_0 $end
$var wire 1 ! ramsel_0_1 $end
$var wire 1 ! ramsel_1_0 $end
$var wire 1 ! ramsel_1_1 $end
```

In this example, VCS assigns ! identifier code to more than one net.

Some back-end tools from other vendors fail when you input such a VCD file. You can use the `vcdpost` utility to create an alternative VCD file in which the identifier codes for all nets and registers, including those instances without value changes, are unique. Following is the example:

```
$var wire 1 ! ramsel_0_0 $end
$var wire 1 " ramsel_0_1 $end
$var wire 1 # ramsel_1_0 $end
$var wire 1 $ ramsel_1_1 $end
```

The `vcdpost` Utility Syntax

The syntax for the `vcdpost` utility is as follows:

```
vcdpost [+scalar] [+unique] input_VCD_file output_VCD_file
```

Where,

`+scalar`

Specifies creating separate `$var` sections for each bit in a vector signal. This option is the default option. You include it on the command line when you also include the `+unique` option and want to create a VCD file that scalarizes the vector nets and uniquifies the identifier codes.

`+unique`

Specifies uniquifying the identifier codes. When you include this option without the `+scalar` option, `vcdpost` uniquifies the identifier codes without scalarizing the vector signals.

`input_VCD_file`

The name of the VCD file created by VCS.

`output_VCD_file`

The name of the alternative VCD file created by the `vcdpost` utility.

The vcdiff Utility

The `vcdiff` utility compares two dump files and reports any differences it finds. The dump file can be of type VCD, EVCD, or VPD.

Note:

The `vcdiff` utility cannot compare dump files of different types.

Dump files consist of two sections:

- A header section that reflects the hierarchy (or some subset) of the design that was used to create the dump file.
- A value change section, which contains all of the value changes (and times when those value changes occurred) for all of the signals referenced in the header.

The `vcdiff` utility first compares the header sections and reports any signals/scopes that are present in one dump file, but absent in other. Then, the `vcdiff` utility compares the value change sections of the dump files for signals that appear in both dump files. This utility determines value change differences based on the final value of the signal in a time step.

Syntax

The syntax of the `vcdiff` utility is as follows:

```
vcdiff first_dump_file second_dump_file
[-noabsentsig] [-absentsigs scope] [-absentsigiserror]
[-allabsentsig] [-absentfile filename] [-matchtypes] [-ignorecase]
[-min time] [-max time] [-scope instance] [-level level_number]
[-include filename] [-ignore filename] [-strobe time1 time2]
[-prestroke] [-synch signal] [-synch0 signal] [-synch1 signal]
[-when expression] [-xzmatch] [-noxzmatchat0]
[-compare01xz] [-xumatch] [-xdmatch] [-zdmatch] [-zwmatch]
[-showmasters] [-allsigdiffs] [-wrapsize size]
[-limitdiffs number] [-ignorewires] [-ignorereg] [ingoreals]
[-ignorefunctaskvars] [-ignoreretiming units] [-ignorestrength]
[-geninclude [filename]] [-spikes]
```

Options for Specifying Scope/Signal Hierarchy

The following options control how the `vcdiff` utility compares the header sections of the dump files:

`-noabsentsig`

Does not report any signals that are present in one dump file, but are absent in other.

`-absentsigs scope`

Reports only absent signals in the given scope.

`-absentfile [file]`

Prints the full path names of all absent scopes/signals to the given file, as opposed to stdout.

`-absentsigiserror`

If this option is present, and there are any absent signals in either dump file, `vcdiff` returns an error status upon completion even if it does not detect any value change differences. If this option is not present, absent signals do not cause an error.

`-allabsentsig`

Reports all absent signals. If this option is not present, by default, `vcdiff` reports only the first 10 absent signals.

`-ignorecase`

Ignores the case of scope/signal names when looking for absent signals. In effect, it converts all signal/scope names to uppercase before comparison.

`-matchtypes`

Reports mismatches in signal data types between the two dump files.

Options for Specifying Scope(s) for Value Change Comparison

By default, `vcdiff` compares the value changes for all signals that appear in both dump files. The following options limit value change comparisons to specific scopes.

`-scope [scope]`

Changes the top-level scope to be value change compared from the top of the design to the indicated scope. All child scopes/signals of the indicated scope are compared unless modified by the `-level` option (below).

`-level N`

Limits the depth of scope for which value change comparison occurs. For example, if `-level 1` is the only command-line option, then `vcdiff` compares the value changes of only the signals in the top-level scope in the dump file.

`-include [file]`

Reports value change compares only for those signals/scopes given in the specified file. The file contains a set of full path specifications of signals and/or scopes, one per line.

`-ignore [file]`

Removes any signals/scopes contained in the given file from value change comparison. The file contains a set of full path specifications of signals and/or scopes, one per line.

Note:

The `vcdiff` utility applies the `-scope/-level` options first. It then applies the `-include` option to the remaining scopes/signals, and finally applies the `-ignore` option.

Options for Specifying When to Perform Value Change Comparison

The following options limit when `vcdiff` detects value change differences:

`-min time`

Specifies the starting time (in simulation units) when value change comparison is to begin (default time is 0).

`-max time`

Specifies the stopping time (in simulation units) when value change comparison ends. By default, this occurs at the latest time found in either dump file.

`-strobe first_time delta_time`

Only checks for differences when the `strobe` is true. The strobe is true at `first_time` (in simulation units) and then every `delta_time` increment thereafter.

`-prestrobe`

Used in conjunction with `-strobe`, tells `vcdiff` to look for differences just before the strobe is true.

`-when expression`

Reports differences only when the given `when expression` is true. Initially this expression can consist only of scalar signals, combined with `and`, `or`, `xor`, `xnor`, `and not` operators, and employ parentheses to group these expressions. You must fully specify the complete path (from root) for all the signals used in expressions.

Operators may be either Verilog style (`&`, `|`, `^`, `~^`, `~`) or VHDL (`and`, `or`, `xor`, `xnor`, `not`).

`-synch signal`

Checks for differences only when the given signal changes value. In effect, the given signal is a “clock” for value change comparison, where the differences are checked only on the transitions (any) of this signal.

`-synch0 signal`

As `-synch` (above) except that it checks for differences when the given signal transitions to ‘0’.

`-synch1`

As `-synch` (above) except that it checks for differences only when the given signal transitions to '1'.

Note:

The `-max`, `-min` and `-when` options must be true in order for `vcdiff` to report value change difference.

Options for Filtering Differences

The following options filter out value change differences that are detected under certain circumstances. For the most part, these options are additive.

`-ignoretiming time`

Ignores the value change when the same signal in one of the VCD files has a different value from the same signal in the other VCD file for less than the specified time. This is to filter out signals that have only slightly different transition times in the two VCD files. The `vcdiff` utility reports a change when there is a transition to a different value in one of the VCD files, and then a transition back to a matching value in that same file.

`-ignorereg`

Does not report value change differences on signals that are of type register.

`-ignorewires`

Does not report value change differences on signals that are of type wire.

`-ignorereal`

Does not report value change differences on signals that are of type real.

`-ignorefunctaskvars`

Does not report value change differences on signals that are function or task variables.

`-ignorestrength (EVCD only)`

EVCD files contain a richer set of signal strength and directional information than VCD or even VPD files. This option ignores the strength portion of a signal value when checking for differences.

`-compare01xz (EVCD only)`

Converts all signal state information to equivalent 4-state values (0, 1, x, z) before comparison is made (EVCD files only). Also, ignores the strength information.

`-xzmatch`

Equates x and z values.

`-xumatch (9-state VPD file only)`

Equates `x` and `u` (uninitialized) values.

`-xdmatchv`(9-state VPD file only)

Equates `x` and `d` (dontcare) values.

`-zdmatch` (9-state VPD file only)

Equates `z` and `d` (dontcare) values.

`-zwmatch` (9-state VPD file only)

Equates `z` and `w` (weak 1) values. In conjunction with `-xzmatch` (above), this option causes `x` and `z` value to be equated at all times EXCEPT time 0.

Options for Specifying Output Format

The following options change how value change differences are reported.

`-allsigdiffs`

By default, `vcdiff` only shows the first difference for a given signal. This option reports all differences for a signal until the maximum number of differences are reported (see `-limitdiffs`).

`-wrapsize columns`

Wraps the output of vectors longer than the given size to the next line. The default value is 64.

`-showmasters` (VCD, EVCD files only)

Shows collapsed net masters. VCS can split a collapsed net into several sub-nets when this has a performance benefit. This option reports the master signals (first signal defined on a net) when they are different in the two dump files.

`-limitdiffs number_of_diffs`

By default, `vcdiff` stops after the first 50 differences are reported. This option overrides this default behavior. Setting this value to 0 causes `vcdiff` to report all differences.

`-geninclude filename`

Produces a separate file of the given name in addition to the standard `vcdiff` output. This file contains the list of signals that have at least one value change difference. The format of the file is one signal per line. Each signal name is a full path name. You can use this file as an input to the `vcat` tool using the `-include` option of `vcat`.

`-spikes`

A spike is defined as a signal that changes multiple times in a single time step. This option annotates (with #) the value change differences detected when the signal spikes (glitches). It keeps and reports a total count of such differences.

The vcdiff Utility Output Example

The following is an example of the `vcdiff` output:

```
--- top.sig1 --- 200 ---
< 200 0
---
> 100 1

--- top.sig2 --- 200 ---
< 100 1
---
> 200 0
```

In this example, there are two differences between the two compared dump files. The format of a difference is as follows:

```
--- signal_hierarchical_name --- time_of_mismatch ---
< time_of_last_change change_to_this_value
---
> time_of_last_change change_to_this_value
```

Where:

< (line beginning with <)

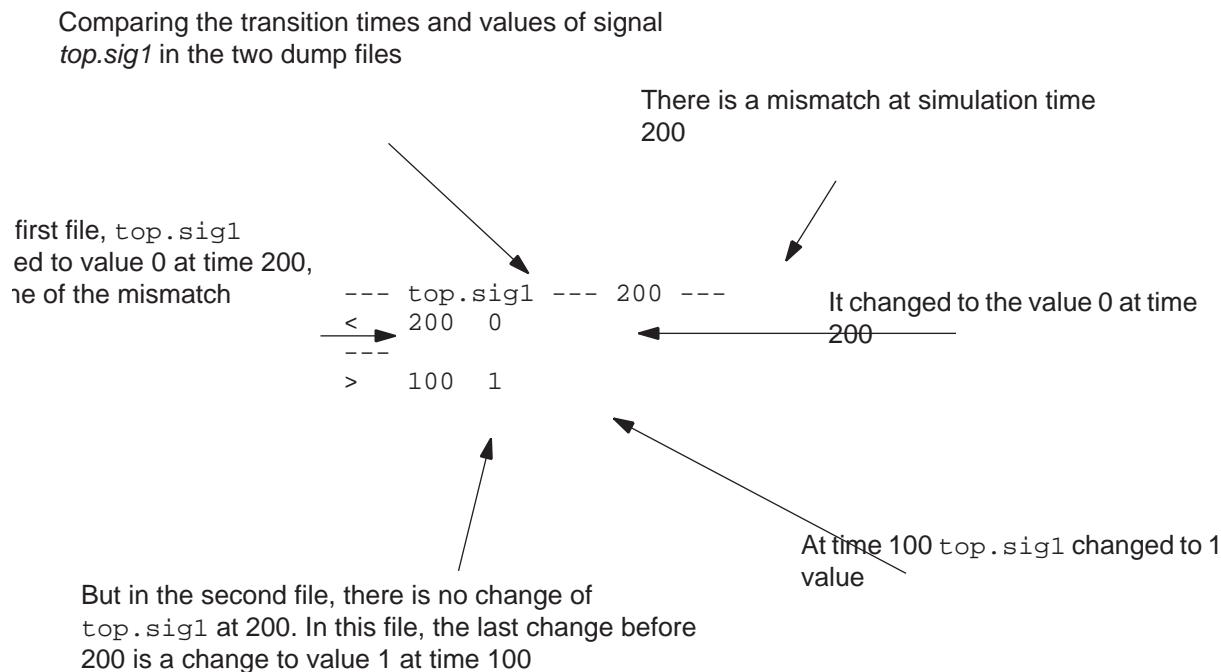
Contains the time of the last value change of the signal (at or before the time of the mismatch) in the first dump file on the `vcdiff` command line.

> (line beginning with >)

Contains the corresponding information in the second dump file on the `vcdiff` command line.

[Figure 116](#) shows the Callout notes on the different names and values in an example difference.

Figure 116 An Annotated Difference Example



You can infer from this example that signal `top.sig1`, in both the first and second dump file, transitioned to 1 at time 100 because there is no mismatch at time 100.

The vcat Utility

The format of a VCD or a EVCD file, although a text file, is written to be read by software and not by human designers. VCS includes the `vcat` utility to enable you to more easily understand the information contained in a VCD file.

The vcat Utility Syntax

The `vcat` utility has the following syntax:

```
vcat VCD_filename [-deltaTime] [-raw] [-min time] [-max time]
[-scope instance_name] [-level level_number]
[-include filename] [-ignore filename] [-spikes] [-noalpha]
```

```
[-wrapsize size] [-showmasters] [-showdefs] [-showcodes]
[-stdin] [-vgen]
```

Where,

`-deltaTime`

Specifies writing simulation times as the interval since the last value change rather than the absolute simulation time of the signal transition. Without `-deltaTime`, `vcat` output looks as follows:

```
--- TEST_top.TEST.U4._G002 ---
0      x
33     0
20000  1
30000  x
30030  z
50030  x
50033  1
60000  0
70000  x
70030  z
```

With `-deltaTime`, `vcat` output looks as follows:

```
--- TEST_top.TEST.U4._G002 ---
0      x
33     0
19967  1
10000  x
30     z
20000  x
3      1
9967   0
10000  x
30     z
```

`-raw`

Displays “raw” value changed data organized by the simulation time rather than signal name.

`-min time`

Specifies the start simulation time from which `vcat` begins to display data.

`-max time`

Specifies an end simulation time up to which `vcat` displays data.

`-scope instance_name`

Specifies a module instance. The `vcat` utility displays data for all signals of an instance and all signals hierarchically under it.

`-level level_number`

Specifies the number of hierarchical levels for which `vcat` displays data. The starting point is either the top-level module or the module instance you specify with the `-scope` option.

`-include filename`

Specifies a file that contains a list of module instances and signals. The `vcat` utility only displays data for these signals or the signals in these module instances.

`-ignore filename`

Specifies a file that contains a list of module instances and signals. However, the `vcat` utility does NOT display data for these signals or the signals in these module instances.

`-spikes`

Indicates all zero-time transitions with the `>>` symbol in the leftmost column. In addition, prints a summary of the total number of spikes seen at the end of the `vcat` output. The following is an example of the new output:

```
--- DF_test.logic.I_348.N_1 ---
0      x
100    0
120    1
>>120  0
4000   1
12000  0
20000  1

Spikes detected:  5
```

`-noalpha`

By default, `vcat` displays signals within a module instance in alphabetical order. This option disables this ordering.

`-wrapsize size`

Specifies value displays for wide vector signals, how many bits to display on a line before wrapping to the next line.

`-showmasters`

Specifies showing collapsed net masters.

`-showdefs`

Specifies displaying signals, but not their value changes or the simulation time of these value changes.

`-showcodes`

Specifies displaying the signal's VCD file identifier code.

`-stdin`

Enables you to use standard input, such as piping the VCD file into `vcat`, instead of specifying the filename.

`-vgen`

Generates (from a VCD file) two types of source files for a module instance: one that models how the design applies stimulus to the instance, and the other that models how the instance applies stimulus to the rest of the design. See [Generating Source Files From VCD Files](#).

The following is an example of the output from the `vcat` utility:

```
vcat exp1.vcd

exp1.vcd: scopes:6 signals:12 value-changes:13
--- top.mid1.in1 ---
0 1

--- top.mid1.in2 ---
0 xxxxxxxx
10000 00000000

--- top.mid1.midr1 ---
0 x
2000 1

--- top.mid1.midr2 ---
0 x
2000 1
```

In this output, for example, you see that signal `top.mid1.midr1` at time 0 had a value of `x`, and at simulation time 2000 (as specified by the `$timescale` section of the VCD file, which VCS derives from the time precision argument of the `'timescale` compiler directive) this signal transitioned to 1.

Generating Source Files From VCD Files

The `vcat` utility can generate Verilog and VHDL source files that are one of the following:

- A module definition that succinctly models how a module instance is driven by a design, that is, a concise testbench module that instantiates the specified instance and applies stimulus to that instance the way the entire design does. This is called testbench generation.
- A module definition that mimics the behavior of the specified instance to the rest of the design, that is, it has the same output ports as the instance and in this module definition the values from the VCD file are directly assigned to these output ports. This is called module generation.

Note:

The `vcat` utility can only generate these source files for instances of module definitions that do not have inout ports.

Testbench generation enables you to focus on a module instance, applying the same stimulus as the design does, but at faster simulation because the testbench is far more concise than the entire design. You can substitute module definitions at different levels of abstraction and use `vcldiff` to compare the results.

Module generation enables you to use much faster simulating “canned” modules for a part of the design to enable the faster simulation of other parts of the design that need investigation.

The name of the generated source file from testbench generation begins with `testbench` followed by the module and instance names in the hierarchical name of the module instance, separated by underscores. For example `testbench_top_ad1.v`.

Similarly, the name of the generated source file from module generation begins with `moduleGeneration` followed by the module and instance names in the hierarchical name of the module instance, separated by underscores. For example `moduleGeneration_top_ad1.v`.

You enable `vcat` to generate these files by doing the following:

1. Writing a configuration file.
2. Running `vcat` with the `-vgen` command-line option.

Writing the Configuration File

The configuration file is named `vgen.cfg` by default, and `vcat` looks for it in the current directory. This file needs three types of information specified in the following order:

1. The hierarchical name of the module instance.
2. Specification of testbench generation with the keyword `testbench` or specification of module generation with the keyword `moduleGeneration`.
3. The module header and the port declarations from the module definition of the module instance.

You can use Verilog comments in the configuration file.

The following is an example of a configuration file:

Example 43 Configuration File

```
top.ad1
testbench
//moduleGeneration
module adder (out,in1,in2);
input in1,in2;
output [1:0] out;
```

You can use a different name and location for the configuration file. In order to do this, you must enter it as an argument to the `-vgen` option. For example:

```
vcat filename.vcd -vgen /u/design1/vgen2.cfg
```

Example 44 Source Code

Consider the following source code:

```
module top;
reg r1,r2;
wire int1,int2;
wire [1:0] result;

initial
begin
$dumpfile("exp3.vcd");
$dumpvars(0,top.pa1,top.ad1);
#0 r1=0;
#10 r2=0;
#10 r1=1;
#10 r2=1;
#10 r1=0;
#10 r2=0;
#10 r1=1;
#10 r2=1;
#10 r1=0;
```

```

#10 r2=0;
#10 r1=1;
#10 r2=1;
#10 r1=0;
#10 r2=0;
#100 $finish;
end

passer pa1 (int1,int2,r1,r2);
adder ad1 (result,int1,int2);
endmodule

module passer (out1,out2,in1,in2);
input in1,in2;
output out1,out2;

assign out1=in1;
assign out2=in2;
endmodule

module adder (out,in1,in2);
input in1,in2;
output [1:0] out;

reg r1,r2;
reg [1:0] sum;

always @ (in1 or in2)
begin
r1=in1;
r2=in2;
sum=r1+r2;
end

assign out=sum;
endmodule

```

Notice that the stimulus from the testbench module named `test` propagates through an instance of a module named `passer` before it propagates to an instance of a module named `adder`. The `vcat` utility can generate a testbench module to stimulate the instance of `adder` in the same exact way, but in a more concise and therefore faster simulating module.

If you use the sample `vgen.cfg` configuration file in [Example 43](#) and enter the following command line:

```
vcat filename.vcd -vgen
```

The generated source file, `testbench_top_ad1.v`, is as follows:

```

module tbench_adder ;
wire [1:0] out ;

```

```

reg in2 ;
reg in1 ;
initial #131 $finish;
initial $dumpvars;
initial begin
    #0 in2 = 1'bx;
    #10 in2 = 1'b0;
    #20 in2 = 1'b1;
    #20 in2 = 1'b0;
    #20 in2 = 1'b1;
    #20 in2 = 1'b0;
    #20 in2 = 1'b1;
    #20 in2 = 1'b0;
end
initial begin
    in1 = 1'b0;
    forever #20 in1 = ~in1 ;
end
adder ad1 (out,in1,in2);
endmodule

```

This source file uses significantly less code to apply the same stimulus with the instance of module `passer` omitted.

If you revise the `vgen.cfg` file to have `vcat` perform module generation, the generated source file, `moduleGeneration_top_ad1.v`, is as follows:

```

module adder (out,in1,in2) ;
input in2 ;
input in1 ;
output [1:0] out ;
reg [1:0] out ;
initial begin
    #0 out = 2'bxx;
    #10 out = 2'b00;
    #10 out = 2'b01;
    #10 out = 2'b10;
    #10 out = 2'b01;
    #10 out = 2'b00;
    #10 out = 2'b01;
    #10 out = 2'b10;
    #10 out = 2'b01;
    #10 out = 2'b00;
    #10 out = 2'b01;
    #10 out = 2'b10;
    #10 out = 2'b01;
    #10 out = 2'b00;
end
endmodule

```

Notice that the input ports are stubbed, and the values from the VCD files are assigned directly to the output port.

The vcsplit Utility

The `vcsplit` utility generates a VCD, EVCD, or VPD file that contains a selected subset of value changes found in a given input VCD, EVCD, or VPD file (the output file has the same type as the input file). You can select the scopes/signals to be included in the generated file either via a command-line argument, or a separate "include" file.

The vcsplit Utility Syntax

Following is the syntax of the `vcsplit` utility:

```
vcsplit [-ooutput_file] [-scopeselected_scope_or_signal]
[-includeinclude_file] [-min min_time] [-max max_time]
[-level n] [-ignoreignore_file] input_file[-v] [-h]
```

Here:

`-o output_file`

Specifies the name of the new VCD/EVCD/VPD file to be generated. If `output_file` is not specified, `vcsplit` creates the file with the default name `vcsplit.vcd`.

`-scope selected_scope_or_signal`

Specifies a signal or scope whose value changes are to be included in the output file. If a scope name is given, then all signals and sub-scopes in that scope are included.

`-include include_file`

Specifies the name of an include file that contains a list of signals/scopes whose value changes are to be included in the output file.

The include file must contain one scope or signal per line. Each presented scope/signal must be found in the input VCD, EVCD, or VPD file. If the file contains a scope, and separately, also contains a signal in that scope, `vcsplit` includes all the signals in that scope, and issues a warning.

Note:

If you use both `-include` and `-scope` options, `vcsplit` uses all the signals and scopes indicated.

`input_file`

Specifies the VCD, EVCD, or VPD file to be used as input.

If the input file is either VCD or EVCD, and it is not specified, `vcsplit` takes its input from `stdin`. The `vcsplit` utility has this `stdin` option for VCD and EVCD files so that you can pipe the output of gunzip to this tool. If you try to pipe a VPD file through `stdin`, `vcsplit` exits with an error message.

`-min min_time`

Specifies the time to begin the scan.

`-max max_time`

Specifies the time to stop the scan.

`-ignore ignore_file`

Specifies the name of the file that contains a list of signals/scopes whose value changes are to be ignored in the output file.

If you specify neither `include_file` nor `selected_scope_or_signal`, then `vcsplit` includes all the value changes in the output file except the signals/scopes in the `ignore_file`.

If you specify an `include_file` and/or a `selected_scope_or_signal`, `vcsplit` includes all value changes of those signals/scopes that are present in the `include_file` and the `selected_scope_or_signal`, but absent in `ignore_file` in the output file. If the `ignore_file` contains a scope, `vcsplit` ignores all the signals and the scopes in this scope.

`-level n`

Reports only `n` levels hierarchy from top or scope. If you specify neither `include_file` nor `selected_scope_or_signal`, `vcsplit` computes `n` from the top level of the design. Otherwise, it computes `n` from the highest scope included.

`-v`

Displays the current version message.

`-h`

Displays a help message explaining usage of the `vcsplit` utility.

In general, any command-line error (such as illegal arguments) that VCS detects causes `vcsplit` to issue an error message and exit with an error status. Specifically:

If there are any errors in the `-scope` argument or in the include file (such as a listing a signal or scope name that does not exist in the input file), VCS issues an error message, and `vcsplit` exits with an error status.

If VCS detects an error while parsing the input file, it reports an error, and `vcsplit` exits with an error status.

If you do not provide either a `-scope`, `-include` or `-ignore` option, VCS issues an error message, and `vcsplit` exits with an error status.

Limitations

- MDAs are not supported.
- Bit/part selection for a variable is not supported. If this usage is detected, the vector is regarded as all bits are specified.

The vcd2vpd Utility

The `vcd2vpd` utility converts a VCD file generated using `$dumpvars` or UCLI dump commands to a VPD file.

Following is the syntax of the `vcd2vpd` utility:

```
vcd2vpd [-bmin_buffer_size] [-fmax_output_filesize] [-h]
[-m] [-q] [+glitchon] [+nocompress] [+nocurrentvalue]
 [+bitrangenospace] [+vpdnoreadopt] [+dut+dut_sufix] [+tf+tf_sufix]
 vcd_file vpd_file
```

Usage:

`-b<min_buffer_size>`

Minimum buffer size in KB used to store Value Change Data (VCD) before writing it to disk.

`-f<max_output_filesize>`

Maximum output file size in KB. Wrap around occurs if the specified file size is reached.

`-h`

Translate hierarchy information only.

`-m`

Give translation metrics during translation.

`-q`

Suppress printing of copyright and other informational messages.

`+deltacycle`

Add delta cycle information to each signal value change.

`+glitchon`

Add glitch event detection data.

`+nocompress`

Turn data compression off.

+nocurrentvalue

Do not include object's current value at the beginning of each VCD.

+bitrangenospace

Support non-standard VCD files that do not have white space between a variable identifier and its bit range.

+vpdnoreadopt

Turn off read optimization format.

Options for Specifying EVCD Options

+dut+dut_sufix

Modifies the string identifier for the Device Under Test (DUT) half of the split signal. The default value is _DUT.

+tf+tf_sufix

Modifies the string identifier for the Test-Fixture half of the split signal. The default value is _TF.

+indexlast

Appends the bit index of a vector bit as the last element of the name.

vcf_file

Specify the vcd filename or use "-" to indicate VCD data to be read from stdin.

vpd_file

Specify the VPD file name. You can also specify the path and the filename of the VPD file, otherwise, the VPD file is generated with the specified name in the current working directory.

The vpd2vcd Utility

The `vpd2vcd` utility converts a VPD file generated using the system task `$vcfpluson` or UCLI dump commands to a VCD or EVCD file.

The syntax is as shown below:

```
vpd2vcd [-h] [-q] [-s] [-x] [-xlrn] [+zerodelayglitchfilter] [+morevhdl]
 [+start+value] [+end+value] [+splitpacked] [-f cmd_filename]
 [-expand_dumpport_scope] vpd_file vcd_file
```

Here:

-h

Translate hierarchy information only.

-q

Suppress the copyright and other informational messages.

-s

Allow sign extension for vectors. Reduces the file size of the generated `vcf_file`.

-x

Expand vector variables to full length when displaying `$dumpoff` value blocks.

-x1rm

Convert uppercase VHDL objects to lowercase.

+zerodelayglitchfilter

Zero delay glitch filtering for multiple value changes within the same time unit.

+morevhdl

Translates the VHDL types of both directly mappable and those that are not directly mappable to Verilog types.

This option may create a non-standard VCD file.

+start+time

Translate the value changes starting after the specified start time.

+end+time

Translate the value changes ending before the specified end time.

Specify both start time and end time to translate the value changes occurring between start and end time.

-f cmd_filename

Specify a command file containing commands to limit the design converted to VCD or EVCD. See the [The Command File Syntax](#) section for more information.

-expand_dumpport_scope

Generates a VCD file with `$scope` information in a hierarchical format, versus the default flattened format. Printing in a hierarchical format with this option matches the `$scope` print format generated through an EVCD file dumped from UCLI.

```
+splitpacked
```

Use this option to change the way packed structs and arrays are reported in the output VCD file. It does the following:

- Treats a packed structure the same as an unpacked structure and dumps the value changes of each field.

Consider the following example:

```
typedef logic [1:0] t_vec;

typedef struct packed {
    t_vec f_vec_b;
} t_ps_b;

module test();
    t_ps_b var_ps_b;
endmodule
```

The VCD file created in the previous example is as follows:

```
$scope module test $end
$scope fork var_ps_b $end
$var reg 2 ! f_vec_b [1:0] $end
$upscope $end
$upscope $end
```

- Treats a packed MDA as an unpacked MDA except for the inner most dimensions.

Consider the following example:

```
typedef logic [1:0] t_vec;

module test();
    t_vec [3:2] var_vec;
endmodule
```

The VCD file created in the previous example is as follows:

```
$scope module test $end
$var reg      2 %     var_vec[3] [1:0] $end
$var reg      2 &     var_vec[2] [1:0] $end
$upscope $end
```

- Expands all packed arrays defined in a packed struct.

Consider the following example:

```
typedef logic [1:0] t_vec;

typedef struct packed {
    t_vec f_vec;
    t_vec [3:2][1:0] f_vec_array;
```

```

} t_ps;

module test();
    t_ps var_ps;
endmodule

```

The VCD file created in the previous example is as follows:

```

$scope module test $end
$scope fork var_ps $end
$var reg      2 '   f_vec [1:0] $end
$var reg      2 (   f_vec_array[3][1] [1:0] $end
$var reg      2 )   f_vec_array[3][0] [1:0] $end
$var reg      2 *   f_vec_array[2][1] [1:0] $end
$var reg      2 +   f_vec_array[2][0] [1:0] $end
$upscope $end
$upscope $end

```

- Expands all dimensions of a packed array defined in a packed struct.

Consider the following example:

```

typedef logic [1:0] t_vec;

typedef struct packed {
    t_vec f_vec;
    t_vec [3:2][1:0] f_vec_array;
} t_ps;

module test();
    t_ps [1:0] var_paps;
endmodule

```

The VCD file created in the previous example is as follows:

```

$scope module test $end
$scope fork var_paps[1] $end
$var reg      2 '   f_vec [1:0] $end
$var reg      2 (   f_vec_array[3][1] [1:0] $end
$var reg      2 )   f_vec_array[3][0] [1:0] $end
$var reg      2 *   f_vec_array[2][1] [1:0] $end
$var reg      2 +   f_vec_array[2][0] [1:0] $end
$upscope $end
$scope fork var_paps[0] $end
$var reg      2 ,   f_vec [1:0] $end
$var reg      2 -   f_vec_array[3][1] [1:0] $end
$var reg      2 .   f_vec_array[3][0] [1:0] $end
$var reg      2 /   f_vec_array[2][1] [1:0] $end
$var reg      2 0   f_vec_array[2][0] [1:0] $end
$upscope $end
$upscope $end

```

- Expands and prints the value of each member of a packed union.

Consider the following example:

```
module testit;

    typedef logic [1:0] t_vec;

    typedef union packed {
        t_vec f_vec;
        struct packed {
            logic f_a;
            logic f_b;
        } f_ps;
    } t_pu_v;
    typedef union packed {
        struct packed {
            logic f_a;
            logic f_b;
        } f_ps;
        t_vec f_vec;
    } t_pu_s;
    t_pu_v var_pu_v;
    t_pu_s var_pu_s;
endmodule
```

The VCD file created in the previous example is as follows:

```
$scope module testit $end
$scope fork var_pu_v $end
$var reg      2 -      f_vec [1:0] $end
$scope fork f_ps $end
$var reg      1 .      f_a $end
$var reg      1 /      f_b $end
$upscope $end
$upscope $end
$scope fork var_pu_s $end
$scope fork f_ps $end
$var reg      1 0      f_a $end
$var reg      1 1      f_b $end
$upscope $end
$var reg      2 2      f_vec [1:0] $end
$upscope $end
$upscope $end
```

The Command File Syntax

Using a command file, you can generate:

- A VCD file for the whole design or for the specified instances.
- Only the port information for the specified instances.
- An EVCD file for the specified instances.

Note the following before writing a command file:

- All commands must start as the first word in the line, and the arguments for these commands should be written in the same line. For example:

```
dumpvars 1 adder4
```

- All comments must start with “//”. For example:

```
//Add your comment here
dumpvars 1 adder4
```

- All comments written after a command must be preceded by a space. For example:

```
dumpvars 1 adder4 //can write your comment here
```

A command file can contain the following commands:

```
dumports instance [instance1 instance2 ....]
```

Specify an instance for which an EVCD file has to be generated. You can generate an EVCD file for more than one instance by specifying the instance names separated by a space. You can also specify multiple `dumports` commands in the same command file.

```
dumpvars [level] [instance instance1 instance2 ....]
```

Specify an instance for which a VCD file has to be generated. `[level]` is a numeric value indicating the number of levels to traverse down the specified instance. If not specified, or if the value specified is "0", then all the instances under the specified instance are dumped.

You can generate a VCD file for more than one instance by specifying the instance names separated by a space. You can also specify multiple `dumpvars` commands in the same command file.

If this command is not specified or the command has no arguments, then a VCD file is generated for the whole design.

```
dumpvcdports [level] instance [instance1 instance2 ....]
```

Specify an instance whose port values are dumped to a VCD file. `[level]` is a numeric value indicating the number of levels to traverse down the specified instance. If not specified, or if the value specified is "0", then the port values of all the instances under the specified instance are dumped.

You can generate a dump file for more than one instance by specifying the instance names separated by a space. You can also specify multiple `dumpvcdports` commands in the same command file.

Note:

`dumpvcdports` splits the inout ports of type wire into two separate variables:

- one shows the value change information driven into the port. VCS adds a suffix `_DUT` to the basename of this variable.
- the other variable shows the value change information driven out of the port. VCS adds a suffix `_TB` to the basename of this variable.

`dutsuffix DUT_suffix`

Specify a string to change the suffix added to the variable name that shows the value change data driven out of the inout port. The default value is `_DUT`. The suffix can also be enclosed within double quotes.

`tbsuffix TB_suffix`

Specify a string to change the suffix added to the variable name that shows the value change data driven into the inout port. The default value is `_TB`. The suffix can also be enclosed within double quotes.

`starttime start_time`

Specify the start time to start dumping the value change data to the VCD file. If this command is not specified, the start time is the start time of the VCD file.

Only one `+start` command is allowed in a command file.

`endtime end_time`

Specify the end time to stop dumping the value change data to the VCD file. If this command is not specified, the end time will be the end time of the VCD file.

Only one `+end` command is allowed in a command file, and must be equal to or greater than the start time.

Limitations

- `dumpports` is mutually exclusive with either the `dumpvars` or `dumpvcdports` commands. The reason for this is that `dumpports` generates an EVCD file while both `dumpvars` and `dumpvcdports` generates standard VCD files.
- Escaped identifiers must include the trailing space.
- Any error parsing the file causes the translation to terminate.

The vpdmerge Utility

Using the `vpdmerge` utility, you can merge different VPD files storing simulation history data for different simulation times, or the parts of the design hierarchy into one large VPD file.

The syntax is as shown below:

```
vpdmerge [-h] [-q] [-hier] [-v] -o merged_VPD_filename
         input_VPD_filename input_VPD_filename ...
```

Usage:

`-h`

Displays a list of the valid options and their purpose.

`-o merged_VPD_filenames`

Specifies the name of the output merged VPD file. This option is required.

`-q`

Specifies quiet mode. Disables the display of most output to the terminal.

`-hier`

Specifies merge input based on unique hierarchy. (Default is merge input based on time.)

If `-hier` option is specified:

- The VPD files being merged must be split on scope boundaries.
- The first VPD file containing signals for a scope is used to create the values for those signals. Values for those signals in all other VPD files are ignored.

`-v`

Specifies verbose mode. Enables the display of warning and error messages.

Restrictions

The `vpdmerge` utility includes the following restrictions:

- VCS must have written the input VPD files on the same platform as the `vpdmerge` utility.
- The input VPD files cannot contain delta cycle data (different values for a signal during the same time step).
- The input VPD files cannot contain named events.

- The merged line stepping data does not always accurately replay scope changes within a time step.
- If you are merging VPD files from different parts of the design using the `-hier` option, the VPD files must be used for distinctly different parts of the design, they cannot contain information for the same scope.
- You cannot use the `vpdmerge` option on two VPD files created based on timing, for both timing and hierarchy (using the `-hier` option) based merging.

Limitations

The verbose option `-v` may not display error or warning messages in the following scenarios:

- If the reference signal completely or coincidentally overlaps the compared signal.
- During hierarchy merging, if the design object already exists in the merged file.

During hierarchy merging, the `-hier` option may not display error or warning messages in the following scenarios.

- If the start and end times of the two dump files are the same.
- If the datatype of the hierarchical signal in the dump files do not match.

Value Conflicts

If the `vpdmerge` utility encounters conflicting values for the same signal with the same hierarchical name, but in different input VPD files, it does the following when writing the merged VPD file:

- If the signals have the same end time, `vpdmerge` uses the values from the first input VPD file that you entered on the command line.
- If the signals have different end times, `vpdmerge` uses the values for the signal with the greatest end time.

In cases where there are value conflicts, the `-v` option displays messages about these conflicts.

The `vpdutil` Utility

The `vpdutil` utility generates statistics about the data in the `vpd` file. This utility takes a single VPD file as input. You can specify options to this utility to query at design, module, instance, and node levels.

This utility supports time ranges and input lists for query on more than one object. Output is in ASCII to stdout with option to redirect to an output file.

For more information, see “Using the vpduilt Utility to Generate Statistics” .

9

Compile Time Productivity

High compile time is the most common issue in the verification process. It is important to reduce the scratch compile time and the incremental compile time because of minor changes in design or testbench. Small changes in the design or verification environment lead to recompilation of the complete design and test environment. The recompilation time affects the productivity and efficiency of the verification engineers and impacts the release cycle and time to market.

Partition Compile

Partition Compile is a VCS feature that allows you to compile portions of the design and get a faster turnaround time during the iterative process of compiling and recompiling. During scratch compile, VCS Partition Compile identifies partitions in the design and compiles them in parallel. During incremental compile, the same mechanism recompiles only the modified partition that is suitable for debug or development phase of the project.

Partition Compile creates separate partitions for various parts of the design and testbench. It splits the DUT and testbench to smaller partitions for maximum parallel gains. During recompile, partitions of the modified part are revised and compiled. You can see a faster turnaround time by recompiling only some partitions and not repeating the entire compilation of the design.

You can enable Partition Compile with the `-partcomp` compile-time option. VCS partitions the design using autopartitioning and give you the benefits in compile time for all the compile steps.

You can specify these partitions in the following ways:

- Using Autopartitioning

Use autopartitioning where VCS identifies the partition automatically. This is the recommended mode and very helpful when using Partition Compile for the first time on your design. See [Autopartitioning](#).

- Specifying the partitions manually in one of the following ways:
 - In an `+optconfigfile` configuration file (for Verilog/SV designs) see [Specifying Partitions in an +optconfigfile File](#).
 - In a V2K or SV configuration (for Verilog/SV/VHDL/SystemC and Mixed HDL designs) see [Specifying Partitions in a V2K or SystemVerilog Configuration](#).

Some of the Partition Compile features are:

- Compile partitions independently and in parallel
- Reduced disk space across multiple partitions
- Faster IP integration in SOC environments

The Partition Compile use model is very similar to the existing VCS regular compile use model. No changes are required to the source code of the design and testbench when migrating to Partition Compile when compared to the regular VCS compilation use model.

Partition Compile creates separate partitions for various parts of the design and testbench. During recompile, only a partition or partitions for the modified part are revised and compiled. You then see faster turnaround times from recompiling only some partitions and not repeating entire compilation of the design.

You can also improve the compile-time performance by compiling the partitions in parallel on multicore machines instead of serial compiling in regression mode.

Partition Compile also helps reduce the disk space for multiple tests if the design and test are in separate partitions.

This chapter explains how to use Partition Compile in the following sections:

- [Autopartitioning](#)
- [Specifying Partitions Manually](#)
- [Partition Compile Use Model](#)
- [Partition Compile Example](#)
- [Parallel Compilation With Partition Compile](#)
- [Relocation Methods with Partition Compile](#)
- [Cross-Module References \(XMRs\)](#)
- [Coding Guidelines](#)
- [Scenarios Causing a Recompilation of a Partition](#)
- [Achieving the Best Turnaround Time with Partition Compile](#)

- [Profiling of Compilation Time](#)
- [Improving Partition Compile Performance for SDF Designs](#)
- [Rewriting XMRs to VPI Calls](#)
- [Partition Compile Limitations](#)

For Partition Compile with SystemC, see *SystemC Features* Chapter in the *VCS LCA Features Guide*.

Autopartitioning

You enable autopartitioning when you use the `-partcomp` compile-time option. This is the default and recommended mode of partitioning. Autopartitioning automatically partitions the design and the testbench. To specify the partitions yourself apart from the autopartitioning see section [Specifying Partitions Manually](#).

Use the following compile-time options to control autopartitioning:

`-partcomp`

Applies autopartitioning to both modules and SystemVerilog packages.

`-partcomp=autopart_low`

Applies autopartitioning to both modules and packages with a low threshold, which results in smaller and more numerous partitions.

`-partcomp=autopart_high`

Applies autopartitioning to both modules and packages with a high threshold, which results in larger and fewer partitions.

Larger and fewer partitions take more time to compile or elaborate than smaller and more numerous partitions.

`-partcomp=autopart_relax`

Applies autopartitioning in relax mode. Use `-partcomp=autopart_relax` option when `-partcomp=autopart_high` gives less balanced partitions. It relaxes the partitioning with a high threshold to get maximum gain in turnaround time.

`-partcomp=autopartdbg`

Creates the `vcs_partition_config.file` file, which contains the design partitioning information. Note that this option can be used along with any of the other autopartitioning options to create this configuration file.

Once the partition config file is generated, it can be modified to add/delete any partitions based on user requirements. This partition config file can be passed at the `vcs` step to pick the modified partitioning.

You can enter the `-partcomp=autopartdbg` compile-time option along with any of the other partition compile-time options and arguments.

`-partcomp=nomodautopart,nopkgautopart`

Disables autopartitioning of modules and packages. You can use the `nomodautopart` and `nopkgautopart` options individually also.

Note:

- VCS recommends using autopartitioning as default and decides the number of partitions, particularly when Partition Compile is used for the first time. Later you can experiment with the `-partcomp=autopart_low` or `high` compile-time options.
- You can use manual partitioning for part of the design and testbench and use autopartitioning to partition the remainder.
- You must pass the `-partcomp` option directly in the `vcs` command line and not within any other file that is passed using the `-f` or `-F` option.

Specifying Partitions Manually

This section describes the ways to manually specify partitions in the following sections:

- [Specifying Partitions in a V2K or SystemVerilog Configuration](#)
- [Specifying Partitions in an +optconfigfile File](#)
- [Specifying Partitions in a VHDL Configuration File](#)
- [Cell or Instance Based Partitions](#)
- [Instance or Cell Based Partitions in a V2K/SV Configuration](#)
- [Package Based Partitions](#)

Note:

VCS recommends auto partitions for the optimum compile turn around time. If required, you can use manual partitions for your design. You may not always see the partitions as per your input because VCS does not shut down the auto partitioning and creates the partitioning based on combined inputs from manual and auto partitioning.

Specifying Partitions in a V2K or SystemVerilog Configuration

There are two ways to specify a partition in Partition Compile:

- with a `+optconfigfile` configuration file
- with a V2K/SV configuration.

This section describes using a V2K/SV configuration.

Partition Compile adds an extension keywords to V2K/SV configurations to specify partitions. [Example 45](#) is a V2K/SV configuration for the example source code in [Example 55](#).

Example 45 Partition Compile V2K/SV Configuration File

```
// topcfg.v
config topcfg;
design top; // top-level module
partition instance top.t1; // partition for program test
instance top.m1 use multiplier;
instance top.m2 use multiplier;
partition instance top.m1; // partition for multiplier
                        //instance m1
partition instance top.m2; // partition for multiplier
                        // instance m2
endconfig
```

The new keyword `partition` begins a partition specification.

The keyword `instance` specifies that the partition is for an instance of a module or a program block. The keyword `cell` specifies that the partition is for all instances of a module or a program block. For a discussion of which to use see [Instance or Cell Based Partitions in a V2K/SV Configuration](#)

The above configuration specifies:

- one partition for program block `test`
- one partition for the `m1` instance
- one partition for the `m2` instance.

The default and unspecified partition contains the top-level module `top` and module `adder`.

2 step commands for Partition Compile

Add the `topcfg.v` file and `-top config_name` option to the `vcs` command line, for example:

```
% vcs -partcomp -top topcfg topcfg.v top.v test.v \
add_mult.v [other options]
```

Then run the simv executable.

```
% simv
```

3 step commands for Partition Compile

Analyze the partition configuration file `topcfg.v` file, for example:

```
% vlogan -work rtllib -sverilog test.v add_mult.v top.v topcfg.v [other options]
```

Then you enter only the configuration name on the vcs command line, for example:

```
% vcs -partcomp topcfg [other options]
```

Then run the simv executable.

```
% simv
```

Specifying Partitions in an +optconfigfile File

You specify a partition in this file with a line beginning with the keyword `partition`. The keyword `cell` specifies that the partition is for all instances of the module name or program block name that follows.

Example 46 Partition Compile +optconfigfile Configuration File

```
// +optconfigfile configuration file cfg.txt
partition cell test;
partition cell multiplier;
```

This configuration file specifies:

- one partition for program block `test`
- one partition for both instances of module `multiplier`

The default and unspecified partition contains the top-level module `top` and module `adder`.

Compile the design and testbench with the `-partcomp` option and specifying the configuration file with the `+optconfigfile` option, for example:

```
% vcs -partcomp top.v test.v add_mult.v \ +optconfigfile+config.txt
[other options]
```

Specifying Partitions in a VHDL Configuration File

VHDL and VHDL-on-top, mixed-HDL designs require a VHDL configuration file to specify the partitions (see [Example 47](#)). The syntax and semantics of the configuration file are the same as for a regular VHDL configuration file, with the addition of the new keyword `partition`.

Example 47 VHDL Configuration File for Partition Compile

```
//file topcfg.vhd
library IEEE;
library work;
USE STD.textio.all;
USE IEEE.STD_LOGIC_TEXTIO.all;
use IEEE.STD_LOGIC_1164.all;
use work.all;
configuration topcfg of top is
for top_arch
    for partition i1: child use entity work.child(module);
    end for;
end for;
end;
```

Note:

The `partition` keyword with `for all` and `for other` clauses of a VHDL configuration is not allowed in Partition Compile.

Analyze the VHDL configuration file along with other VHDL source files using the `vhdlan` command. For example:

```
% vhdlan top.vhd topcfg.vhd
```

Cell or Instance Based Partitions

As shown in [Example 55](#), you can specify a partition for all instances of a module definition or program block with the keyword `cell` following the keyword `partition`.

When you specify a partition with the keyword `cell`, it applies to its instances and sub instances. See [Example 48](#),

Example 48

```
module top;
sub s();
endmodule

module sub;
bot b();
endmodule

module bot;
endmodule

cfg1
partition cell sub;
```

For `cfg1`, the `sub` and its child instance `bot` are compiled in the partition `sub`.

You can specify the Verilog library for a module or program block by prepending the library to the module name. [Example 49](#) is an example of specifying a library for a module definition in an `+optconfigfile` configuration file.

Example 49 Cell Based `+optconfigfile` Configuration File

```
partition cell MYLIB.mod;
partition cell tb;
```

In the first partition in [Example 49](#), the module definition named `mod` is in the Verilog Library directory named `MYLIB`.

You can also specify a partition for an instance of a module or program block with the keyword `instance` instead of the keyword `cell` as shown in [Example 50](#).

Example 50 Instance Based `+optconfigfile` Configuration File

```
partition instance top.t1;
partition instance top.m1;
partition instance top.m2;
```

[Example 50](#) is for the SystemVerilog code in [Example 55](#) and specifies three partitions:

- one for the instance `t1` of program block `test`
- one for instance `m1` of module definition `multiplier`
- one for instance `m2` of module definition `multiplier`

A line in the `+optconfigfile` configuration file beginning with the `partition` and `instance` keywords can only have the hierarchical name for one instance. You cannot use instance based configurations to specify a partition for multiple instances, for that you need to use a module or cell based partition.

Instance or Cell Based Partitions in a V2K/SV Configuration

You can specify the partitions in the V2K/SV configuration file based on the `instance` or `cell` keyword. The configuration shown in [Example 51](#) uses `cell` entries, following the keyword `partition`, to specify a partition for all instances of a module or testbench.

Example 51 Partitions By Module or Cell

```
// topcfg.v
config topcfg;
design rtllib.top; // top-level module name
default liblist rtllib;
partition cell test; // partition for test cell
instance top.m1 use multiplier;
instance top.m2 use multiplier;
partition cell multiplier; // partition for multiplier cell
endconfig
```

In this configuration separate partitions are specified for:

- all instances of the `cell multiplier`
- all instances of the `cell test`.

The configuration shown in [Example 52](#) uses `instance` entries, following the keyword `partition`, to specify partitions for specific instances.

Example 52 Partitions By Instances

```
// topcfg.v
config topcfg;
design rtllib.top; // top-level module name
default liblist rtllib;
partition instance top.t1; // partition for
                           // testbench instance t1
instance top.m1 use multiplier;
instance top.m2 use multiplier;
partition instance top.m1; // partition for
                           // multiplier instance m1
partition instance top.m2; // partition for
                           // multiplier instance m2
endconfig
```

In this configuration separate partitions are specified for:

- the `instance top.t1` of the testbench program `test`
- the `instance top.m1` of module definition `multiplier`
- the `instance top.m2` of module definition `multiplier`

Package Based Partitions

Partition Compile allows you to create separate partitions for SystemVerilog packages, using the keyword `package`. For example:

```
partition package library1.package1
partition package library2.package2;
```

This example creates one partition for package `package1` and one partition for `package2`.

This example prepends the SystemVerilog library name to the package. Prepending a library is not required.

```
partition package library1.package1 library2.package2;
```

This example creates a single partition for both `package1` and `package2`.

Note:

- Partitions for VHDL packages are not supported.
- VCS also creates an internal or default partition for all SystemVerilog packages, but this internal partition does not effect a package partition you specify.

Partition Compile Use Model

Partition Compile can be used with VCS Two-step Flow or Three-step Flow.

The Partition Compile Two-Step Flow

1. Synopsys recommends using autopartitioning using the `-partcomp` option. This is the default mode in partition compile flow. For more details, see the [Autopartitioning](#) section.
2. You can create a partition configuration file to generate different partitions than autopartitioning to fine tune the performance. This is not mandatory step and can be chosen only when you are looking for different partitioning.

For an elementary design that has a module named `mod` in a Verilog library named `MYLIB` (`filename mod.v`) and another module named `tb` (`filename tb.v`). To create a partition for each module definition, the configuration file is in [Example 53](#).

Example 53 Configuration File for Partition Compile

```
// configuration file config.txt
partition cell MYLIB.mod;
partition cell tb;
```

In [Example 53](#) one partition is for all instances of module `mod` and another partition is for all instances of module `tb`.

You can specify different partitions for different instances (see [Cell or Instance Based Partitions](#)).

For V2K or SV packages (see [The Partition Compile Three Step Flow](#)).

You can also use a V2K or SV configuration to specify partitions instead of using an `+optconfigfile` configuration (see [Instance or Cell Based Partitions in a V2K/SV Configuration](#)).

- In addition to the partitions you specify, VCS creates internal or default partitions for the following parts of the design:
 - a partition for the top-level module
 - a partition for all SystemVerilog packages

a partition for all SystemVerilog interfaces

- Run the simulation.

```
% simv
```

There is no runtime option for Partition Compile.

1. Based on the simulation results, modify the source code. Let's say you only modify tb.v.

```
% vi tb.v
```

2. Re-compile the design as follows for Partition Compile again by entering the same exact compilation command line:

```
% vcs tb.v -y MYLIB +libext+.v -partcomp
```

For user-defined partitions, re-compile the design as follows:

```
% vcs tb.v -y MYLIB +libext+.v -partcomp \ +optconfigfile+config.txt
```

- Run the simulation again.

```
% simv
```

The Partition Compile Three Step Flow

Partition Compile supports the three step flow too (analysis, elaboration, and simulation).

Here are the steps to run the Partition Compile in the three step flow:

Analysis of Code

- Analyze the files using the regular VCS commands. There is no extra option needed at the analysis stage for Partition Compile.
 - Verilog/SV designs and testbenches use the `vlogan` command.
 - VHDL designs and test benches use the `vhdlan` command. For more information see [Specifying Partitions in a VHDL Configuration File](#).
 - Mixed HDL designs and testbenches use the `vlogan` and `vhdlan` commands.

Here is the `synopsys_sim.setup` file for example [Example 54](#):

Example 54 synopsys_sim.setup File

```
WORK > DEFAULT
DEFAULT : ./rtllib
rtllib : ./rtllib
```

The example `vlogan` command line to analyze the source code in [Example 55](#) is as follows

```
% vlogan -work rtllib -sverilog test.v add_mult.v top.v \
[other options]
```

After analyzing the files, the next step is elaboration. The example `vcs` command to compile the design in the Partition Compile flow is as follows:

```
% vcs -partcomp top [other options]
```

To specify additional partitions, use the `+optconfigfile` option as follows:

```
% vcs -partcomp top +optconfigfile+cfg.txt [other options]
```

For more details, refer to [Step 2](#) of Two-Step flow.

Commonly Used Partition-Compile Options

You can use the following options for partition compile flow:

`-dir/-partcomp_dir=<dir_name>`

Generates partition information in the specified directory.

`-sharedlib/-partcomp_sharedlib=<dir_name>`

Reuses the existing partition information from the specified partition database and recompiles only the modified design-specific partitions.

Limiting Partitions

You can use the following options to limit the module and package partitions in partition compile flow:

`-partcomp=modpart_limit:N`

Limits the module partitions to the specified number.

`-partcomp=pkgpart_limit:N`

Limits the package partitions to the specified number.

Note:

The `modpart_limit` and `pkgpart_limit` are considered as guidelines by the VCS partitioner. The tool can still create slightly more or slightly less partitions based on the module or package sizes and inter-dependencies.

Specifying a Location for the Partition Data Generation

Use the `-partcomp_dir=dir_path` option to specify a directory for the partition data, for example:

```
% vcs -partcomp top.v test.v add_mult.v \
-partcomp_dir=./PARTCOMP [other options]
```

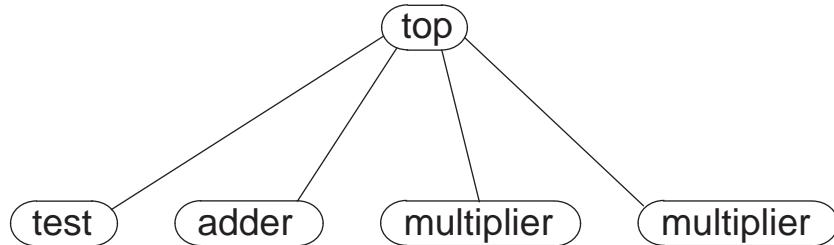
Using this example, VCS creates a directory named `PARTCOMP` to contain the partition data.

For mixed HDL or VHDL designs, a directory named `vhdl_objs_dir` is created for data related to VHDL partitions. This directory is located in the current path by default. The `-partcomp_dir` option changes the location of the `vhdl_objs_dir` to the directory specified by `-partcomp_dir`.

Partition Compile Example

Figure 117 shows the hierarchy in [Example 55](#).

Figure 117 Partition Compile Example Module Hierarchy



The top-level module named `top` instantiates the following (see [Example 55](#)):

- a testbench program block named `test`
- a module named `adder`
- two instances of a module named `multiplier`

Example 55 Partition Compile Example Source Files

```
// file top.v
module top;
  logic [7:0] log1, log2, log3, log4;
  wire [31:0] sum, prod1, prod2;
  adder a1 (sum,log1,log2);
  multiplier m1 (prod1, log1, log2);
  multiplier m2 (prod2, log3, log4);
```

```

test t1 (log1, log2, log3, log4);
endmodule

// file test.v
program test (output [7:0] log1, log2, log3, log4);
...
endprogram

// file add_mult.v
module adder (output [31:0] sum, input [7:0] addend1, addend2);
...
endmodule

module multiplier (output [31:0] product, input [7:0] multiplier,
multipliand);
...
endmodule

```

Compile the design and testbench using the default autopartitioning mode of partition compile. This mode ensures that the optimal number of balanced partitions are created. For example:

```
% vcs -partcomp top.v test.v add_mult.v [other options]
```

You must repeat this same command line precisely each time you compile and elaborate the design and testbench.

You then simulate the design and testbench by running the `simv` executable as follows:

```
% simv [other options]
```

There is no runtime option for Partition Compile.

You can modify the source code for the `multiplier` module and `test` program block, by editing the `add_mult.v` and `test.v` files.

You can compile these partitions by entering the same `vcs` command line:

```
% vcs -partcomp top.v test.v add_mult.v [other options]
```

VCS recompiles only the revised source code for the `multiplier` module and `test` program block.

You can then rerun the updated `simv` executable:

```
% simv [other options]
```

Parallel Compilation With Partition Compile

During compilation, VCS builds the design hierarchy. The first VCS compilation step is called scratch compilation.

When scratch compile times are too long, partition the DUT and compile the partitions in parallel. Parallel compilation of partitions can significantly improve scratch compile times.

To enable the parallel compilation of partitions, use the `-fastpartcomp=j<N>` option on the `vcs` command line. For example:

```
% vcs top.v test.v add_mult.v -partcomp -fastpartcomp=j4 [other options]
```

This command allows the compilation of a maximum of 4 partitions in parallel on a 4 core (or above) machine.

Adaptive Scheduling Using fastpartcomp

Adaptive scheduling helps you to work out the optimal value for `fastpartcomp`. In this option, VCS auto detects the number of parallel partition processes the machine can handle considering its configuration and current load.

Use Model

Use the `-partcomp=adaptive_sched` option to enable adaptive scheduling for parallel compilation. For example,

```
% vcs top.v test.v add_mult.v -partcomp=adaptive_sched [other options]  
[other partcomp options]
```

If you want to specify the maximum limit of parallel processes to use, specify `-fastpartcomp=j<N>`. For example,

```
% vcs top.v test.v add_mult.v -partcomp=adaptive_sched -fastpartcomp=j8  
[other options] [other partcomp options]
```

This command allows the compilation of a maximum of 8 partitions in parallel.

Turbo Compile Flow

Verification engineers change only the verification environment (testbench or tests) and the DUT remains constant. Many engineers are involved in the design and verification of small to large designs. Some of the common scenarios are:

- Changes are local to the testbench or the verification environment, but not in the design.
- Changes are local to a design, but not in the testbench or the verification environment.

Each change can lead to huge compile time. Partition compile creates partitions of the design and the testbench that are compiled separately and enables faster compilation of design and verification environment during incremental compile.

The incremental compile flow with partition compile does not trigger the compilation of unused or unchanged common portions of the design or tests. Incremental compilation is faster because it shares the compiled data from other portions of the design. Hence it reduces compile-time and disk space consumption across multiple tests.

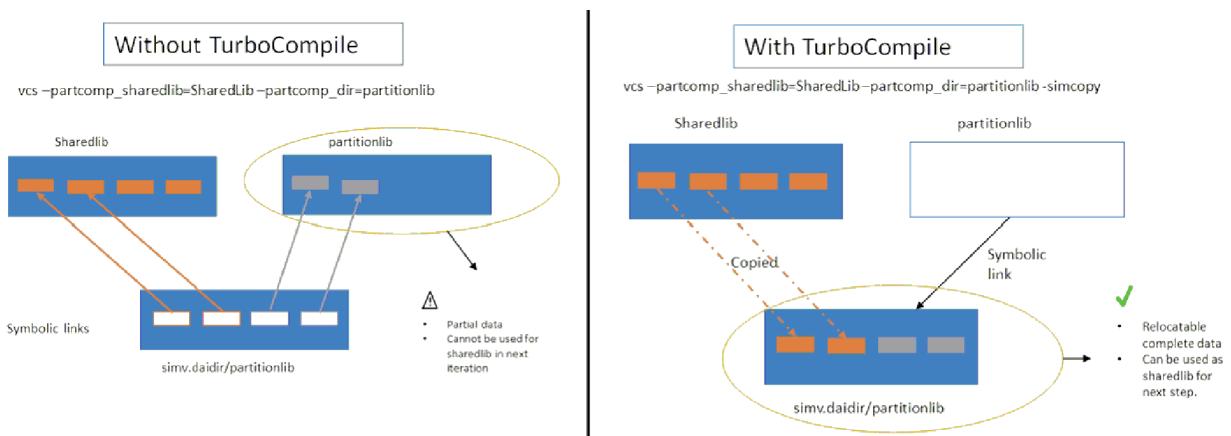
Also, incremental compilations can be done in parallel to further speed up the compile turnaround time across multiple tests or multiple users in SOC designs.

At the same time, the compile areas need to be relocatable, as anything can be moved anywhere to reduce NFS access for users.

All these can be achieved using Turbo Compile Flow with Partition Compile. It helps to create a relocatable database along with the benefits of incremental compile with partition compile flow.

The following diagram illustrates the advantage of turbo compile flow.

Figure 118 Turbo Compile Flow with simcopy Advantages



The turbo compile can be used in the following models for running regressions, debug, or development phases.

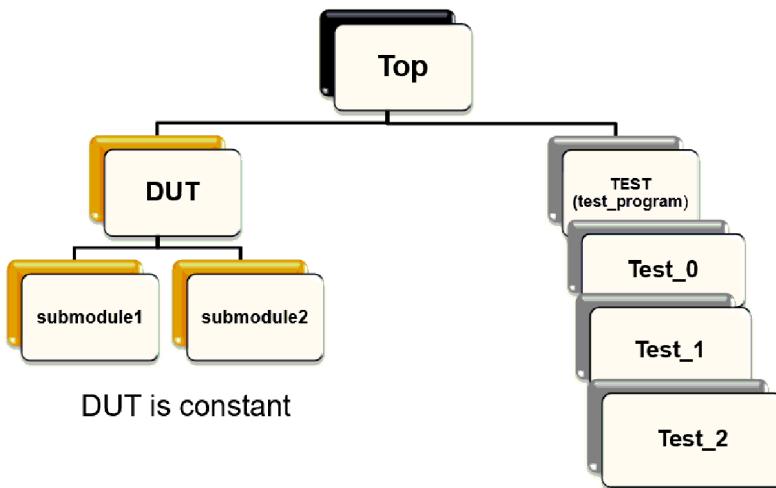
- [Multiple Test Model](#)
- [Multiple User Model](#)

Multiple Test Model

This model helps you to compile multiple tests in parallel. In this model, the DUT code is compiled once in the scratch compile, and the different test cases are compiled in different directories in parallel.

The following diagram illustrates a constant DUT code and different tests (TEST, test_0, test_1, and so on).

Figure 119 Multiple Test Use Model



The use model is as follows:

1. Generate the scratch partition database as follows:

```

Compile test_0
% vlogan test_0.sv -f hdl_files \
% vcs test -partcomp -partcomp_dir=common_dir \
-o simv_0 <other options>

```

2. After generating the scratch partition database, you can perform parallel compilations for multiple test cases with -partcomp sharedlib usage as follows:

```

Compile test_1
% cd test_1
% vlogan test_1.sv -f hdl_files
% vcs test -partcomp -partcomp_sharedlib=../common_dir \
-partcomp_dir=test_1_dir -o simv_1 <other options>
Compile test_2
% cd test_2
% vlogan test_2.sv -f hdl_files
% vcs test -partcomp -partcomp_sharedlib=../common_dir \
-partcomp_dir=test_2_dir -o simv_2 <other options>

```

Here, the scratch partition database common_dir is shared with other test cases.

Note:

If you use the same directory names for `-partcomp_dir` and `-partcomp_sharedlib`, VCS issues an error. Ensure that you use different directory names.

For example,

```
% vcs -partcomp -partcomp_dir=recompile_dir  
-partcomp_sharedlib=common_dir <source files>
```

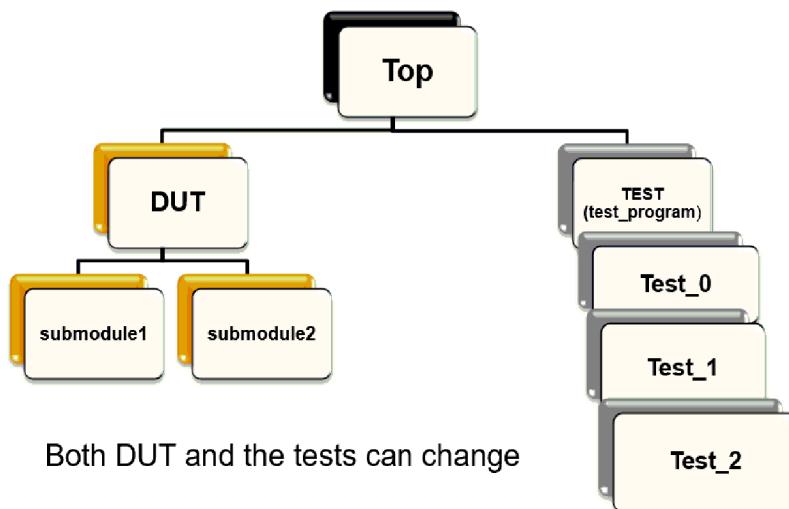
Multiple User Model

This model is suitable for multiple users to compile the design incrementally and in parallel, while referring to a central database in debug, or development phases.

This model supports change in both DUT and testbench.

The following diagram illustrates that multiple users work on a DUT code and different tests (TEST, test_0, test_1, and so on).

Figure 120 Multiple User Use Model



In multiple user environments, you can perform the compilation in two ways.

1. The parallel compilation of multiple user use model is as follows:

First Compile

```
% vlogan -f hdl_files
```

Shared Directory

```
% vcs test -partcomp -partcomp_dir=common_dir \
-o simv_0 <other options>
```

User 1 **Change in Design or Test Case by User 1**

```
% cd user_1
% vlogan -f hdl_files
% vcs test -partcomp -partcomp_sharedlib=../common_dir \
-partcomp_dir=local_dir \
-o simv_1 <other options>
```

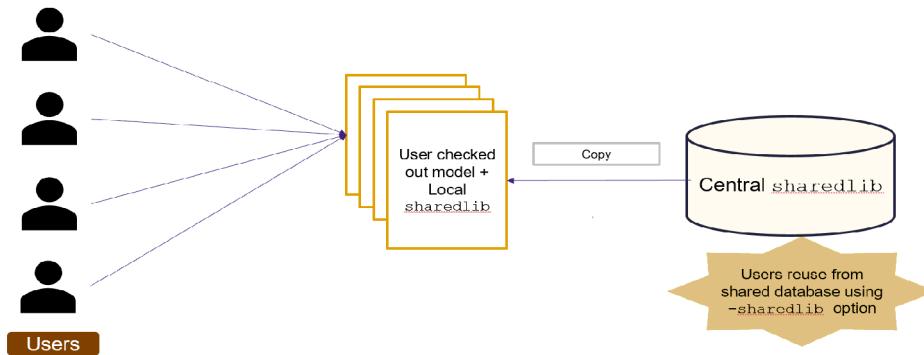
User 2 **Change in Design or Test Case by User 2**

```
% cd user_2
% vlogan -f hdl_files
% vcs -partcomp -partcomp_sharedlib=../common_dir \
-partcomp_dir=local_dir \
-o simv_2 <other options>
```

2. The incremental compilation of multiple user use model is as follows:

In this model, there is one scratch compile, and the rest of the compiles are incremental.

Figure 121 Incremental Compile Use Model



Consider User1 and User2.

1. Generates the `-partcomp_dir` scratch partition database for a new project.

2. User1 performs the following steps:

- checkouts a model from the repository
- copies `-partition_dir` and use it as `-partcomp_sharedlib` to reuse the compiled database

```
%cp -rf /<central_sharedlib_path>/partition_dir /user1_local_path/
% vlogan <new_source_file/changed source/options addition/removal>
%vcs -partcomp -partcomp_dir=usr1_partition_dir
-partcomp_sharedlib=partition_dir <other options>
```

- makes changes to some files and elaborates the model

3. User2 performs the following steps:

- clones User1's model
 - copies `-partition_dir` and use it as `-partcomp_sharedlib`
- ```
%cp -rf /<central_sharedlib_path>/partition_dir /user2_local_path/
% vlogan <new_source_file/changed source/options addition/removal>
%vcs -partcomp -partcomp_dir=usr2_partition_dir
-partcomp_sharedlib=partition_dir <other options>
```
- makes more changes and elaborates model

## Diagnostics

In the `-partcomp_sharedlib` flow, during recompile, the following compile-time diagnostic messages are issued along with the reason for recompilation:

| Compile-time Message | Scenario                 | Diagnostics                                                   | Example                                                                                                                   |
|----------------------|--------------------------|---------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------|
| PC_SHARED            | No change in the design. | Partition is not recompiled. The PC_SHARED message is issued. | Note-[PC_SHARED] Reusing shared partition Reusing partition 'dut_top' from shared library './central_dir/dut_top_wqtq3c'. |

| Compile-time Message          | Scenario                                                                               | Diagnostics                                                                                                                                                                                                                                                                       | Example                                                                                                                                                                  |
|-------------------------------|----------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| PC_NOT_SHARED or PC_RECOMPILE | Local scope changes to the modules, program, or interface.                             | The PC_NOT_SHARED or PC_RECOMPILE messages are issued for partitions that need to be recompiled and the message points to the module line that is subjected to recompilation. Recompiling DUT from the ./central_dir/dut_to pshared library and the ./central_dir/test test file. | Warning-[PC_NOT_SHARED]<br>Cannot reuse shared partition<br>partition 'dut_top' from shared library<br>'./common_dir' Modified Design Units :                            |
| PC_SKIP_FULLDR                | Local scope change to the internal modules and no change in the global view of design. | The PC_SKIP_FULLDR message is issued.                                                                                                                                                                                                                                             | Note-[PC_SKIP_FULLDR]<br>Skipping partcomp design resolution<br>Skipping partition compile global design resolution as there was no change in the global view of design. |

#### Note:

If you use the `-q` option during compilation:

- In scratch compile, the PC\_GEN\_PARTITION message is suppressed.
- In incremental compile, the PC\_RECOMPILE message is suppressed.

## Relocation Methods with Partition Compile

VCS allows you to move the simulation executable to the location of your choice. With the relocated database, you can get the benefit of a faster compile turnaround time by reusing the existing partitions and compiling (relocatable compilation) or running the simulation (relocatable simulation) in a new directory.

The following are the two methods to relocate the database in partition compile - turbo compile flow:

- [Using -simcopy Compile Time Options](#)
- [Using simcopy Utility](#)
- [Turbo Compile Example](#)

## Using -simcopy Compile Time Options

When the `-simcopy` compile-time options are used along with `-partcomp_sharedlib`, it enables VCS TurboCompile flow (by reusing existing partitions) and also enables relocatable compilation or relocatable simulation in a new directory. It helps to run the simulation in parallel for multiple user environments.

The `-simcopy` options are listed in the following table:

*Table 17 The -simcopy Compile Time Options*

| Option                               | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                  | Use Model                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                    |
|--------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <code>-simcopy</code> (default mode) | This option moves the partitions from the partition library (default: <code>partitionlib</code> unless specified with <code>-partcomp_dir</code> ) into the <code>simv.daidir/partitionlib</code> directory. In the subsequent compile step with <code>-partcomp_sharedlib</code> option is used, this option does not copy all partitions, only the partitions shared from the partition library which is specified with <code>-partcomp_sharedlib</code> are copied into the <code>simv.daidir/partitionlib</code> directory in your current working area. | % vcs -partcomp -simcopy<br>The above command moves the partition library, <code>partitionlib</code> directory into the <code>simv.daidir</code> directory and ensures that the complete content is relocatable. Also, enables this directory to be used in the relocatable compilation. Now, you can use the <code>unix copy (cp)</code> command to relocate the <code>simv*</code> folders and run the simulation from the destination directory.% cp -r simv simv.daidir<br>The above command copies the <code>simv.daidir</code> directory to the specified destination directory. If you run incremental compile in the destination directory, you must remove the simulation database using the following commands:% rm -rf simv*<br><b>Note:</b> If you add the <code>-simcopy</code> option to an incremental compilation (that is not present in the scratch compilation), it causes recompilation. |

**Table 17     The -simcopy Compile Time Options (Continued)**

| Option           | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                        | Use Model                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                |
|------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| -simcopy_opts=mv | This option moves partitionlib located in the simv.daidir directory to the specified destination directory. Therefore, saves disk space and compilation time. Note: If the -partcomp sharedlib is specified, the partitions that are reused are also copied to simv.daidir/partitionlib directory. However, when -simcopy_opts=mv is specified, then the partitions reused from the sharedlib are moved from the original sharedlib to the simv.daidir/partitionlib directory. This makes the original -partcomp_sharedlib unusable. Hence, you must use the -simcopy_opts=mv option with caution. | % vcs <options> -partcomp -simcopy -simcopy_opts=mv<br>You can also use the -simcopy option to enable relocatable compilation. Consider the following example:<br>% vcs -partcomp -simcopy.<br>The above compile command moves the content of partitionlib with three partitions P1, P2, and P3 into simv.daidir/partitionlib at the end of compile. Here, the simcopy_opts=mv option does not affect the scratch compile.<br>% cp -r simv.daidir/partitionlib <destination dir><br><destination dir>/scratch_partitionlib><br>The above command copy partitionlib with three partitions P1, P2, and P3 into the specified destination directory.<br>% cd destination dir% \rm -rf simv* csrc*% vcs -partcomp -simcopy -simcopy_opts=mv -partcomp_sharedlib=<scratch_partitionlib><br>The above commands apply incremental compilation to the sharedlib. Considering that the partitions P1 and P2 are reused and P3 is recompiled, the partitionlib directory has only one partition P3. So, in this case -simcopy_opt=mv option ensures that the partitions P1 and P2 are moved from scratch_partitionlib to simv.daidir/partitionlib. |

## Using simcopy Utility

VCS provides the `simcopy` utility that is available inside the `simv.daidir` directory to relocate databases after the compilation. Multiple users can use this utility as it supports relocatable compilation and relocatable simulation which can also be done in parallel.

Following is the use model:

1. Use the following command to generate `partitionlib`:

```
% vcs <options> -partcomp
```

**Note:** The `partitionlib` directory need to be retained to get the coverage score, when the partition compile is used in the flow. Otherwise, URG or Verdi shows empty coverage data.

2. The following command moves `partitionlib` to `simv.daidir` and creates a copy of `simv` at the specified destination directory. The original `simv` is not deleted.

```
% simv.daidir/simcopy <destination dir>
```

**Note:**

The `simcopy` utility also has `simv.daidir/simcopy -inline` option which runs `simcopy` utility in inline mode.

You must follow below mentioned guidelines when you use the `simcopy` utility:

- After you copy the database using the `simcopy` utility, you must point to the `partitionlib` inside the `simv.daidir` directory and not to your current directory.
- If you are updating the `sharedlib` area, you should first clean the existing `sharedlib` and then copy the new `partitionlib` to avoid overwriting.

## Turbo Compile Example

This section describes turbo compile flow with the use case of `-partcomp_sharedlib` and `-simcopy` options.

In this use case,

- Compiling source (RTL and test) files in different directories and the source is changed during recompile.

Consider the following use case:

Table 18     *rtl.v* and *test.v*

| <code>%cat ./src/rtl.v</code>                                                                                                                                                                                                                    | <code>%cat ./src/test.v</code>                                                                                                                                                                                                                                                                              |
|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <pre>module rtl_count(input     reg clk,rst, output reg     [3:0] cnt);     always@(posedge clk or     posedge rst)     begin         if(rst)             cnt &lt;= 4'b0000;         else             cnt &lt;= cnt + 1;     end endmodule</pre> | <pre>module top;     reg clk,rst;     reg [3:0] cnt;     always #5 clk = ~ clk;     rtl_count inst (clk,rst,cnt);     initial \$monitor(\$time," clk     = %b rst = %b cnt =     %0d",clk,rst,cnt);     initial begin         rst = 1'b1;         clk= 1'b0;         #100 \$finish;     end endmodule</pre> |

Run the design using the following commands:

```
% rm -rf simv* csrc* partition* *dir*
% vlogan -sverilog ./src/rtl.v ./src/test.v
% vcs -sverilog -partcomp -simcopy top
% cp -r simv.daidir/partitionlib ./some/user/path/last_pc_db
% cd ./some/user/path/
```

The above example shows the scratch compile command. After successful compile, copy the partitionlib generated inside simv.daidir to the incremental compile location.

VCS generates the following messages at the scratch compile step that shows the partitions generated with the PC\_GEN\_PARTITION message.

```
Note-[PC_GEN_PARTITION] Generating partition
Generating new partition '_vcs_pc_package_' at
'./partitionlib/_vcs_pc_package__B0vusd'.
```

```
Note-[PC_GEN_PARTITION] Generating partition
Generating new partition 'top' at './partitionlib/top_1YNkOc'.
```

The source file is changed as shown in the following table.

**Table 2** rtl.v and test.v

Table 19    *rtl.v* and *test.v*

| %cat ./src/rtl.v                                                                                                                                                                                                                                 | %cat ./src/test.v                                                                                                                                                                                                                                                                                                                                                                      |
|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <pre>module rtl_count(input     reg clk,rst, output reg     [3:0] cnt);     always@(posedge clk or     posedge rst)     begin         if(rst)             cnt &lt;= 4'b0000;         else             cnt &lt;= cnt + 1;     end endmodule</pre> | <pre>module top;     reg clk,rst;     reg [3:0] cnt;     always #5 clk = ~ clk;     rtl_count inst (clk,rst,cnt);     initial \$monitor(\$time," clk     = %b rst = %b cnt =     %0d",clk,rst,cnt);     initial begin         rst = 1'b1;         clk= 1'b0;         #3;         `ifndef RUN_COUNTER         rst = 1'b0;         `endif         #100 \$finish;     end endmodule</pre> |

Run the design using the following commands:

```
% vlogan -sverilog ./src/rtl.v ./src/test.v
+define+RUN_COUNTER
% vcs top -sverilog -partcomp -simcopy
-partcomp_sharedlib=last_pc_dir
% simv
```

During recompilation, use the `-partcomp_sharedlib` option that reuses the existing partitions copied from the scratch compile and regenerates only the changed partitions.

VCS generates the following messages at the recompile step that shows the partitions that are newly generated with the `PC_GEN_PARTITION` message and the partitions that are reused with the `PC_SHARED` message.

Note-[`PC_SHARED`] Reusing shared partition  
    Reusing partition '`_vcs_pc_package_`' from shared library '`last_pc_db`'.

Warning-[`PC_NOT_SHARED`] Cannot reuse shared partition  
    Cannot reuse partition '`top`' from shared library '`last_pc_db`'

Modified Design Units :

```
top : "./src/test.v", 1

Note-[PC_GEN_PARTITION] Generating partition
 Generating new partition 'top' at './partitionlib/top_3dviq'.
```

**Note:**

VCS generates an error for the following use cases:

- Compiling source files in same directories with same partition directory `-partcomp_dir` and `-partcomp_sharedlib` directory names during scratch or incremental compilation.
- If `-simcopy` options are not used consistently at all compile commands, then the relocatable compilation fails.

## Cross-Module References (XMRs)

When your design is partitioned there is a chance that there are cross module references (XMRs) across partitions; that is, signals and variables from one partition being accessed from another partition. You can handle XMRs as follows when using the Partition Compile flow:

- If XMRs are not changing, no extra option or configuration file is needed.
- If XMRs change, both the referrer and the referee partitions are recompiled.
- If XMRs change, you can optimize the compile behavior as follows:
  - Any change in XMRs, trigger recompilation of the referee partition.
    - Use the `-partcomp=gen_xmr_config=top_module` option to generate configuration file that contains the XMR information of the design.
    - Pass this configuration file to see optimized recompile behavior where only referred partitions are recompiled. Use the following syntax for the configuration file:

```
xmr {module_or_program_name} {signal_name};
```

- If you do not specify any option, your design is till compiled or recompiled.

This section discusses the following topics:

- [Support for Skipping Intra Position Rewrite in Partition Compile Flow](#)
- [Usage](#)
- [Examples](#)
- [Limitation](#)

## Support for Skipping Intra Position Rewrite in Partition Compile Flow

VCS skips XMR rewrite for intra partition XMR during child compilation.

In the following scenarios, the intra partition XMR cannot be skipped:

- XMR to interface
- XMR target is structure or structure member
- XMR is upwards
- XMR target is also the target of cross partition XMR

## Usage

This feature is enabled using the MHID *PC\_SKIP\_XMR\_REWRITE*.

## Examples

The following are the examples of this feature:

*Example 56 test1.sv*

```
module top;
sub s();
initial $display(s.a);
endmodule
module sub;
int a;
endmodule
```

| Partition Compile Flow With MHID<br><i>PC_SKIP_XMR_REWRITE</i>                                            | Default Partition Compile Flow                                                                                                                                                                                                                     |
|-----------------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <pre>pre_rewrite_rad.v: /* M#0 */ module top; sub_7saKSD s(); initial \$display(top.s.a); endmodule</pre> | <pre>pre_rewrite_rad.v: /* M#0 */ module top; int_vcs_pc____vcs_0_a_00000D; sub_7saKSD s(); initial \$display(_vcs_pc____vcs_0_a_00000D); initial begin \$\$xmr_map_rel(0, _vcs_pc____vcs_0_a_00000D, "top.s.a", -2147483648); end endmodule</pre> |

**Example 57 test2.sv**

```
module top;
sub s();
mid m(s.w);
endmodule
module sub;
wire w;
endmodule
module mid(w);
input w;
leaf l(w);
endmodule
module leaf(w);
input w;
endmodule
```

| Partition Compile Flow With MHID<br><b>PC_SKIP_XMR_REWRITE</b>                                      | Default Partition Compile Flow                                                                                                                                                                                                                              |
|-----------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <pre>pre_rewrite_rad.v: /* M#0 */ module top; sub_7saKSD s(); mid_7saKSD m(top.s.w) endmodule</pre> | <pre>pre_rewrite_rad.v: /* M#0 */ module top; wire_vcs_pc____vcs_0_w_00000D; sub_7saKSD s(); mid_7saKSD m(_vcs_pc____vcs_0_w_00000D);  initial begin     \$\$xmr_map_rel(0,     _vcs_pc____vcs_0_w_00000D, "top.s.w",     -2147483648); end endmodule</pre> |

## Limitation

- Fine-Grained Parallelism (FGP) is not supported for this feature.

---

## Coding Guidelines

Here are some guidelines to follow when using the partition compile flow to get the best turn around time:

### 1.1 Wrap testbench code in SV packages.

- Avoid code in \$unit (code changes in the \$unit scope triggers recompilation of the entire design).
- Multiple packages can be used to make logical partitions of the code.

1.2 Avoid XMRs in the testbench code where possible.

- Use SV virtual interfaces.
- XMRs prevent putting code in packages (this is an SV LRM restriction).
- Runtime impact with partition compile (resolutions at runtime).
- Test to test compile time can be high if a large portion of testbench is compiled with each test due to `$unit` and the testbench contains XMRs to the design.
- If XMRs are required only for debugging purpose, use ``ifdef` to include the XMR for compilation.

1.3 Import packages only where required.

- Package changes can cause recompile of multiple partitions (if all packages are imported in a file and that file is included in many partitions).

1.4 Do not assume any specific initial blocks order of execution.

1.5 Do not combine source code from multiple partitions in the same file.

- Changes in one partition can cause recompilation of other partitions in the same file.
- Limit optimization in vlogan.

1.6 Scripts (Makefile) should not have options that translate to `+define+` on the vlogan command line for the testbench or DUT, especially for multiple tests.

- Ensure that test to test does not have to recompile the testbench or DUT. Test specific code should be limited to a test case file.
- Different modes (for example speed) can be configured through variables instead of ``define`.

1.7 Use multiple vlogan commands.

- Logical vlogan commands are:
  - 1 for each package
  - 1 for test cases
  - 1 or more for DUT
- Optimize time for reanalysis.
- Reanalysis of source code (vlogan or vhdlan) with different command-line options causes the recompilation of the source code and all partitions that share the logical library.

- \$unit changes are local.
- Makefile targets should analyze only required set of files.

1.8 For parallel compilation of test cases, preferably have one directory for one test case.

- The test cases should be setup in such a way that multiple test cases can be compiled in parallel and do not over-write the compiled data of other test cases.

1.9 If the same file (especially with interfaces and packages) is included in multiple files, the vlogan commands of those multiple files should not direct the analyzed data to different logical libraries.

1.10 Interface definition to be passed before the virtual interface is created to the vlogan command.

- Prevents creation of one extra dummy interface partition.

1.11 Makefile should not clean by default.

By default, VCS allows calling non-blocking tasks such as tasks without delay, event control or wait statements, from within a function. For blocking task calls, an error message is issued.

In the partition compile flow, the blocking and non-blocking nature of a task call may not always be determined at compile-time. Compilation may complete without errors when calling a blocking task from within a function.

You can use the `-partcomp=LRM_1800_2012` compile-time option to enable checking of stricter SystemVerilog compliant rule. When the option is specified, an error message is issued in the partition compile flow for calling any tasks, blocking or non-blocking, from within a function.

## Scenarios Causing a Recompilation of a Partition

The following cases cause a recompilation of a partition:

- Change of a module definition in a partition, either instance- or cell-based.
- Change in VCS command-line option (addition, deletion, or sub-argument value change).
- Change in the `synopsys_sim.setup` or `+optconfigfile` file.
- Global design property changes like adding `$dumpvars` to a partition, in which case all partitions need to be recompiled.
- Changes to the ports or parameters that affect the ports of a partition. If there is a parent and child partition, and this change is only to the child partition, the parent need not be recompiled.

- Changes to `$unit` scope. Any change (addition of a new signal, type, module, or interface) to a `$unit` causes recompilation of all the partitions associated with the `$unit`.
- Changes to an interface require the recompilation of all modules connected to that interface.
- Changes to a shared package. In this case, all the partitions sharing this package require recompilation.
- Reanalysis of source code (`vlogan` or `vhdlan`) with different command-line options causes recompilation of the source code and all partitions that share the logical library.
- Changes in input files like assert hier file, `-cm_hier` file, UPF file, and XPROP config files.
- The change in a package partition leads to recompilation of its dependent partitions.
- When two or more modules/packages/interfaces are grouped into one partition, then even if only one module/package/interface is changed, the whole partition is recompiled.

## Achieving the Best Turnaround Time with Partition Compile

To get the maximum gain in turnaround time from the partition compile flow, note the following:

- Do not create too many partitions
  - Start with autopartitioning and then tune for performance, if needed.
- Avoid code in `$unit`
  - Code changes in the `$unit` may affect many partitions.
  - `$unit` is not sharable across multiple tests.
- Create separate partitions for each top-level design in hierarchy, if more than one top-level change in subsequent compilation.
  - All parallel top-level are a single partitions, by default.
  - Create a separate partition for the top-level that changes frequently compared to the other parallel top-level that are idle.
- Create separate partitions for each test (program) and only test partition gets recompiled in this case.
- Make external IP/VIP modules as separate partitions.

- Partitions for mixed HDL designs
  - Create separate partition for a VHDL entity if it is going to change frequently.
  - If it is not a separate partition then it is compiled as part of default VHDL top partition.
- Create one or more partitions for SV packages that are going to change frequently.
- Import packages only where required.
  - Package changes can cause recompile of multiple partitions (if all packages are imported in a file and that file is included in many partitions).

## Profiling of Compilation Time

The `-pcmakeprof` compile-time option enables profiling of time spent in each step of a compilation. The profiling result is tabulated and printed at the end of the compilation log. The following are usage examples of the option in different compilation flows:

### Single Compile, Two-step

```
vcs -sverilog -pcmakeprof test.sv
```

### Single Compile, Three-step

```
vlogan -sverilog test.sv
vcs top -pcmakeprof
```

### Partition Compile, Two-step

```
vcs -sverilog -partcomp -partcomp_dir=p_dir -pcmakeprof test.sv
```

### Partition Compile, Three-step

```
vlogan -sverilog test.sv
vcs top -partcomp -partcomp_dir=p_dir -pcmakeprof
```

The following are examples of the profiling result:

### Single Compile, Two-step

| Activity    | Real(s) | User(s) | Sys(s) |
|-------------|---------|---------|--------|
| Parsing     | 0.00    | 0.00    | 0.00   |
| Compiling   | 0.42    | 0.08    | 0.10   |
| Elaboration | 0.39    | 0.24    | 0.04   |
| Total_time  | 2.00    | 0.51    | 0.55   |

#### Partition Compile, Two-step

| Activity                          | Real(s) | User(s) | Sys(s) |
|-----------------------------------|---------|---------|--------|
| Parsing                           | 0.00    | 0.00    | 0.00   |
| Global_analysis                   | 0.65    | 0.08    | 0.10   |
| _Partition:_vcs_pc_package_2Aiove | 1.57    | 0.33    | 0.45   |
| _Partition:top_NoR9Nc             | 1.18    | 0.32    | 0.36   |
| All_partition_time                | 2.78    | 0.66    | 0.83   |
| Stitching_&_Elaboration           | 1.56    | 0.48    | 0.61   |
| Total_time                        | 6.00    | 1.43    | 2.33   |

#### Parsing

Shows the amount of time consumed by parsing all design files.

#### Global\_analysis

Shows the amount of time consumed by loading an analyzed design, resolving the design and creating partitions.

#### Partition

Shows the amount of time consumed by compiling a particular partition.

#### Stitching\_&\_Elaboration

Shows the amount of time consumed by stitching all partitions together.

#### All\_partition\_time

Shows the amount of time consumed by all partitions and stitching the partitions. This number is the sum of the “Partition” numbers and the “Stitching\_&\_Elaboration” number.

Total\_time

Shows the total amount of time consumed by an entire compilation. This number is the sum of all the “Activity” numbers.

**Note:**

Due to floating number precision, the value of “All\_partition\_time” and “Total\_time” may differ slightly from the exact sum of numbers.

## Improving Partition Compile Performance for SDF Designs

In case of SDF interconnect delays, both the source and destination of the interconnect delays are in same partition and therefore DUT is not broken into several partitions. To break the DUT into multiple partitions, support for manual partitions for designs that have SDF interconnect delays across partitions has been added. This allows you to specify explicit partitions to the DUT to achieve better turnaround time.

- [Use Model](#)
- [Usage Example](#)
- [Limitations](#)

### Use Model

To enable this feature, use the `-hsopt=hsimsdfic` compile-time option.

In case of uphierarchy interconnect delays, use the `-hsopt=uphierarchyic` switch.

### Usage Example

Consider the following examples:

*Example 58 basic.v*

```
`timescale 1ns/1ns
//----- Top level ASIC modules-----
//*** top level asic dut_1 ***
module dut_1 (clk,in,out);
 input clk;
 inout in;
 inout out;
 buf (out, in);
endmodule
//*** top level asic dut_2 ***
module dut_2 (clk,in,out);
 input clk;
 inout in;
 inout out;
 not (out, in);
```

```

endmodule
//----- System Modules : Interconnect top level ASIC modules

module top(inout in1,inout in2,inout out1,inout out2,input clk);
dut_1 dut_1_inst_1(clk,in1,out1);
dut_2 dut_2_inst_1(clk,in2,out2);
endmodule
//----- Testbench Module : Stimulus generation -----
module tb();
wire in_1,in_2;
wire out_dut_1,out_dut_2;
reg rin1,rin2;
assign in_1 = rin1;
assign in_2 = rin2;
reg clk=0;
always
#20 clk = !clk;
top asic_top(in_1,in_2,out_dut_1,out_dut_2,clk);
sdf_loader sdf_loader();
initial begin
#21 rin1 = 1;
#80 rin1 = 0;
#80 rin1 = 1;
#80 rin1 = 0;
#100;
#40 rin2 = 1;
#80 rin2 = 0;
#80 rin2 = 1;
#80 rin2 = 0;
#100;
$finish();
end
endmodule
//----- SDF LOADER -----
module sdf_loader();
initial begin
$ssdf_annotation("basic_top.sdf", tb.asic_top);
end
endmodule

```

**Example 59 basic\_cfg**

```

// separating asic_top and sdf loader
partition instance tb.asic_top;
partition instance tb.sdf_loader;

// separating top level asic modules
partition instance tb.asic_top.dut_1_inst_1;
partition instance tb.asic_top.dut_2_inst_1;

```

**Example 60 basic\_top.sdf**

```
(DELAYFILE
(SDFVERSION "OVI 3.0")
(DIVIDER .)
(TIMESCALE 1ns)
(CELL
(CELLTYPE "top")
(INSTANCE)
(DELAY
(ABSOLUTE
(INTERCONNECT in1 dut_1_inst_1.in (1) (4))
(INTERCONNECT in2 dut_2_inst_1.in (2) (3))
)
)
)
)
)
```

You can use the following command to run the example:

```
% vcs basic.v -sverilog +optconfigfile+basic_cfg -partcomp
-hsopt=hsimsdfic

% simv
```

VCS creates two new partitions `dut_1` and `dut_2`.

## Limitations

The feature has the following limitations:

- The feature only works for manual partitioning with cross-partition INTERCONNECT delays in top partition.
- The feature only works for INTERCONNECT delays on partition boundaries.
- Cross-partition SDF for mixed HDL design is not supported.
- Instance instantiation inside a `generate` block is not supported.
- Different partitions with different timescale is not supported.
- Instance arrays are not supported.
- Extended identifiers are not supported.

## Rewriting XMRs to VPI Calls

VCS rewrites the XMRs to VPI calls using `-partcomp=xmr_mark_for_incr` option. VCS provides better incremental compile time performance when DUT remains same but TB has changing XMRs from DUT.

This feature supports the following XMRs:

- XMR must be top down XMR only.
- Used in force/release statements.
- Load usage.

## Use Model

Use the following use model to enable this feature:

```
$vcs test.v -sverilog -full164 +optconfigfile+test.cfg -partcomp
-partcomp=xmr_mark_for_incr
```

Rewrite occurs only for the XMRs specified in the modules in `xmr_mark_for_incr` configuration rule.

## Configuration File

User needs to specify all the partitions and the XMRs which are changing across the compile in `xmr {all_module}` configuration rule to avoid recompilation of target partitions.

Following are the configurations used for this feature:

### *Example 61 test.cfg*

```
// +optconfigfile configuration file
partition cell mid;
partition cell bdie_behav;xmr_mark_for_incr {top};
xmr_mark_for_incr {bdie_behav};
xmr {all_modules} {vec};
xmr {all_module} {vec2};
```

## Example

Following is the example of this feature:

```
module top;
mid m();
bdie_behav b();
wire [7:0] TEMPERATURE_BDIE;
`ifdef INCR
assign TEMPERATURE_BDIE = top.m.vec2[7:0];
`else
assign TEMPERATURE_BDIE = top.m.vec[7:0];
`endif
always @(TEMPERATURE_BDIE) begin
$display("\nvalue:%d", TEMPERATURE_BDIE);
end
endmodule
module mid;
reg [7:0] vec;
```

```

reg [7:0] vec2;
endmodule
module bdie_behav;
initial begin
force top.m.vec = 20;
#20 release top.m.vec;
#30 force top.m.vec = 56;
#10 release top.m.vec;
#100 $finish;
end
endmodule

```

## Limitations

The following are the limitations of this feature:

- Relative XMRs
- Biidirect XMRs
- Non-constant RHS in force
- MDA
- Low power flow
- Partial Elaboration (PE)
- Internal modules generated from VCS
- SVA properties and sequences
- Any other driver usage than force/release
- XMRs in interface and SV code

## Partition Compile Limitations

The following are the limitations with partition compile:

- The `-race` option is not supported.
- Utilities like `-metadump`, `-Xrawtokens` can be used in regular compile to generate the information.
- Dynamic Reconfiguartion
  - Dynamic configuration with SDF is not supported.
  - Dynamic configuration with VHDL

VHDL instance inside dynamic configuration enabled module

### Dynamic configuration enabled instance in VHDL

- Interface instances with reference ports inside dynamic configuration enabled module.
- XMR across PE is not supported in partition compile flow.
- XMR referrer in PE module connected as module reference port is not supported in partition compile flow.
- Cross-partition SDF (VHDL) is not supported.
- Only designs with SystemC on top are supported.
- When using VMC/SWIFT models then the VMC model cannot be a top-level module of the design.
- Verilog-AMS Mixed Signal Simulation
  - Verilog-AMS `amsd ie` and `setd` commands are supported.
  - GenBlock in Verilog-AMS flow is supported.
  - Mixed Signal Simulation with FINESIM. (No HSIM and Nanosim compatibility with VCS).
  - Partition of top-level cell in the Mixed-Signal setup file is not supported in partition compile flow.
  - Multiple drivers for `wreal` net are not supported in partition compile flow.
- Coverage
  - Constant analysis (`-cm_noconst/-cm_seqnoconst`) is more conservative in partition compile, so some objects which are marked "unreachable" in single compile are not marked in partcomp.
- The `stop -thread <thread_no>` command is expected to be different in partition compile flow due to different thread ID between single compile and partition compile flow.
- For mixed designs, processing of assertion configuration file (through compile-time option `-assert_hier`) occurs at runtime in partition compile flow. This can have some impact on runtime performance or may capture assertion related errors during runtime instead of compile-time.
- Cross-partition DPI routine declaration mismatch error is generated at compile-time in single compile flow for inconsistent declarations of DPI and DPI-C. You can use the

`-error=nodPI-DRWIDT` compile-time option to workaround this error in single compile flow.

- Partition Compile does not support the interface port when it is a multi-dimensional array.

# 10

## VCS Distributed Simulation

---

This chapter explains how to use VCS Distributed Simulation in this release in the following sections:

- [VCS Distributed Simulation](#)
  - [Quick Reference: VCS Distributed Simulation Options](#)
- 

### VCS Distributed Simulation

SoCs compile and simulate individually as separate blocks developed for multi-die verification. However, integrating these SoCs in a top-level SoC poses various challenges including the resolution of modules with the same names and class/package name collision. This process needs to be repeated with every new revision of SoC/sub-systems. Also, the high memory consumption of huge designs and ever-increasing memory requirement of simulation requires high-capacity machines to simulate multi-SoC designs.

VCS Distributed Simulation solution allows you to perform the following simulations:

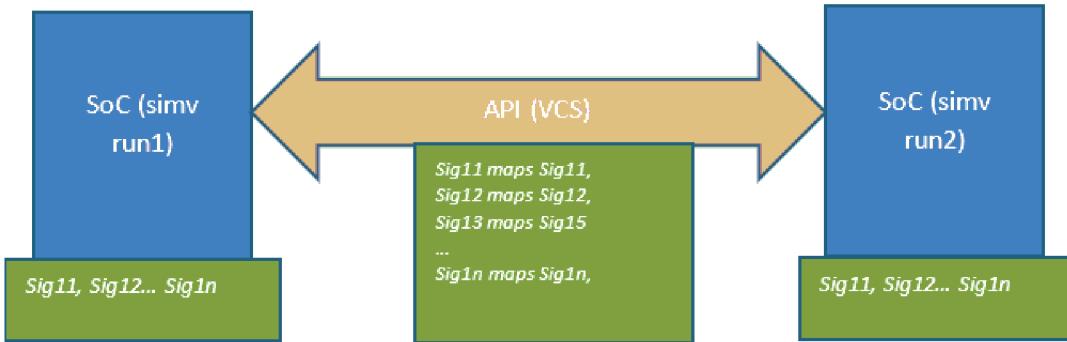
- [Homogeneous Simulation](#)
- [Heterogeneous Simulation](#)

#### Homogeneous Simulation

In this mode, a simulation executable (simv) is duplicated as multiple runs on the same or different machines representing multiple instances of the same module. A common sync signal driving each simv run is used for synchronization of all data transfers across SoCs.

In the following figure, `simv run1` and `simv run2` are same simv processes executed on two different machines.

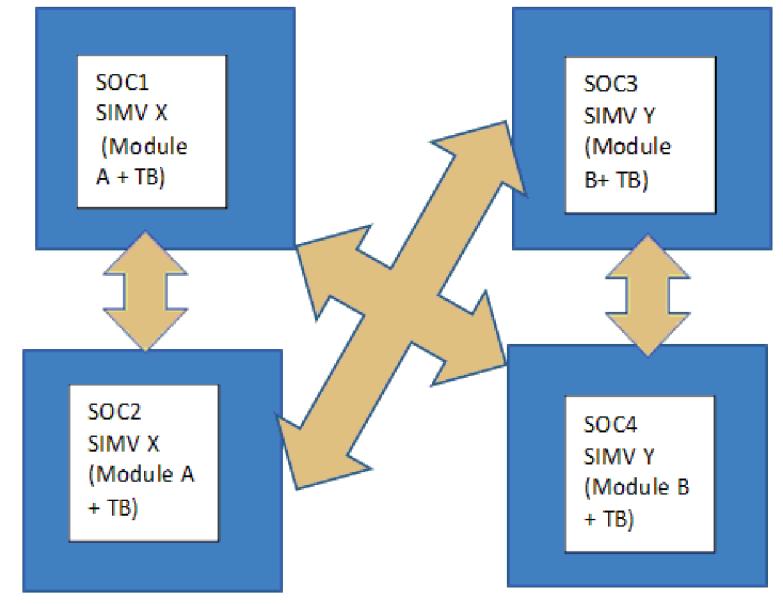
Figure 122 Homogeneous Simulation



### Heterogeneous Simulation

In this mode, multiple simulation executables (simvs) are created for different modules with their own testbenches running on different machines in parallel. Communication across the simvs for data transfers is based on a common sync signal or specific time intervals.

Figure 123 .Heterogeneous Simulation



The Distributed Simulation solution allows you to overcome synchronization issues, maintain deterministic simulation behavior during a rerun of the simulation, and achieve better runtime performance along with a reduction in memory footprint, thereby reducing the need for large-capacity machines.

The following sections describe the VCS Distributed Simulation feature in detail:

- [Distributed Simulation Setup](#)
- [Defining Communication Among Client Simvs](#)
- [RTL Connection](#)
- [Testbench Phase Synchronization](#)
- [Testbench Data Transfer of Class Objects](#)
- [Save Replay Support](#)
- [Save Restore Support](#)
- [User Task to Print Class Objects](#)
- [VCD Dump per Client](#)
- [Bidirect Support Through Wired OR/AND](#)
- [Include Syntax in Configuration File](#)
- [Server Launch as Independent Process](#)
- [HDL Access Across Simvs from Verilog Source](#)
- [Distsim Release Recv Call for TB Data Transfer](#)
- [TB Data Open Channels](#)
- [Distributed Simulation Profile](#)
- [Configuration File Checker Utility](#)
- [Specifying User-Defined TCP Port for Server Launch](#)
- [Specifying Early Simulation Finish Mechanism Across Simvs](#)
- [Cygnus PDF Profile Summary](#)
- [TB Sync Comp Class](#)
- [Packed Structure Support](#)
- [Constant Support in the Configuration File](#)
- [Support for Multiple TB Data Recv Requests](#)
- [Compile Time Configuration File Support](#)
- [Distsim Flow Without Shared File System](#)

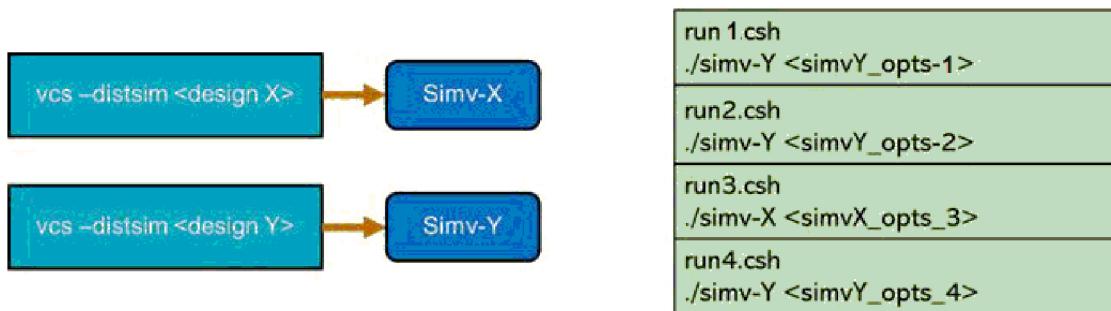
- Early Finish Behavior
- Limitations

## Distributed Simulation Setup

Multiple SoCs which are developed (or can be partitions of one big SoC) individually but are to be verified together become part of the Distributed Simulation setup. A Distributed Simulation setup can take advantage of multiple machines where maximum number of machines required is equal to the number of SoCs in the setup. The minimum is one machine which means multiple SoCs running on the same machine.

A simv corresponding to an SoC which is a part of a Distributed Simulation setup is referred to as **Client Simv**. All the SoCs which are to be run in distributed fashion using Distributed Simulation setup are compiled like a regular simv using the `-distsim` compile option as shown in the following figure:

*Figure 124 Distributed Simulation Setup*



You can use the `-distsim=help` compile option to view all the supported `-distsim` options along with the purpose.

## Defining Communication Among Client Simvs

- To define the communication among the Client Simvs, you must provide a runtime configuration file using the following option:

| Option                                              | Description                   | Requirement                                                            |
|-----------------------------------------------------|-------------------------------|------------------------------------------------------------------------|
| <code>-distsim=config_file:&lt;file_path&gt;</code> | Specifies configuration file. | The configuration file which holds connectivity and other information. |

The configuration file can contain the following synchronization or connectivity information:

### **When to Synchronize?**

- Synchronization Signal
  - Clients can sync based on changes in the value of the specified signal.
- Or Synchronization Time Interval
  - Periodic sync with a given time period
- Which Region?
  - NBA
  - Postponed (End of timestamp)
  - Active (Immediate)

### **What to Synchronize?**

- RTL Connections
  - Set of RTL signals representing RTL data transfers across SoCs/simvs.
- Testbench Phase Synchronization
  - Specification of the OVM/UVM/User-defined phase required for the testbench phase synchronization.
- Testbench Data Transfer
  - Compatible testbench objects to send/receive.

The Client Simvs transmit/receive the required data with respect to the Synchronization signal/time interval as specified above.

The following table describes the configuration file syntax:

*Table 20 Configuration File Syntax*

| Section                                | Syntax                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                        |
|----------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>Sync Mechanism</b>                  | <p>It is mandatory to provide either a sync signal with <code>master_sync_signal</code> or a time interval with <code>master_sync_interval</code> as mentioned below:</p> <p><b>a) Sync Signal Specification</b></p> <p>Syntax:</p> <pre>&lt;master_sync_signal&gt;: s&lt;Client ID&gt;: &lt;Edge Identifier&gt; &lt;Hierarchical Path of Sync Signal&gt;</pre> <p>Where,</p> <p><code>&lt;master_sync_signal&gt;</code>: A required field.</p> <p><code>s&lt;Client ID&gt;</code>: Optional field. Specifies Client specific Sync Signal.</p> <p><code>&lt;Edge Identifier&gt;</code>: Optional field. Use <code>posedge/negedge</code>.</p> |
|                                        | <p><b>b) Time Interval-Based Synchronization</b></p> <p>Syntax:</p> <pre>&lt;master_sync_interval&gt;: s&lt;Client ID&gt;: &lt;interval&gt;&lt;time unit&gt;</pre> <p>Where,</p> <p><code>master_sync_interval</code>: A required field.</p> <p><code>s&lt;Client ID&gt;</code>: Optional field. Specifies Client specific time interval.</p> <p><code>&lt;interval&gt;</code>: Specifies numeric time period.</p> <p><code>&lt;time unit&gt;</code>: Optional field. If not provided, then the timescale of the design is used; otherwise, you can use <code>s/ms/us/ns/ps/fs</code>.</p>                                                    |
|                                        | <p><b>c) Mode Identifier Specification (optional)</b></p> <p>Syntax:</p> <pre>sync:&lt;Mode Identifier&gt;</pre> <p>Valid mode identifier: <code>postponed/active/nba</code></p> <p>Default mode identifier is <code>active</code>.</p>                                                                                                                                                                                                                                                                                                                                                                                                       |
| <b>RTL Connections</b>                 | <p>Syntax:</p> <pre>s&lt;Destination Client ID&gt;:&lt;Hierarchical Path of Driver Signal&gt; = s&lt;source Client ID&gt;:&lt;Hierarchical Path of Load Signal&gt;</pre> <p>For more information, see <a href="#">RTL Connection</a>.</p>                                                                                                                                                                                                                                                                                                                                                                                                     |
| <b>Testbench Phase Synchronization</b> | <p>Syntax:</p> <pre>sync_phase=&lt;Phase_Name&gt;::s&lt;Client_ID_i&gt;:s&lt;Client_ID_j&gt;</pre> <p>Where,</p> <p><code>Client_ID_i</code>, <code>Client_ID_j</code>, and so on are the clients where <code>Phase_Name</code> is to be synced. <code>Phase_Name</code> is User/UVM Phase, it must be present in the specified client. For more information, see <a href="#">Testbench Phase Synchronization</a>.</p>                                                                                                                                                                                                                        |

*Table 20 Configuration File Syntax (Continued)*

| Section                        | Syntax                                                                                                                                                                                                                                                                                                                                                                                                                                                  |
|--------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>Testbench Data Transfer</b> | <p>Syntax:</p> <pre>tbtrans=&lt;Source_Channel_Name&gt;:s&lt;Source_Client_ID&gt;::&lt;Dest_Channel_Name&gt;:s&lt;Dest_Client_ID&gt;</pre> <p>Where, <code>Source_Channel_Name/Dest_Channel_Name</code> are the strings used in Verilog in the <code>distsim</code> system task and <code>Source_Client_ID/Dest_Client_ID</code> are the respective client IDs. For more information, see <a href="#">Testbench Data Transfer of Class Objects</a>.</p> |

## Key Points to Note

- Each Client Simv is identified with a unique integer ID passed with a runtime argument called `client_id`. The following table describes the usage:

| Option                                     | Description                     | Requirement                                                                         |
|--------------------------------------------|---------------------------------|-------------------------------------------------------------------------------------|
| <code>-distsim=client_id:&lt;id&gt;</code> | Specifies a unique ID for simv. | Clients are distinguished based on <code>id</code> to parse the configuration file. |

- The total number of clients is passed with the runtime argument `num_sockets`. The following table describes the usage:

| Option                                                  | Description                                         | Requirement                                                                                        |
|---------------------------------------------------------|-----------------------------------------------------|----------------------------------------------------------------------------------------------------|
| <code>-distsim=num_sockets:&lt;total_Clients&gt;</code> | Specifies the total number of clients in the setup. | The simulation can only proceed when all the clients are connected as they are mutually dependent. |

- The communication initialization is established at time zero using a separate process called **Server Process** which is launched by one of the Client Simvs. The Client Simv which is designated to launch the Server Process is later referred to as **Master Simv**.
- The Server Process is launched on the same machine as the Master Simv.

- You can use the `launch_server` runtime argument to identify which Client Simv would be launching the Server Process as master simv. The following table describes the usage:

| Option                              | Description                  | Requirement                                                                       |
|-------------------------------------|------------------------------|-----------------------------------------------------------------------------------|
| <code>-distsim=launch_server</code> | Launches the Server Process. | It is used to designate one of the simvs to launch the server process internally. |

### Example

Assume number of sockets, `num_sockets = 4`. That is, there are 4 Client Simvs to be run, say `simva`, `simvb`, `simvc`, and `simvd`.

If you perform `simva -distsim=launch_server`, then `simva` becomes Master.

Total number of process running would be 5 (4 simv and 1 Server Process).

- Upon launch, Server Process dumps a file that holds the information of its own IP and TCP Port. This file is called **Server Info File** which is read by all the clients to set up TCP/IP connection with the Server Process.
- You can use the `server_info` runtime argument to specify the path of the Server Info File. This file is written by server and read by all the clients.

The following table describes the usage:

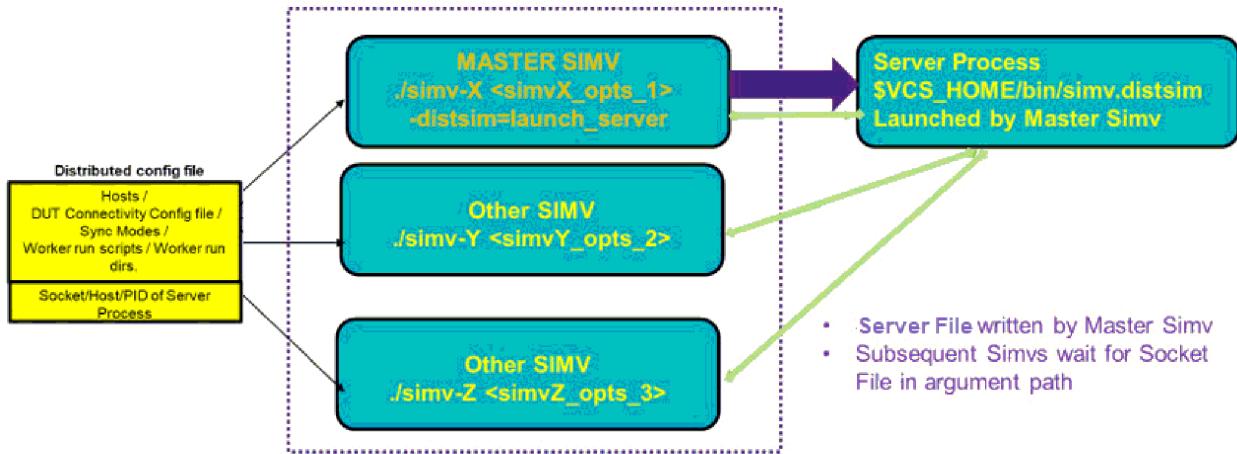
| Option                                              | Description                                  | Requirement                                                                    |
|-----------------------------------------------------|----------------------------------------------|--------------------------------------------------------------------------------|
| <code>-distsim=server_info:&lt;file_path&gt;</code> | Specifies a file to dump server information. | It is used to establish TCP connection amongst all clients through the server. |

- Each synchronization is called a **Sync Point** which is the time when the synchronization signal changes, or the periodic time period hit.
- Communication among the simvs/clients happens only at the specified Sync Point.
- Simulation should exit if any of the simvs are killed or socket connection is lost except in the case where simv is a clean exit/\$finish is called where other simvs should continue to run.
- There can be multiple blocks of synchronization signal (or time period) and their own set of signals.
- You can use the `-distsim=diag:off` option at runtime to disable default diags.

- You can use the `-distsim=diag:data` option at runtime to enable dumping of a file with all the RTL/Testbench value exchanges.
- You can use the `-distsim=log_dir:<dir_path>` option at runtime to specify the path to dump logs.

The following figure shows the simulation block diagram:

Figure 125 Simulation Block Diagram



## RTL Connection

For the RTL connection, synchronization between signals is specified in a text file at runtime. You can use different sync signals for the set of RTL connections/signals to be synchronized or specify a time interval at which the following RTL signals must be synchronized.

Following is the configuration file syntax for RTL connection:

```
s<Destination Client ID>:<Hierarchical Path of Driver Signal> = s<source Client ID>:<Hierarchical Path of Load Signal>
```

A) **Sample Phase:** All the loads (RHS in the above specification of RTL connection) would get sampled together at Sync Point.

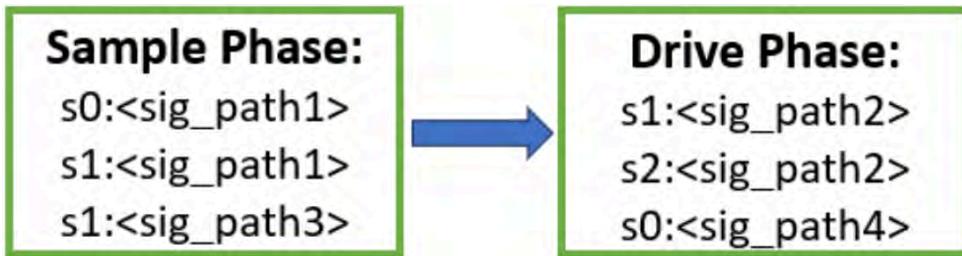
B) **Drive Phase:** All the drive signals (LHS in the above specification of RTL connection) would get driven at Sync Point.

Following is the Configuration file example:

```
master_sync_signal: test.mid.clock_sig
s1:<sig_path2> = s0:<sig_path1>
s2:<sig_path2> = s1:<sig_path1>
s0:<sig_path4> = s1:<sig_path3>
```

There would be no communication between the simvs apart from the sample and drive phases.

Figure 126 RTL Connections



In the above configuration file example, the synchronization happens through the change in the value of the signal `test.mid.clock_sig`.

The line `s1:<sig_path2> = s0:<sig_path1>` implies that the Client Simv with `client_id == 0` has signal `<sig_path1>` which would be driving `<sig_path2>` of Client Simv with `client_id == 1`.

Similarly, the line `s2:<sig_path2> = s1:<sig_path1>` implies that the Client Simv with `client_id == 1` has signal `<sig_path1>` which would be driving `<sig_path2>` of Client Simv with `client_id == 2`.

## Key Points to Note

- Scalars and vectors of the following data types are supported for the sync signal:
  - reg
  - wire
  - logic
  - bit
  - enum
- Memory, real, time, memword, and SV data types are not supported.
- Posedge and negedge keywords can also be specified for sync signals.
- Client-specific Sync signal or Sync interval can be specified using “`s<Client ID>:`” as the prefix.

## Multiple Sync Signals/Intervals

The RTL connections following the Sync Signal or Sync Interval are associated with Sync Signal or Sync Interval. Multiple Sync Signals/Intervals can be used in a configuration file with their own set of RTL connections.

In the case of multiple Sync signals/intervals, one Sync signal/interval is used for the master sync mechanism, and the rest are used for sampling their set of RTL connections.

You must use the identifier, `master_sync_signal` or `master_sync_interval`, to designate the Master sync signal or Master sync interval respectively. The connections that are to be sampled or driven with respect to a different sync signal can be specified in the following format:

```
<RTL Connection> (sample:<sync signal>, drive:<sync signal>)
```

**Note:**

Currently, this format is not supported for sync intervals.

For Client specific Sync signal or interval, add “`s<Client ID>:`” as the prefix in the configuration file.

The following is an example for the same configuration file with multiple sync signals or intervals:

Configuration File:

```
master_sync_signal:s0: top.clk[0]
master_sync_signal:s1: top.clk[0]
master_sync_signal:s2: top.clk[0]

s0:top.A1 = s1:top.b1.a
s0:top.A2 = s2:top.b2.a

s0:top.B1 = s1:top.b1.b (sample:top.clk1, drive:top.clk3)
s0:top.B2 = s2:top.b2.b (sample:top.clk1, drive:top.clk3)

s0:top.C1 = s1:top.b1.c (sample:top.clk2)
s0:top.C2 = s2:top.b2.c (drive:top.clk4)
```

## Testbench Phase Synchronization

TB phase synchronization is a way to synchronize the phases (predefined or user-defined phases) in the OVM/UVM/SystemVerilog Testbench environments. The phase change information is communicated across simvs at the nearest Sync Point only.

The following is the configuration file syntax for the testbench phase synchronization:

```
sync_phase=<Phase_Name>::s<Client_ID_i>:s<Client_ID_j>
```

Where, `Client_ID_i`, `Client_ID_j` are the clients where `Phase_Name` is to be synced. `Phase_Name` is the user-defined/OVM/UVM Phase which must be defined in the specified client.

## Testbench Phase Synchronization Based on User Task

You must add a VCS API macro which can sync or block a phase until all the simvs attain the required simulation phase. This API macro must be added at each place where the Phase Sync is required in your Verilog Files (added in testbench at each Switch Point).

The following is the use model:

```
`VCSDISTSIM_PHASE_SYNC (PHASE_NAME)
```

Where, `PHASE_NAME` is the string used for Phase Sync across the clients.

By default, the API macro is considered for Phase Sync at each phase. You can also specify it in the configuration file if certain phases are to be synched among some specific clients. For example,

```
task run_phase();
 ...
 //Phase Sync Point: SYNC1
 `VCSDISTSIM_PHASE_SYNC("SYNC1");
 ...
 //Phase Sync Point: SYNC2
 `VCSDISTSIM_PHASE_SYNC("SYNC2");
endtask
```

Macro `VCSDISTSIM_PHASE_SYNC` waits until all worker simvs reach same phase/stage (that is, first all worker simvs wait until Sync point reaches `SYNC1`, then they will block for `SYNC2`). Testbench resumes when all worker simvs have switched phases.

## UVM Testbench Phase Synchronization

- Allows you to start default UVM phases together in all simvs.

To apply implicit UVM phase synchronization, you must add a predefined UVM component `dist_tbsync_comp` in the user environment.

The above component adds automatic UVM switch point for all predefined UVM runtime phases across simvs. That is, `reset_phase` of all clients is synchronized. Similarly, `main_phase` is also synchronized across all clients. Next runtime phase will not start until the current phase is completed across all the clients.

```
class test_env extends uvm_env;
 dist_tbsync_comp comp0;
 function void build_phase(uvm_phase phase); super.build_phase(phase);
 comp0 = dist_tbsync_comp::type_id::create("comp0",this);
 endfunction
endclass
```

- Specifying a specific user-defined phase as end of testbench phase synchronization:

`-distsim=end_phase:<FLUSH_PHASE>`

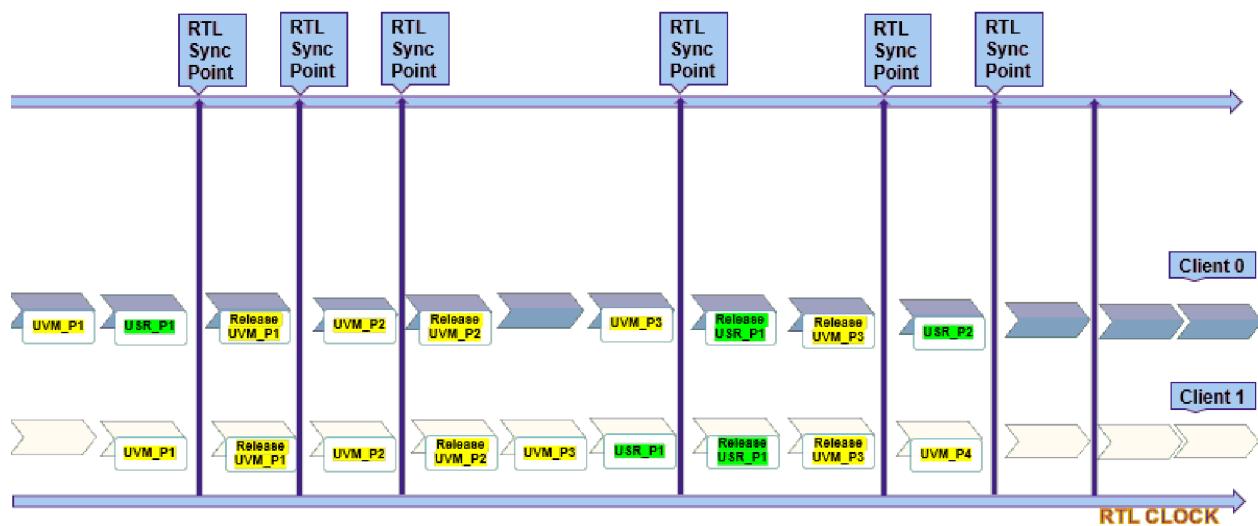
`FLUSH_PHASE` is common among all Worker Testbenches. The order of Phases remains identical in Worker Testbenches.

There can be additional client-specific phases:

- Sync not required for them as they are absent in other clients.
- Can be specified using runtime option or by not adding phase sync macro for them.

The following figure shows UVM Testbench Phase synchronization with RTL sync signal:

Figure 127 UVM Testbench Phase Synchronization With RTL Sync Signal



## Key Points to Note

- The testbench phase sync support is a general support which can be used for UVM, OVM, or SVTB.
- In case of UVM, to sync the default UVM Phases, the `Distsim` Macro is added in UVM's methodology `phase_ready_to_end()` function.
- If the call to `phase_ready_to_end` is done multiple times, then only the first call is honored.
- The testbench phase and testbench data synchronization happen with respect to the Master Sync Signal. This is done to maintain deterministic behavior of simulation even during rerun since the sync is always with respect to the Sync Signal and network delays/lags does not affect the simulation's behavior during rerun.

- Irrespective of the testbench data and testbench phase sync, the simulation or RTL sync do not stop. Therefore, any client might be waiting for a phase sync point or some testbench data, but that does not stop the simulation.
- The macros for testbench phase and testbench data are implemented in the user environment where the sync is required.
- For the testbench phase syncing, the calls must be unique based on the user-defined string which is called as User Phase.
- User-defined phases can be inserted in runtime phases, within the run phase, or separately in parallel with runtime phases in UVM environment.
- The phase sequence order remains identical in all the clients with the following exception:
  - Selected phases might be skipped for Phase Sync.
- Rewind/fast-forward of phases is not allowed if the exact order assumption is not maintained.
- Cross UVM phase syncing is not allowed.
- By default, each phase is synced in all clients. It is assumed that the Phase Sync `distsim` macro is present at all required places, otherwise the set up might hang as missed places never achieve the phase for which Distributed Simulation set up is waiting.
- If any phase is to be skipped for certain clients, then the same must be specified in the configuration file.
- Even if the phase change happens between the Master Sync Signal, it is only transmitted/received at nearest Sync Point, that is when the Master Sync Signal changes.
- Jump backward/forward of phases and cross UVM phase sync is not allowed.
- Multiple UVM components syncing to different phases is not allowed.

## Example

Consider three Clients (`s0`, `s1` and `s2`), UVM Phases (`P1`, `P2`, and `P3`), and User Phases (`CP1`, `CP2`, `CP3`, and `CP4`) in order of their occurrence in a simulation.

The following configuration file specifies phases to sync. No entry implies sync to all.

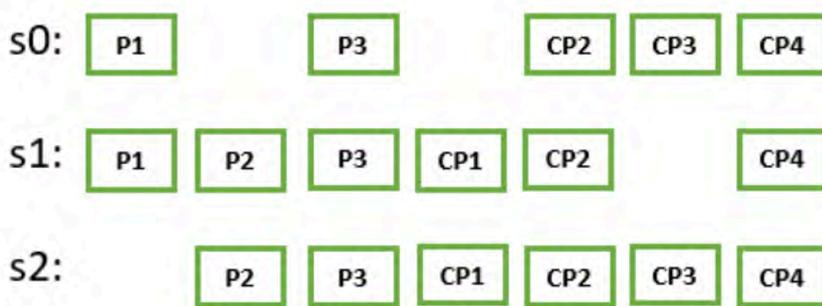
```
master_sync_signal: test.mid.clock_sig

s1:<sig_path2> = s0:<sig_path1>
s2:<sig_path2> = s1:<sig_path1>
s0:<sig_path4> = s1:<sig_path3>
```

```
sync_phase=P1::s0:s1
sync_phase=P2::s1:s2
sync_phase=CP1::s1:s2
sync_phase=CP3::s0:s2
```

Following is the phase order:

*Figure 128 Phase Order*



## Testbench Data Transfer of Class Objects

The testbench data transfer is one-on-one transfer, so for every `send` call there would be one `recv` call. These calls are mapped using unique channel name string whose mapping is provided in the configuration file.

For testbench data transfer, perform the following:

1. Specify the configuration file for the testbench data transfer connections across the clients
2. Use VCS APIs for testbench data send and data receive with channel name as “bit array”. If you are using UVM/OVM, you can create bit array using the `pack()` method on the send side and `unpack()` on the receive side of the testbench.

The requirement is to use compatible testbench data objects.

### Configuration File Syntax for Testbench Data Transfer

Following is the configuration file syntax for testbench data transfer:

```
tbtrans=<source_channel_name>:s<source_client_id>:<destination_channel_name>:<destina
```

Where, `source_channel_name` and `destination_channel_name` are the strings used in the Verilog system task `distsim` and `source_client_id` and `destination_client_id` are the respective client IDs.

### Example-1:

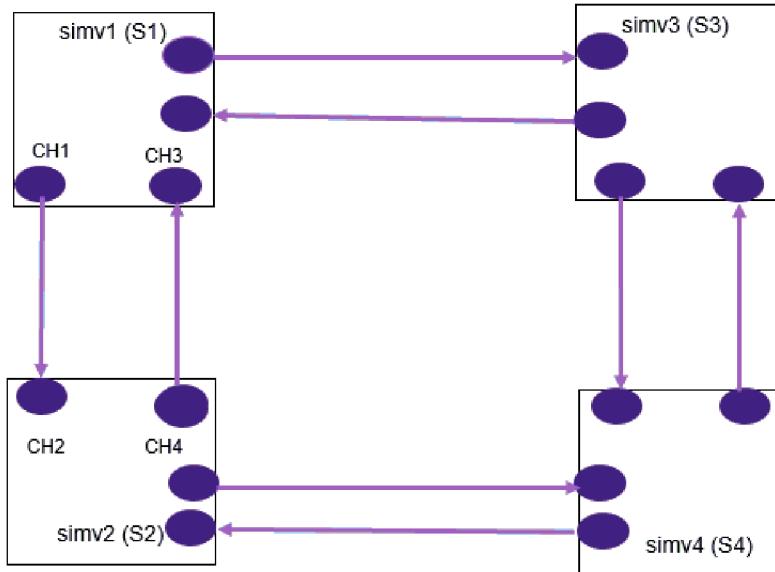
The following configuration file example shows testbench transfer between two clients:

```
tbtrans=CH1::S1::CH2::S2
tbtrans=CH4::S2::CH3::S1
```

This example implies the following:

- Data sent by client S1 through channel CH1 is destined at client S2's channel CH2.
- Data sent by client S2 through channel CH4 is destined at client S1's channel CH3.

*Figure 129 Testbench Transfer Between Two Clients*



### Example-2:

```
master_sync_signal: test.mid.clock_sig

s1:<sig_path2> = s0:<sig_path1>
s2:<sig_path2> = s1:<sig_path1>
s0:<sig_path4> = s1:<sig_path3>

sync_phase=P1::s0:s1
sync_phase=P2::s1:s2
sync_phase=CP1::s1:s3
sync_phase=CP3::s0:s3

tbtrans=CH1:s1::CH0:s2
tbtrans=CH3:s0::CH6:s2
tbtrans=CH4:s1::CH5:s3
```

Signal `tbtrans=CH1:s1::CH0:s2` implies channel `CH1` in the send call of Client 1 is to be received by channel `CH0` in the receive call of Client 2.

Signal `tbtrans=CH3:s0::CH6:s2` implies channel `CH3` in the send call of Client 0 is to be received by channel `CH6` in the receive call of Client 2.

### Testbench Data Transaction Passing Across Simvs

The testbench data is transmitted/received with respect to the nearest Sync Point. The transaction passing mechanism uses bit streams by packing the transaction at the sender and then unpacking the bit stream in transaction at the receiver side. UVM/OVM's `pack/unpack` method can be used to create bit streams and accordingly pass these bit streams across simvs using VCS APIs.

VCS issues an error message if there are two testbench data transfers with same channel name. VCS supports multiple channels with different channel names between two simvs.

The member objects of the class should have static size (nested classes are supported). If dynamic members are present, they must be unpacked accordingly.

#### Note:

VCS supports only bit arrays for TB data transactions.

#### User Task for Send

```
begin
 `VCSDISTSIM_TB_SEND("Const Channel_name", bit_array)
end
```

The highlighted code is the macro you can use to send transaction.

`Channel_name`: Unique channel string name corresponding to data transfer.

`bit_array`: User-defined `bit_array` of dynamic array type containing data to be sent.

#### User Task for Receive (Recv)

```
begin
 `VCSDISTSIM_TB_RECV("Const channel name", bit_array)
end
```

The highlighted code is the macro you can use to receive transaction.

`Channel_name`: Unique channel string name corresponding to data transfer.

`bit_array`: User-defined `bit_array` of dynamic array type in which data is received.

The macros `VCSDISTSIM_TB_SEND`, `VCSDISTSIM_TB_RECV`, and `VCSDISTSIM_PHASE_SYNC` are defined in a SystemVerilog file `vcsdistsim_pkg.sv` using a SystemVerilog package `vcsdistsim_pkg`.

The `vcsdistsim_pkg.sv` file must be included in the test environment along with the import of package in the required testbench, for example,

```
'include "vcs_distsim_pkg.sv" // Include this package file in global
level so that it is parsed before the macros in env
Import vcsdistsim_pkg::*;
class user_env;
task ... (...);
`VCSDISTSIM_PHASE_SYNC(...);
endtask
..
endclass
```

### Key Points to Note

- The member objects of the class should have static size. (nested classes are allowed).
- If dynamic members are present, you must unpack them accordingly. Packing/unpacking into/from a dynamic array can be achieved using the user-defined UVM `do_pack()`/`do_unpack()` methods.
- As the same code can be used for transferring objects across multiple clients, for example, from `client_1` to `client_2` and `client_3` to `client_4`, it is not possible to hard-code sender and receiver IDs during compile time.

## Save Replay Support

The Save Replay feature enables simulation of the Single Simv with the stimulus captured from other clients during the initial run. The initial run is Save Mode, and the re-run of the Single Simv is Replay Mode. It is a Single Mode run that simplifies debugging with a Single Simv.

During the Replay Mode, you can force the RTL signals or the testbench channels to any value using a Force configuration file. The force is applicable for the testbench channels and the RTL signals that are driven in the `-distsim` configuration file. This would be effective from the next Sync Point.

The following steps are involved to use the Save Replay feature:

### 1. Save Mode

In Save Mode, all the RTL receive calls, testbench receive calls, and the testbench phase sync calls to the client are logged with respect to the Sync Point, and the logging files are called replay database (replay db).

The following is the use model:

To enable Save, use the `-distsim=dumpreplay` runtime switch.

The naming format of replay db is as follows:

```
worker_c<Client_ID>_replay.dat
```

Where `Client_ID` specifies the ID of the client whose replay db is dumped for every simulation. By default, it is specified as mentioned above.

By default, the current simulation directory is used as the path for the replay db. However, you can also specify the path by using the following option:

```
-distsim=log_dir:<Directory_path>
```

## 2. Replay Mode

In the Replay Mode, all the RTL drive signals (LHS in the configuration file of RTL connection, see section [RTL Connection](#)) are loaded using the replay db that is dumped in the Save Mode with respect to the Sync Point.

The configuration file must remain the same in the Save and Replay modes.

The testbench behavior in the Replay Mode is as follows:

- The Testbench Send Calls act as dummy.
- The Testbench Receive Calls load the data from the replay db with respect to the Sync Point.
- The Testbench Phase Sync Calls are attended based on Sync Points from the replay db.

Following is the use model:

```
-distsim=replay
-distsim=replay_file:<file path of replay db>
```

For example,

```
-distsim=replay -distsim=replay_file:./worker_c1_replay.dat
```

## 3. Replay Mode with Force

When an RTL signal is driven through a `distsim` configuration file or when the testbench channel is present in the Force configuration file, the value for the specified duration from the Force configuration file is forced instead of the value from the replay db or default simulation.

The syntax for the Force configuration file for replay force is as follows:

```
<Start_time, End_time> <Signal_Name> <value>
```

For example,

```
// <start-time>, <end-time>, <signal_name>, <value>
30,30 top.il.sig h'a23 //To force and release at time 30 and the value
continues to hold till next value update on signal
30,400 top.il.sig h'a23a //To force at time 30 and release at 400. Any
new force while this force is active is an error and new value is
ignored.
60,$ top.il.sig h'a23 // To force at time 60 and let force be active
till the end of simulation.
```

**Note:**

Force happens at the next sync event.

Following is the use model:

```
-distsim=replay
-distsim=replay_file:<file path of replay db>
-distsim=replay_force:<file path of force file>
```

For example,

```
-distsim=replay -distsim=replay_file:./worker_c1_replay.dat
-distsim=replay_force:force.dat
```

**Limitations**

- Specifying time unit is not supported. By default, the simulator timescale is considered.
- Having both, replay force and UCLI force, can generate race conditions.

#### 4. Replay Checker

Replay checker checks whether the Send data in Save run matches with the Send data in Replay run for the client. If there is a mismatch, Replay Checker displays an error in the simulation log.

When a simv design runs in replay mode, Replay Checker gets enabled by default.

The use model for the Replay Checker does not require switch. At the time of failure, it displays the following error in the simulation log:

```
Send message checker failed @ <Simulation time>
```

## Save Restore Support

The VCS Save Restore feature is supported with the Distributed Simulation technology, and it is available only in the UCLI mode.

## Save

The Save feature saves the process image the same as the Single Simulation with the following differences:

- The Save call is attended at the next `Distsim Sync Point`, same as the Distribution Simulation flow. All communication happens at Sync Point. The Sync Point is the master sync signal change or the master sync interval-based simulation time.
- The save command is specified in the UCLI file.
- The process image is saved for the clients when one or more clients invoke the Save call.
- If some clients out of multiple clients do not call for save explicitly, then the Save feature saves the process image for those clients in the following format:

`<user_save_file_prefix>_dist_c<Client_ID>_<save_simulation_time>.SAVE`

Where the string `<user_save_file_prefix>` is optional and can be provided using the following `distsim runtime` switch:

`-distsim=save_file:<user_save_file_prefix>`

Consider three clients, `client0`, `client1`, and `client2`. `Client0` calls for save at time 10 using the `ucli% save save0` option that results in save for all the three clients.

For `client0`, the same image gets dumped at time 10 with the `save0` file name. For `client1` and `client2`, the save image gets dumped at time 10 with the `dist_c1_10.save` and `dist_c2_10.save` file names, respectively.

- The Save Calls are attended in the order they arrive with respect to Sync Points across all the clients.
- Subsequent Save calls from other clients are attended similarly with respect to Sync Points across all clients.

Following is the use model:

`ucli% save <filename>`

Where `filename` is the name of the file to which simulation snapshot is written.

For the Save feature, except VCS Save commands, no extra switches are required.

For example,

The sample run script for two client IDs (0 and 1) is as follows:

```
#!/bin/csh -fx
rm -rf simv* csrv* S*
```

```
vcs w0.v -sverilog -full64 -o simv0 -Mdir=csrc0 -distsim
 -debug_access+all | & tee vcs.log0 &
vcs w1.v -sverilog -full64 -o simv1 -Mdir=csrc1 -distsim
 -debug_access+all | & tee vcs.log1 &
wait

mkdir S0
mkdir S1

cd S0
 ./simv0 -distsim=launch_server
 -distsim=client_id:0,num_sockets:2,config_file:../mapping,server_info:
 ../server.txt -l sim.log0 -ucli -i ./save0.tcl > client.log0 &
cd ..
cd S1
 ./simv1
 -distsim=client_id:1,num_sockets:2,config_file:../mapping,server_info:
 ../server.txt -l sim.log1 -ucli -i ./save1.tcl
 -distsim=save_file:save_snps > client.log1 &
cd ..

wait
```

To enable save in the `-distsim` mode, use the regular ucli save as highlighted in the above script. The `-distsim=save_file` option adds a user-defined prefix to the save file.

The following are the Tcl files for `client0` and `client1`:

#### *save0.tcl*

```
run 20
save save1
run 20
save save2
run 20
save save3
run
```

#### *save1.tcl*

```
run 20
save save1
run 20
save save2
run
```

For `client0`, the save image gets dumped at time 20, 40, and 60 with the `save1`, `save2`, and `save3` file names, respectively.

For `client1`, the save image gets dumped at time 20, 40, and 60 with the `save1`, `save2`, and `save_snps_dist_c1_60` file names, respectively.

## Restore

The Restore feature resumes simulation using the previously used Save Image, same as the Single Simulation with the following difference:

- The restore call in `distsim` flow is only supported at time zero from the UCLI file. Also, the `distsim` switch `distsim=restore` must be provided at runtime.

Following is the use model:

```
-distsim=restore
```

For example,

The sample run script for two client IDs (0 and 1) is as follows:

```
#!/bin/csh -fx

mkdir R0
mkdir R1

cd R0
 ./simv0 -distsim=launch_server
 -distsim=client_id:0,num_sockets:2,config_file:../mapping,server_info:
 ../server.txt -l sim.log0 -ucli -i ../restore0.tcl -distsim=restore >
 client.log0 &
cd ..

cd R1
 ./simv1
 -
 distsim=client_id:1,num_sockets:2,config_file:../mapping,server_info:../s
 erver.txt -l sim.log1 -ucli -i ../restore1.tcl -distsim=restore >
 client.log1 &
cd ..

wait
```

### Note:

To enable restore in the `distsim` mode, use regular ucli restore along with an additional `distsim` option, `-distsim=restore`.

The following are the Tcl files for `client0` and `client1`:

*restore0.tcl*

```
restore ../S0/save3
echo "Time after restore : $now"
run
```

*restore1.tcl*

```
restore ../S1/save_snps_dist_c1_60.SAVE
echo "Time after restore : $now"
run
```

The simulation gets restored from time 60 for both the clients.

## Limitations

The following are the limitations of the Save Restore feature:

- The Save Restore feature does not support the system task (`$save`, `$restart`, `-r` command line) and the Verdi interactive methods.
- Non-zero time restore is not supported, and only time0 restore is supported.
- Automatic restore of all clients is not supported. The user must restore all clients from UCLI restore.
- Restore requires an extra option, `-distsim=restore`, to pass.
- Multiple restores in single simulation are not supported.

## User Task to Print Class Objects

The task `$vcsdistsim_user_print_file()` is used in Verilog to print any string to the Distributed Simulation log files. It prints the single Verilog string passed as an argument in a Distsim data diagnostic file available in the current working directory along with other Distsim logs. Typically, it is useful to print class objects which are used for Testbench Data Transfer.

For example,

```
string s = obj.print();
$vcsdistsim_user_print_file(s);
```

## VCD Dump per Client

Using the VCD Dump per Client feature, you can dump the load and driver RTL connection signals of the Distributed Simulation Configuration File in the VCD format. You can load RTL connection signals into Verdi to view the waveform and can debug these files using the nCompare and vcdiff features.

Following is the use model:

```
-distsim=diag:dumpvars
```

By default, it dumps the VCD file per Client in the current simulation directory. You can also specify the path for dumping the VCD file using the following option:

```
-distsim=log_dir:<Directory_path>
```

## Bidirect Support Through Wired OR/AND

The Bidirect Support feature connects signals across different clients through the Wired OR and Wired AND gates behavior. Additional switches are not required to enable it, except adding the required connections in the configuration file in the specified format.

The following is the configuration file syntax:

```
<Gate Identifier> s<Client_ID>:<Signal> <=> s<Client_ID>:<Signal> <=>
s<Client_ID>:<Signal>
```

For example,

```
WOR s0:top.b1.c <=> s1:top.b2.c <=> s2:top.c3
WAND s1:top.a1 <=> s3:top.a3
```

The following are the expected behaviors:

- The signals connected through a gate are to be resolved based on the gate type specified in the configuration file in the Server Process.
  - WOR: If any signal is 1, all the connections become 1.
  - WAND: If any signal is 0, all the connections become 0.

The WOR and WAND gate behavior is shown in the following truth table:

| <b>A</b> | <b>B</b> | <b>WOR</b> | <b>WAND</b> |
|----------|----------|------------|-------------|
| 0        | 0        | 0          | 0           |
| 0        | 1        | 1          | 0           |
| 0        | X        | X          | 0           |
| 0        | Z        | X          | 0           |
| 1        | 0        | 1          | 0           |
| 1        | 1        | 1          | 1           |
| 1        | X        | 1          | X           |
| 1        | Z        | 1          | X           |

- The signals on the Client side need to drive the value sent by the server with the deposit nature of force. So, the local drivers can override the values.
- On the Client side, the value is fetched using the drivers of the connection signal and not directly from the signal. For example,

```
Wire [3:0] a;
Assign a = <rhs1> //Driver 1
Assign a = <rhs2> //Driver 2
```

Consider `a` as a part of the Bidirect gate in the `distsim` configuration file, then the value of `a` is `Resolve (value of Driver 1, Value of Driver 2)`.

- The `Resolve` works in the same way as the VCS XZ resolution:

| A | B | Resolved Value |
|---|---|----------------|
| 0 | 0 | 0              |
| 0 | 1 | X              |
| 0 | X | X              |
| 0 | Z | 0              |
| 1 | 0 | X              |
| 1 | 1 | 1              |
| 1 | X | X              |
| 1 | Z | 1              |

For example,

```
wire [1:0] a;
reg [1:0] b;
reg [1:0] c;
reg [1:0] d;

assign a = b + c; //First Driver
assign a = d; //Second Driver

WOR s0:top.a <=> s1:top.a <=> s2:top.a
```

| Signal       | Client 0 | Client 1 | Client 2 | Server |
|--------------|----------|----------|----------|--------|
| Sync Point   |          |          |          |        |
| First Driver | 00       | 00       | 10       |        |

| Signal          | Client 0 | Client 1 | Client 2 | Server   |
|-----------------|----------|----------|----------|----------|
| Second Driver   | 00       | 00       | 10       |          |
| Top.a GET       | 00       | 00       | 10       |          |
| Top.a Drivers   | 00       | 00       | 10       | 10 (WOR) |
| Top.a PUT       | 10       | 10       | 10       |          |
| Next Sync Point |          |          |          |          |
| First Driver    | 00       | 00       | 00       |          |
| Second Driver   | 00       | 00       | 00       |          |
| Top.a GET       | 10       | 10       | 00       |          |
| Top.a Drivers   | 00       | 00       | 00       | 00       |
| Top.a PUT       | 00       | 00       | 00       |          |
| Next Sync Point |          |          |          |          |
| First Driver    | 11       | 10       | 00       |          |
| Second Driver   | 01       | 10       | 00       |          |
| Top.a GET       | X1       | 10       | 00       |          |
| Top.a Drivers   | X1       | 10       | 00       | 11       |
| Top.a PUT       | 11       | 11       | 11       |          |

## Limitations

The following are the limitations of the Bidirect Support Through Wired OR/AND feature:

- Supports only WOR and WAND gates.
- Supports wires and logic used as wire.
- Supports only Scalar and Full vectors.
- Does not support Registers, Memories, Bit Select, Part Select, and Memory Word.
- Distsim does not support tran gate drivers for the Bidirect connections. VCS displays a warning message on the first occurrence of an active tran gate driver.

## Include Syntax in Configuration File

The Include Syntax in Configuration File feature enables a top level connectivity file to include lower level connectivity file. The top level connectivity file is passed with the following option: `-distsim=config_file:<top level connectivity file>`

## Configuration File Syntax

Use `include` or '`include`' identifier to specify the configuration file with the full or relative file path with respect to the top level connectivity file. You must use double quotation marks to specify the configuration file.

For example,

```
-distsim=config_file:top.connectivity.txt
> cat top.connectivity.txt
include "sub1.connectivity.txt"
//RTL connections
...
> cat sub1.connectivity.txt
//RTL connections
include "sub2.connectivity.txt"
```

and so on...

## Server Launch as Independent Process

You can use the `-distsim=launch_server` runtime flag to launch the Server Process from the Master Client, Client 0.

Alternatively, you can launch the Server Process as an independent process using the following points:

- No client should pass `-distsim=launch_server`
- Server Process Executable: `$VCS_HOME/bin/simv.distsim`
- Server Process to pass all the `-distsim` runtime flags as passed by Client 0 in this mode.

For example,

```
$VCS_HOME/bin/simv.distsim
-distsim=num_sockets:3,config_file:../mapping,server_info:../server.txt
> ../server.log &
simv0
-distsim=client_id:0,num_sockets:3,config_file:../mapping,server_info:
../server.txt -l ../sim.log0 > ../client.log0 &
```

```
simv1
-distsim=client_id:1,num_sockets:3,config_file:../mapping,server_info:
..//server.txt -l ..//sim.log1 > ..//client.log1 &
simv2
-distsim=client_id:2,num_sockets:3,config_file:../mapping,server_info:
..//server.txt -l ..//sim.log2 > ..//client.log2 &
```

---

## HDL Access Across Simvs from Verilog Source

The HDL Access feature provides access to signals across simvs in distributed simulation from the Verilog source. Using this feature, you can deposit/force/release from other simvs in distributed simulation. It supports scalars, vectors, and selects.

The following is the syntax:

```
VCSDISTSIM_HDL_FORCE(<Label String>, <signal Name>, <value Expr>,
<isForce>)
VCSDISTSIM_HDL_RELEASE(<Label String> , <signal Name>)
```

Where,

- **<Label String>** is the unique string label for the destination client.
- **<signal Name>** is the hierarchical path for the destination signal.
- **<value Expr>** is any valid Verilog expression.
- **<isForce>** is 1 for force freeze and 0 for deposit.

You must have a Label String mapping to the client ID in the `distsim` mapping file as shown below:

```
hdl_label:<Label String>:<client Id>
```

For example, consider label string mapping as follows: `hdl_label:CH2:2`

```
hdl_label:CH3:3
`VCSDISTSIM_HDL_FORCE("CH2","socket2.a2", a1, 0);
#10 `VCSDISTSIM_HDL_RELEASE("CH3", socket3.a2);
```

The following is the sample Verilog snippet demonstrating the usage of HDL access:

```
initial
begin
#1;
 a22 = 100;
 b22[31:0] = 32'h33;
 b22[63:32] = 32'h77;
 c22 = 1;
 `VCSDISTSIM_HDL_FORCE("mysim0", "cpu_1.a11", a22, 1);
 `VCSDISTSIM_HDL_FORCE("mysim0", "cpu_1.b11", b22, 0);
#5;
```

```
`VCSDISTSIM_HDL_RELEASE("mysim0", "cpu_1.a11");
end
```

The label string mapping for client 0 and 1 are as follows:

```
master_sync_signal: s0:cpu_1.clk_1
master_sync_signal: s1:cpu_2.clk_1
hdl_label:mysim0:0
hdl_label:mysim1:1
```

Where, `s0` and `s1` are client IDs 0 and 1 respectively. HDL label `mysim0` is linked to client ID 0 and `mysim1` is linked to client ID 1.

The sample run script for two client IDs (0 and 1) is as follows:

```
#!/bin/csh -fx

rm -rf simv* csrc* S*
vcs cpu_1.v -sverilog -full164 -o simv0 -Mdir=csrc0 -distsim
 -debug_access+f| & tee vcs.log0 &
vcs cpu_2.v -sverilog -full164 -o simv1 -Mdir=csrc1 -distsim
 -debug_access+f| & tee vcs.log1 &

wait

mkdir S0
mkdir S1

cd S0

$VCS_HOME/bin/simv.distsim
-
distsim=num_sockets:2,config_file:../mapping,server_info:../server.txt
 /server.log &

..../simv0
 -distsim=client_id:0,num_sockets:2,config_file:../mapping,server_info:
 ../server.txt -l/sim.log0 >/client.log0 &
cd ..

cd S1
..../simv1
 -distsim=client_id:1,num_sockets:2,config_file:../mapping,server_info:
 ../server.txt -l/sim.log1 >/client.log1 &
cd ..

wait
```

The script launches the server process for client ID 0 and 1. In the above example,

- `VCSDISTSIM_HDL_FORCE` forces the value of signal `a22` at current time to signal `a11` of `mysim0 simv` which has client id 0 at the coming nearest `distsim` sync point.
- `VCSDISTSIM_HDL_FORCE` deposits the value of signal `b22` at current time to signal `b11` of `mysim0 simv` which has client id 0 at the coming nearest `distsim` sync point.
- `VCSDISTSIM_HDL_RELEASE` releases the signal `cpu_1.a11` of client id 0 at the coming nearest `distsim` sync point.

## Distsim Release Recv Call for TB Data Transfer

When you use `'VCSDISTSIM_TB_RECV_RELEASE(Channel_Name)` to release any existing TB Data Recv Call '`VCSDISTSIM_TB_RECV("Channel_Name", bit_array)` in the design, then the following things happen:

- Release the wait in the Recv call for the specified channel.
- When there is no active Recv call for the specified channel, then no action is performed.
- When it is called in the same sync cycle, it takes precedence over Recv call.

## TB Data Open Channels

The channel used in TB Data Send and Recv calls are configured during runtime using runtime flags or the Distsim connectivity file. However, if any channel is left open, it is honored.

### Open Channel For Sender

The open channel for sender implies that there is no receiver channel specified for it. The call made by sender, `VCSDISTSIM_TB_SEND`, issues a warning, ignores the data to be sent, and continues the simulation.

### Open Channel For Receiver

The open channel for receiver implies that there is no sender channel specified for it. The call made by receiver, `VCSDISTSIM_TB_RECV`, keeps waiting in the Recv call, issues a warning, and blocks the corresponding TB Recv thread.

#### Note:

You can use `VCSDISTSIM_TB_RECV_RELEASE` to unblock the thread corresponding to that channel.

## Distributed Simulation Profile

The Distributed Simulation Profile feature dumps a time profile at the end of simulation irrespective of the exit status. For each Client, the profile is dumped at the end of the simulation log. For the Server process, the profile is dumped into a separate file, `cygnus.profile.txt`, created in the working directory of the master client.

The Distributed Simulation Profile feature compares the synchronization time taken by different clients with the elapsed time for simulation. By default, it dumps profiles for all the processes.

The time profile contains the following information:

- Total elapsed time: It is total real elapsed time by the Client.
- Initial time: It is the time spent from beginning till first sync.
- Active simulation time: It is the time elapsed in running simulation besides distsim processing.
- Imbalance time: It is the total time spent while waiting for other clients to reach the distributed simulation sync points.
- Process time: It is the time spent in fetching and sending the data.
- Wait time: It is the time spent in waiting for messages as well as in network calls.

The following example displays the performance summary of a client with 1 as <Client id>:

```

Performance Summary for Client: 1
Safe Finish with PID: 25325
Total Sync Points: 2001

Total Elapsed Time: 15.00s (15104611us)
- Init Time: 1.00s
- Active Sim Time: 10.47s (Avg: 5231us)
- Process Time: 1.82s (Avg: 908us)
- Wait Time: 1.82s (Avg: 908us)
Imbalance Time: 0.07s (Avg: 34us)

```

In the example, the `Safe Finish with PID` field displays the Process ID and the `Total sync points` field displays the total number of Syncs based on Master Sync Signal or interval.

## Configuration File Checker Utility

The Configuration File Checker Utility feature provides a separate utility to check the configuration file sanity that avoids simulation to run into errors related to the configuration file.

The configuration file checker utility provides the following checks:

- Syntax of the configuration file so that it is compatible with the VCS Distsim format.
- Existence of all the specified signals in the design.

The following are the requirements to perform the checks:

- Enabled compilation with KDB
- VERDI\_HOME set

The following arguments are required for the checker utility:

- KDB database directory path (`-distsim=dbdir:<dbdir path>`)
- Configuration file path (`-distsim=config_file:<file path>`)
- Total number of Clients (`-distsim=num_sockets:<number of clients>`)
- Client ID only when you need to check a particular client (`-distsim=client_id:<id>`) (optional)

For example,

```
$VCS_HOME/bin/simv.distsimchecker
-distsim=config_file:mapping
-distsim=dbdir:simv0.daidir
-distsim=num_sockets:3
```

The output of the checker is displayed in the terminal at the end.

## Specifying User-Defined TCP Port for Server Launch

All the ports in a network might not be open to establish a TCP/IP connection due to network security reasons. The User Defined TCP Port feature passes the required port in the command line to the Master Simv.

### Note:

The user defined TCP port feature is required only for the Server, when launched separately, or for the Master Simv to launch the server. It is ignored for rest of the Clients.

The following is the runtime option:

```
-distsim=port:<TCP port number>
```

## Key Points to Note

- The configuration allows only single set of distributed simulation to run on a unique port.
- You need to provide different unique port numbers during a regression or you need to run multiple test cases simultaneously while using the `-distsim=port:<TCP port number>` option.

## Specifying Early Simulation Finish Mechanism Across Simvs

The Finish Order feature controls the finish behavior in a distributed simulation setup. The default behavior in case of Safe Finish is to wait for all the Clients to finish.

With the `-distsim=finish:first` option, the behavior is to finish when any client finishes first. When a client finishes/exists, the rest of the clients exit their simulation at the next sync point instead of continuing till their end of simulation/exit.

## Cygnus PDF Profile Summary

The Cygnus PDF Profile Summary feature dumps a PDF file containing the distsim simulation graphical summary for the elapsed time and waiting time with respect to the design time.

By default, the PDF file is dumped by the server at the end of the simulation. You can disable this feature using the `-distsim=profile:off` option.

## TB Sync Comp Class

The TB Sync Comp Class feature is an alternative way to sync all UVM phases without writing a TB Phase Sync macro. You need to include the following file in the testbench containing definition for the `dist_tbsync_comp` class, a derived class of `uvm_component`:

```
$VCS_HOME/etc/distsim/vcsdistsim_tbsync_comp.svp
```

The TB Sync Comp class can be instantiated to sync all the UVM phases automatically.

For example,

```
include "vcsdistsim_tbsync_comp.svp"
dist_tbsync_comp comp0;
function void build_phase(uvm_phase phase);
super.build_phase(phase);
```

```
comp0 = dist_tbsync_comp::type_id::create("comp0",this);
endfunction
```

## Packed Structure Support

The Packed Structure Support feature allows you to use the connectivity signals of the packed structure datatypes. It can be used for the RTL connections and the HDL Force.

To enable the Packed Structure Support feature, use the following option:

```
-distsim=struct
```

## Key Points to Note

- VCS supports the packed full structures and their members.
- Unpacked structures are not supported.
- Nested packed structures are supported.
- If there is a width mismatch between the load and the driver side, the output is similar to that in the Verilog assignments.

## Constant Support in the Configuration File

The Constant Support in the Configuration File feature drives any supported RTL signal using the Verilog constants. It supports hexadecimal, binary, and decimal constants.

For example,

```
s0:top.a = 1'b1
s1:top.b = 8'hzx
s2:top.c = 2'd10
s3:top.d = 25
```

## Support for Multiple TB Data Recv Requests

The Support for Multiple TB Data Recv Requests feature allows you to attend Multiple Recv Requests at a time on the same TB Channel. This feature is used when you are expecting to receive the TB Data in the parallel Verilog threads simultaneously. By default, the Support for Multiple TB Data Recv Requests feature is disabled.

To enable this feature, you can use the following option:

```
-distsim=multiple_same_channel_req
```

## Compile Time Configuration File Support

Using the Compile Time Configuration File Support feature for the limited debug capabilities, you can avoid adding the global `debug_access` debug capabilities for distsim. Instead, you can pass the distsim configuration file at compile time which enables the required debug capabilities like deposit/force/release only on the relevant connectivity signals.

When you use `-debug_access+<options>` along with the `-distsim=config_file` at compile time, it results in the union of all the capabilities.

The following is the compile option:

```
-distsim=config_file:<Distsim Config File Path>
-distsim=client_id:<Client ID> // For Heterogeneous systems. For Homogeneous systems, you can avoid this option.
```

For example,

```
#!/bin/csh -fx

rm -rf simv* csrc* S*

vcs w0.v -sverilog -full64 -o simv0 -Mdir=csrc0 -distsim
-distsim=config_file:mapping.txt
-distsim=client_id:0 | & tee vcs.log0
vcs w1.v -sverilog -full64 -o simv1 -Mdir=csrc1 -distsim
-distsim=config_file:mapping.txt -distsim=client_id:1 | & tee vcs.log1

mkdir S0
mkdir S1

cd S0
../simv0
-distsim=client_id:0,num_sockets:2,config_file:../mapping.txt,server_info:../server.txt -l ../sim.log0
-distsim=struct -distsim=launch_server > ../client.log0 &
cd ..

cd S1
../simv1 -distsim=client_id:1,num_sockets:2,
config_file:../mapping.txt,server_info:../server.txt -l ../sim.log1
> ../client.log1 &
cd ..
```

The above example is of a Heterogeneous system which consists of the distsim configuration file (`-distsim=config_file:mapping.txt`), and the client ID, (`-distsim=client_id:0 or 1`), options to enable the required debug capabilities for the design and save runtime. The `-distsim=struct` option enables the packed structure support feature.

## Key Points to Note

- The Compile Time Configuration File Support feature is supported only for the RTL Connections and Bidirect Support, and you can avoid adding the global `debug_access` option.
  - The Compile Time Configuration File Support feature is not supported for HDL Force/Release, TB Phase Sync, and TB Data Transfer. Therefore, you need to add the global `debug_access` option. For example, `-debug_access+class` and `-debug_access+f+class`.
- 

## Distsim Flow Without Shared File System

When the common file system is not available to provide the Server Info File, the Distsim Flow Without Shared File System feature is used to run Distsim flow. Therefore, instead of passing Server Info File in all the clients commands, you need to pass the IP address of the Server and the TCP port through the runtime arguments. The IP address can be a host name or an IPv4 address.

When the provided Server IP address is not correct, the client processes get timeout and exit. The default timeout is 60 minutes and you can use the following option to specify time in minutes:

```
-distsim=timeout:<time in minutes>
```

The following is the use model:

```
-distsim=ip:<> -distsim=port:<>
```

### Examples

In the following examples, instead of using the `server_info` runtime argument, the `-distsim=ip:<>` option is used to pass the IP address of the server and `-distsim=port:<>` option is used to pass the TCP port:

- Launching the server using `-distsim=launch_server`

```
mkdir S0; cd S0
vcs ..\w0.v -sverilog -full64 -distsim -distsim=client_id:0
-distsim=config_file:..\mapping.txt -o simv0 -Mdir=csrc0 |& tee
vcs.log.distsim.0 &; cd ..
mkdir S1; cd S1
vcs ..\w1.v -sverilog -full64 -distsim -distsim=client_id:1
-distsim=config_file:..\mapping.txt -o simv1 -Mdir=csrc1 |& tee
vcs.log.distsim.1 &
cd ..
wait
cd S0
./simv0 -distsim=client_id:0,num_sockets:2,config_file:..\mapping.txt
-distsim=port:8080
```

```
-distsim=ip:10.192.194.52 -distsim=launch_server -l sim0.log >
 client.log.0 &
cd ..
cd S1
./simv1 -distsim=client_id:1,num_sockets:2,config_file:../mapping.txt
 -distsim=port:8080
-distsim=ip:127.0.0.1 -l sim1.log > client.log.1 &
cd ..
```

- **Launching the server manually**

```
mkdir S0; cd S0
vcs ..//w0.v -sverilog -full64 -distsim -distsim=client_id:0
 -distsim=config_file:../mapping.txt -o simv0 -Mdir=csrc0 |& tee
 vcs.log.distsim.0 &; cd ..
mkdir S1; cd S1
vcs ..//w1.v -sverilog -full64 -distsim -distsim=client_id:1
 -distsim=config_file:../mapping.txt -o simv1 -Mdir=csrc1 |& tee
 vcs.log.distsim.1 &; cd ..
wait
cd S0
$VCS_HOME/bin/simv.distsim -full64 -distsim=num_sockets:2
 -distsim=config_file:../mapping.txt -distsim=port:8080
 -distsim=ip:localhost > server.log &
./simv0 -distsim=client_id:0 -distsim=num_sockets:2
 -distsim=config_file:../mapping.txt -distsim=port:8080
-distsim=ip:localhost -l sim0.log > client.log.0 &; cd ..
cd S1
./simv1 -distsim=client_id:1 -distsim=num_sockets:2
 -distsim=config_file:../mapping.txt -distsim=port:8080
-distsim=ip:localhost -l sim1.log > client.log.1 &; cd ..
```

In the above example, localhost is the local computer with the loop back address, 127.0.0.1. You can get the IPv4 address or hostname using the following unix command: hostname -i

## Early Finish Behavior

When a client finishes early, the Early Finish Behavior feature allows you to control the finish time behavior of the RTL signals specified in the Distsim configuration file.

By default, VCS releases all the RTL signals of the Distsim configuration file in rest of the clients driven by the finished client and in the absence of the Distsim driver, VCS allows the local drivers to take over. However, using the Early Finish Behavior feature, you can choose not to release RTL signals and freeze them to their last forced values.

The following is the use model:

```
-distsim=finish:freeze
```

For example,

```
vcs test.v -sverilog -full164 -distsim
-distsim=config_file:mapping.txt -o simv

vcs test.v -sverilog -full164 -distsim
-distsim=config_file:mapping.txt -o simvf

./simvf
-distsim=client_id:0,num_sockets:2,config_file:mapping.txt,server_info:s
erver.txt -distsim=launch_server -l sim0.log
-distsim=finish:freeze

./simv
-distsim=client_id:1,num_sockets:2,config_file:mapping.txt,server_info:s
erver.txt -l sim1.log &
```

---

## Limitations

The following are the limitations of VCS Distributed Simulation:

- The maximum number of clients cannot exceed 16.
- RTL connections, HDL Across Simvs, and Bidirect Support do not support the following data types: Memory/Real/Time/MemWord/Dynamic types.
- Jump backward or forward of UVM Phases is not supported for TB Phase Syncing; and cross UVM phase sync (synchronizing a phase of one client with another phase of different clients) is not supported for TB Phase Syncing.
- Multiple UVM components syncing to different phases is not supported for TB Phase Syncing.
- The Distributed Simulation Profile feature does not dump profile for a process that crashes.
- The Compile Time Configuration File Support for the limited debug Caps feature is not supported for HDL/TB Phase/TB Data.

---

## Quick Reference: VCS Distributed Simulation Options

Except the `-distsim=launch_server` and `-distsim=client_id:<id>` options, you must pass the following options where the server is launched, that is where `-distsim=launch_server` is passed. If the Server Process is launched independently, pass the following options on the Server Process Executable (`$VCS_HOME/bin simv.distsim`).

| Option                                        | Description                                                       | Requirement                                                                                                                   |
|-----------------------------------------------|-------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------|
| -distsim=help                                 | Lists all -distsim options along with the description.            | As quick reference.                                                                                                           |
| -distsim=num_sockets:<total_Clients>          | Specifies total number of clients in the set up.                  | The simulation can only proceed when all the clients are connected as they are mutually dependent.                            |
| -distsim=client_id:<id>                       | Specifies unique ID for simv.                                     | Clients are distinguished based on <code>id</code> to parse the configuration file.                                           |
| -distsim=launch_server                        | Launches the Server Process.                                      | Allows you to designate one of the simv to launch the Server Process internally. Specify unique server file name for new run. |
| -distsim=server_info:<file_path>              | Specifies a file to dump server information.                      | Allows you to establish TCP connection among all the clients through a server.                                                |
| -distsim=config_file:<file_path>              | Specifies configuration file.                                     | Configuration file which holds connectivity and other information.                                                            |
| -distsim=diag:off                             | Disables default tool diagnostics.                                | Optional. You can use this option if you want the tool to run silently.                                                       |
| -distsim=diag:data                            | Enables dumping of a file with all RTL/testbench Value exchanges. | Optional. You can use this option for debugging.                                                                              |
| -distsim=log_dir:<dir_path>                   | Specifies the path to dump Distsim related logs.                  | Optional. You can use this option if you want to dump Distsim logs at a different location.                                   |
| -distsim=replay                               | Starts the Replay Mode.                                           | Optional. You can use this option for debugging by replaying any Client in standalone mode.                                   |
| -distsim=replay_file:<file path of replay db> | Specifies the file path of the replay db.                         | Optional. You can use this option for debugging to pass replay db name.                                                       |

| Option                                          | Description                                                                   | Requirement                                                                                                                  |
|-------------------------------------------------|-------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------|
| -distsim=replay_force:<file path of force file> | Specifies the file path of the Force file.                                    | Optional. You can use this option for debugging in replay mode by forcing connection signals using force configuration file. |
| -distsim=dumpreplay                             | Dumps the replay db in the current simulation directory.                      | Optional. You can use this option for dumping the replay db for any Client in standalone mode in future.                     |
| -distsim=replaybatch:<SIZE_OF_BATCH>            | Reads the replay dbs in batches.                                              | Optional. You can use this option to save the runtime memory by avoiding loading the entire replay file at one go.           |
| -distsim=save_file:<user_save_file_prefix>      | A runtime switch that specifies <User_save_file_prefix>                       | Optional. You can use this option in the Save Restore mode to specify prefix for the Save image file name.                   |
| -distsim=restore                                | Starts the Restore feature.                                                   | Optional. You can use this option in the Save Restore mode to enable restore.                                                |
| -distsim=diag:dumpvars                          | Dumps the VCD file per Client in the current simulation directory.            | Optional. You can use this option for debugging by loading connection signals into Verdi.                                    |
| -distsim=port:<TCP port number>                 | Specifies the TCP port number.                                                | Optional. You can use this option for passing the required port in the command line to the Master Simv.                      |
| -distsim=dbdir:<dbdir path>                     | Specifies the file path of the KDB database directory.                        | Mandatory. You can use this option only while using Configuration File Checker Utility.                                      |
| -distsim=finish:first                           | Specifies simulation exit for all clients when any one client exits/finishes. | Optional. You can use this option to control the finish behavior in a Distributed Simulation setup.                          |
| -distsim=struct                                 | Enables the Packed Structure Support feature.                                 | You can use this option to use the connectivity signals of the packed structure datatypes.                                   |

| Option                             | Description                                                                                                                                               | Requirement                                                                                                                                                                                 |
|------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| -distsim=multiple_same_channel_req | Enables Multiple TB Data Recv Requests feature.                                                                                                           | You can use this option to attend Multiple Recv Requests at a time on the same TB Channel.                                                                                                  |
| -distsim=timeout:<time in minutes> | Specifies the time after which the client processes timeout and exit.                                                                                     | You can use this option to reset the default timeout which is 60 minutes.                                                                                                                   |
| -distsim=ip:<IPv4 address>         | Specifies the IPv4 address or the hostname of the server. To get the hostname or the IP address, use the following unix command: <code>hostname -i</code> | You must use this option when the common file system is not available to provide Server Info File. This option must be used along with <code>-distsim=port:&lt;TCP port number&gt;</code> . |
| -distsim=finish:freeze             | Freezes RTL signals to their last forced values.                                                                                                          | You can use this option to control the finish time behavior in the Distsim setup when a client finishes early.                                                                              |

# 11

## Performance Tuning

---

VCS delivers the best performance during both compile-time and runtime by reducing the size of the simulation executable, and the amount of memory consumed for compilation/elaboration and simulation. By default, it is optimized for the following types of designs:

- Designs with many layers of hierarchy
- Gate-level designs
- Structural RTL-level designs - Using libraries where the cells are RTL-level code
- Designs with extensive use of timing such as delays, timing checks, and SDF back annotation, particularly to INTERCONNECT delays

However, depending on the phase of your design cycle, you can fine-tune VCS for a better compile-time and runtime performance.

This chapter describes the following sections:

- Analysis-time Performance

During analysis, you can analyze all of both Verilog and VHDL files in a single command line. For example, perform the following to analyze Verilog files:

```
% vlogan file1.v file2.v file3.v
```

For additional information, see the section entitled, [Analysis](#) .

- Compile-time Performance

Compile-time performance plays a very important role when you are in the initial phase of your design development cycle. In this phase, you may want to modify and recompile the design to observe the behavior. Since, this phase involves lot many recompiling cycles, achieving a faster compilation is important. For additional information, see the section entitled, [Compile-time Performance](#).

- Runtime Performance

Runtime performance is important in regression phase or in the final phase of the design development cycle. For additional information, see the section entitled, [Runtime Performance](#).

- Obtaining VCS Consumption of CPU Resources

You can now capture the CPU resource statistics for compilation and simulation using the `-reportstats` option. For more information, see [Obtaining VCS Consumption of CPU Resources](#).

- Dumping Design Statistics

VCS generates design statistics covering various construct usages (such as number of modules, number of instances, number of signals and so on) and the design size. For more information, see [Dumping Design Statistics](#).

---

## Compile-time Performance

You can improve compile-time performance in the following ways:

- [Incremental Compilation](#)
  - [Compile Once and Run Many Times](#)
  - [Parallel Compilation](#)
  - [Improving VCS Compile Performance and Capacity](#)
- 

### Incremental Compilation

During compilation/elaboration, VCS builds the design hierarchy. By default, when you recompile the design, VCS compiles only those design units that have changed since the last compilation/elaboration. This is called incremental compilation.

The incremental compilation feature is the default in VCS. It triggers recompilation of design units under the following conditions:

- Changes in the command-line options
- Change in the target of a hierarchical reference
- Change in the ports of a design unit
- Change in the functional behavior of the design
- Change in a compile-time constant such as a parameter/generic

The following conditions do not cause VCS to recompile a module:

- Change of time stamp of any source file
- Change in file name or grouping of modules in any source file

- Unrelated change in the same source file
- Non-functional changes such as comments or white space

## Compile Once and Run Many Times

The VCS use model is devised in such a way that you can create a single binary executable and execute it many times avoiding the elaboration step for all but the first run. For information on the VCS use model, see [Using the Simulator](#).

For example, you can use this feature in the following scenarios:

- Use VCS runtime features, like passing values at runtime, to modify the design, and simulate it without re-compiling or re-elaborating. For information on runtime features, see Chapter [Simulating the Design](#).
- Run the same test with different seeds.
- Create a softlink of the executable and the .daidir or .db.dir directory in a different directory, to run multiple simulations in parallel.

## Parallel Compilation

You can improve the compile-time performance by specifying the number of parallel processes VCS can launch for the native code generation phase of the compilation/elaboration. You should specify this using the compile-time option `-j<num_of_processes>`, as shown below:

```
% vcs -j<num_of_processes> [options] top_entity/module/config
```

### Note:

Parallel compilation applies only for the Verilog portion of the design.

For example, the following command line forks off two parallel processes to generate a binary executable:

```
% vcs -j2 top
```

## Improving VCS Compile Performance and Capacity

For gate-level simulation (GLS) designs, all connected networks are placed in one elaboration module during VCS elaboration. Because of the strong connectivity in GLS design, it always generates a very large elaboration module. If the size (nodes or gates) of the elaboration module exceeds 32-bit, it results in capacity issues with GLS design that uses the `-hsopt=gates` option.

VCS addresses the capacity issues of VCS elaboration by stopping global optimizations and by using partitioning in `vcselab` stage. Partitioning reduces the peak size of the elaboration module and balances the size of the elaboration modules. Thus, it helps in reducing the compile performance overhead.

## Use Model

In GLS design, partitioning is inserted using following options.

- `-hsopt=elabpart` - When you use this option, VCS uses partitioning of modules and connected networks.
- `-hsopt=j` - When you use this option, VCS speeds up the `vcselab` phase by compiling the partitions in parallel using multiple cores on a machine. By default, VCS automatically decides on the number of cores to be used for parallel compilation.

You can also specify the number of cores that VCS uses for parallel compilation using the `-hsopt=j<N>` option, where `N` is the number of cores.

- You can also insert partitioning manually using the `+optconfigfile` configuration file by specifying the module name for which partitioning is required to be inserted. The configuration file contains the following:

```
module {<module name>} {partition};
```

## Runtime Performance

VCS runtime performance is based on the following:

- Coding Style (see *Modeling and Coding Style Guide*)
- Access to the internals of your design at runtime, using PLIs, UCLI, debugging using GUI, dumping waveforms, and so on

### Note:

- Optimizations for the runtime performance improvements can be enabled using the compile-time option, `-Xkeyopt=rtopt`.
- Optimizations related to the UVM operations for the regular expressions can be enabled using the compile-time option, `-Xkeyopt=mvu`.

This section describes the following to improve the runtime performance:

- [Using Radian Technology](#)
- [Improving Performance When Using PLIs](#)
- [Enabling TAB File Capabilities in UCLI Using `-debug\_access`](#)

---

## Using Radian Technology

VCS Radian Technology applies performance optimizations to the Verilog portion of your design while VCS compiles your Verilog source code. These Radian optimizations improve the simulation performance of all types of designs from behavioral, RTL to gate-level designs. Radian Technology particularly improves the performance of functional simulations where there are no timing specifications or when delays are distributed to gates and assignment statements.

## Compiling With Radian Technology

Radian Technology optimizations are not enabled by default. You enable them using the compile-time options:

`+rad`

Specifies using Radian Technology

These optimizations are also enabled for SystemVerilog part of the design.

`+optconfigfile`

Optional. Specifies applying Radian Technology optimizations to part of the design using a configuration file as described in the following section.

## Applying Radian Technology to Parts of the Design

The configuration file enables you to apply Radian optimizations selectively to different parts of your design. You can enable or disable Radian optimizations for all instances of a module, specific instances of a module, or specific signals.

You specify the configuration file with the `+optconfigfile` compile-time option. For example:

`+optconfigfile+file_name`

### Note:

The configuration file is a general purpose file that has other purposes, such as specifying ACC write capabilities. Therefore, to enable Radian Technology optimizations with a configuration file, you must also include the `+rad` compile-time option.

## The Configuration File Syntax

The configuration file contains one or more statements that set Radian optimization attributes, such as enabling or disabling optimization on a type of design object, such as a module definition, a module instance, or a signal.

The syntax of each type of statement is as follows:

```
module {list_of_module_identifiers} {list_of_attributes};
```

or

```
instance {list_of_module_identifiers_and_hierarchical_names}
{list_of_attributes};
```

or

```
tree [(depth)] {list_of_module_identifiers} {list_of_attributes};
```

Usage:

module

Keyword that specifies that the attributes in this statement apply to all instances of each module in the list, specified by module identifier.

list\_of\_module\_identifiers

A comma separated list of module identifiers enclosed in curly braces: { }

list\_of\_attributes

A comma separated list of Radiant optimization attributes enclosed in curly braces: { }

instance

Keyword that specifies that the attributes in this statement apply to:

- All instances of each module in the list specified by module identifier.
- All module instances in the list specified by their hierarchical names.
- The individual signals in the list specified by their hierarchical names.

list\_of\_module\_identifiers\_and\_hierarchical\_names

A comma separated list of module identifiers, hierarchical names of module instances, or signals enclosed in curly braces: { }

Follow the Verilog syntax for signal names and hierarchical names of module instances.

tree

Keyword that specifies that the attributes in this statement apply to all instances of the modules in the list, specified by module identifier, and also apply to all module instances hierarchically under these module instances.

depth

An integer that specifies how far down the module hierarchy, from the specified modules, you want to apply Radiant optimization attributes. You can specify a negative value. A negative value specifies descending to the leaf level and counting up levels of the hierarchy to apply these attributes. This specification is optional. Enclose this specification in parentheses: ()

The valid Radiant optimization attributes are as follows:

noOpt

Disables Radiant optimizations on the module instance or signal.

noPortOpt

Prevents port optimizations such as optimizing away unused ports on a module instance.

Opt

Enables all possible Radiant optimizations on the module instance or signal.

PortOpt

Enables port optimizations such as optimizing away unused ports on a module instance.

Statements can use more than one line and must end with a semicolon.

Verilog style comments characters /\* comment \*/ and // comment can be used in the configuration file.

## Configuration File Statement Examples

The following are examples of statements in a configuration file.

### Module Statement Example

```
module {mod1, mod2, mod3} {noOpt, PortOpt};
```

This module statement example disables Radiant optimizations for all instances of modules mod1, mod2, and mod3, with the exception of port optimizations.

### Multiple Module Statement Example

```
module {mod1, mod2} {noOpt};
module {mod1} {Opt};
```

In this example, the first module statement disables radiant optimizations for all instances of modules mod1 and mod2 and then the second module statement enables Radiant optimizations for all instances of module mod1. VCS processes statements in the order in which they appear in the configuration file so the enabling of optimizations for instances of module mod1 in the second statement overrides the first statement.

### Instance Statement Example

```
instance {mod1} {noOpt};
```

In this example, `mod1` is a module identifier, so the statement disables Radiant optimizations for all instances of `mod1`. This statement is the equivalent of:

```
module {mod1} {noOpt};
```

### Module and Instance Statement Example

```
module {mod1} {noOpt};
instance {mod1.mod2_inst1.mod3_inst1, mod1.mod2_inst1.reg_a} {noOpt};
```

In this example, the module statement disables Radiant optimizations for all instances of module `mod1`.

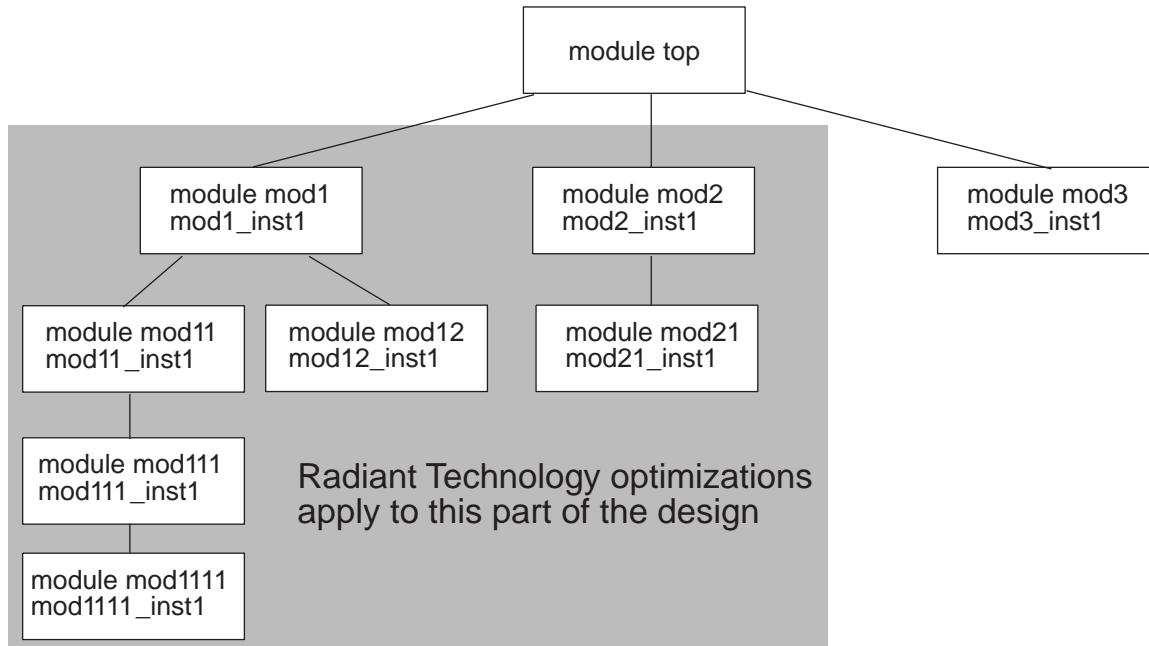
The instance statement disables Radiant optimizations for the following:

- Hierarchical instance `mod1.mod2_inst1.mod3_inst1`
- Hierarchical signal `mod1.mod2_inst1.reg_a`

### First Tree Statement Example

```
tree {mod1,mod2} {Opt};
```

This example is for a design with the following module hierarchy:

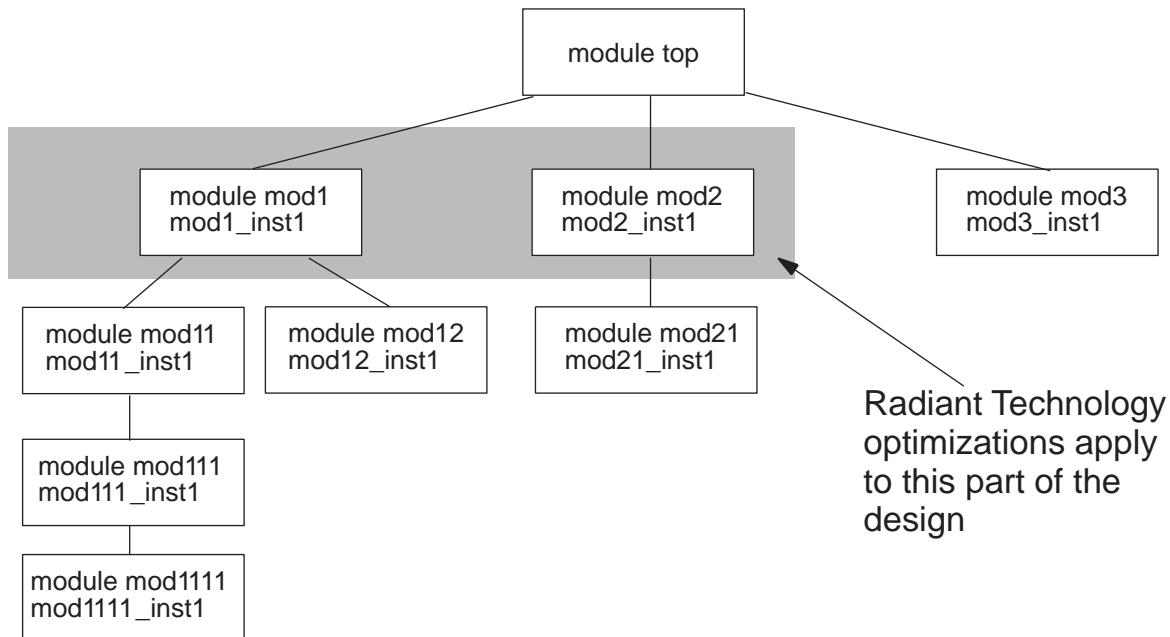


The statement enables Radiant Technology optimizations for the instances of modules `mod1` and `mod2` and for all the module instances hierarchically under these instances.

### Second Tree Statement Example

```
tree (0) {mod1,mod2} {Opt};
```

This modification of the previous tree statement includes a depth specification. A depth of 0 means that the attributes apply no further down the hierarchy than the instances of the specified modules, `mod1` and `mod2`.



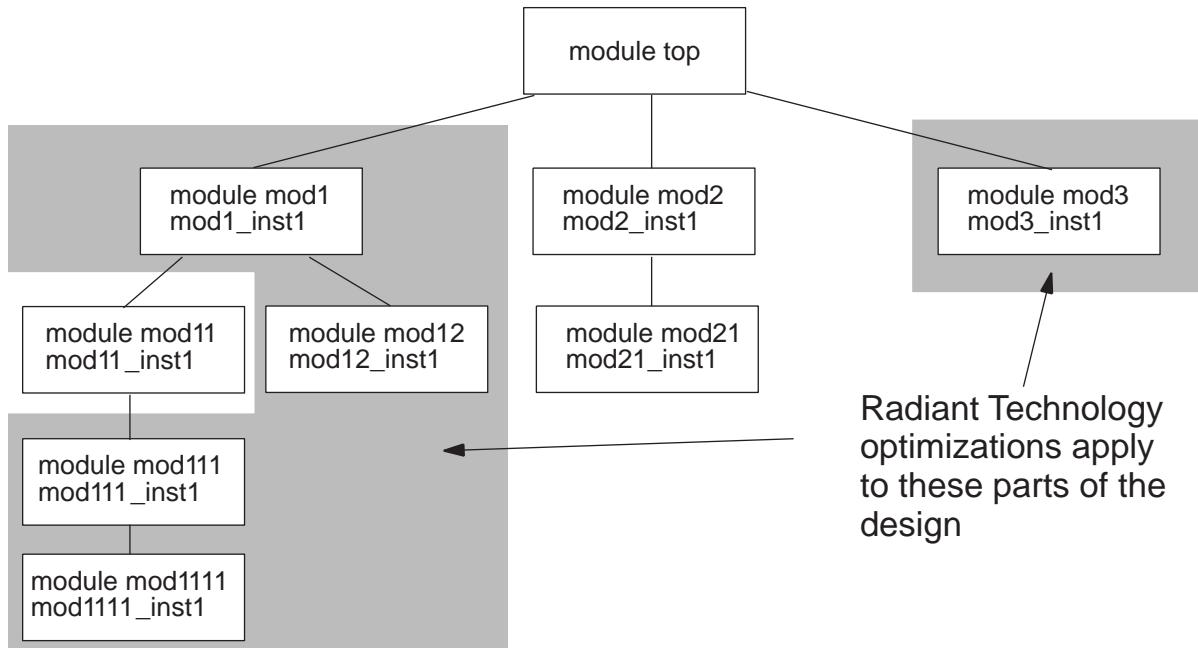
A tree statement with a depth of 0 is the equivalent of a module statement.

### Third Tree Statement Example

You can specify a negative value for the depth value. If you do this, specify ascending the hierarchy from the leaf level. For example:

```
tree (-2) {mod1, mod3} {Opt};
```

This statement specifies looking down the module hierarchy under the instances of modules `mod1` and `mod3` to the leaf level and counting up from there. (Leaf level module instances contain no module instantiation statements.)



In this example, the instances of `mod1111`, `mod12`, and `mod3` are at a depth of -1 and the instances of `mod111` and `mod1` are at a depth of -2. The attributes do not apply to the instance of `mod1111` because it is at a depth of -3.

#### Fourth Tree Statement Example

You can disable Radiant optimizations at the leaf level under specified modules. For example:

```
tree(-1) {mod1, mod2} {noOpt};
```

This example disables optimizations at the leaf level, the instances of modules `mod1111`, `mod12`, and `mod21`, under the instances of modules `mod1` and `mod2`.

#### Known Limitations

Radiant Technology is not applicable to all simulation situations. Some features of VCS are not available when you use Radiant Technology.

These limitations are:

- Back-annotating SDF Files

You cannot use Radiant Technology if your design back-annotates delay values from either a compiled or an ASCII SDF file at runtime.

- SystemVerilog

Radiant Technology does not work with SystemVerilog design construct code. For example, structures and unions, new types of always blocks, interfaces, or things defined in \$root.

The only SystemVerilog constructs that work with Radiant Technology are SystemVerilog assertions that refer to signals with Verilog-2001 data types, not the new data types in SystemVerilog.

### Potential Differences in Coverage Metrics

VCS supports coverage metrics with Radiant Technology and you can enter both the +rad and -cm compile-time options. However, Synopsys does not recommend comparing coverage between two simulation runs when only one simulation was compiled for Radiant Technology.

The Radiant Technology optimizations, though not changing the simulation results, can change the coverage results.

### Compilation Performance With Radiant Technology

Using Radiant Technology incurs longer incremental compile times because the analysis performed by Radiant Technology occurs every time you recompile the design even when only a few modules have changed. However, VCS only performs the code generation phase on the parts of the design that have actually changed. Therefore, the incremental compile times are longer when you use Radiant Technology, but shorter than a full recompilation of the design.

## Improving Performance When Using PLIs

As mentioned earlier, the runtime performance is reduced when you have PLIs accessing the design. In some cases, you may have ACC capabilities enabled on all the modules in the design, including those which actually do not require them. These scenarios unnecessarily reduce the runtime performance. Ideally the performance can be improved if you are able to control the access rights of the PLIs. However, this may not be possible in many situations. In this situation, you can use the +vcs+learn+pli runtime option.

+vcs+learn+pli tells VCS to write a new tab file with the ACC capabilities enabled on the modules/scopes which actually need them during runtime. Now, during recompile, along with your original tab file, you can pass the new tab file using the compile-time option, +applylearn+[tabfile], so that the next simulation will have a better runtime. Therefore, this is a two-step process:

- Using the runtime option +vcs+learn+pli
- Using the compilation/elaboration option +applylearn+[tabfile] during recompile. You do not have to reanalyze the files in this step.

The use model and an example is shown below:

## Use Model

**Step-1: Using the runtime option +vcs+learn+pli.**

**VCS Two-Step Flow:**

**Compilation**

```
% vcs [vcs_options] Verilog_files
```

**Simulation**

```
% simv [sim_options] +vcs+learn+pli
```

**VCS Three-Step Flow:**

**Analysis**

```
% vlogan [vlogan_options] file1.v file2.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

**Note:**

Specify the VHDL bottommost entity first, then move up in order.

**Elaboration**

```
% vcs [vcs_options] top_cfg/entity/module
```

**Simulation**

```
% simv [sim_options] +vcs+learn+pli
```

**Step-2: Using the compilation/elaboration option +applylearn+[tabfile].**

**VCS Two-Step Flow:**

**Compilation**

```
% vcs [vcs_options] +applylearn+[tabfile] Verilog_files
```

**Simulation**

```
% simv [sim_options]
```

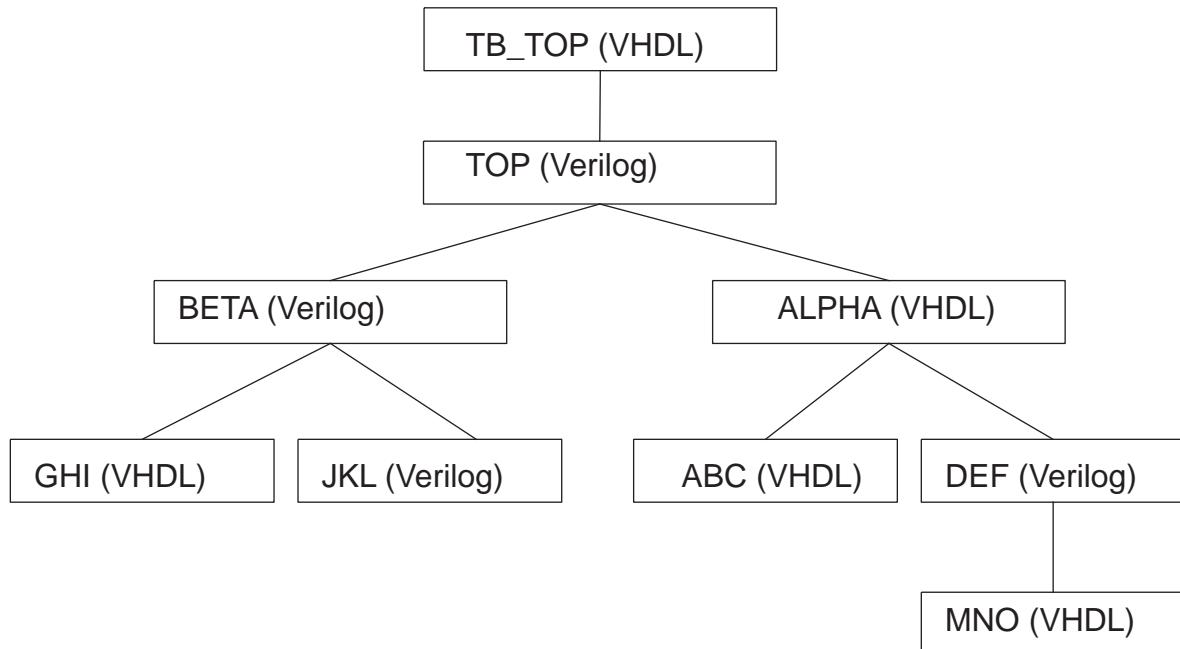
**VCS Three-Step Flow:**

**Elaboration**

```
% vcs [vcs_options] +applylearn+[tabfile] top_cfg/entity/module
```

**Simulation**

```
% simv [sim_options]
```



Consider the above example, and your `pli.tab` file is as follows:

```
% cat pli.tab
////// MY TAB FILE/////
acc=rw:*
```

The above tab file enables ACC read/write capabilities on all the modules in the design. However, in this example, you are only interested in having ACC read/write capabilities on the `jkl` module only.

The use model to invoke `+vcs+learn+pli` is as follows:

Step-1: Using the `+vcs+learn+pli` runtime option in three-step flow.

### Analysis

```
% vlogan def.v jkl.v beta.v top.v
% vhdlan mno.vhd abc.vhd alpha.vhd ghi.vhd tb_top.vhd
```

### Note:

Specify the VHDL bottommost entity first, then move up in order.

### Elaboration

```
% vcs TB_TOP -P pli.tab pli.c
```

## Simulation

```
% simv +vcs+learn+pli
```

By default, the use of the `+vcs+learn+pli` option creates a `pli_learn.tab` file in the current working directory. You can see that the `pli_learn.tab` file has ACC capabilities enabled on only the `jkl` module.

```
% cat pli_learn.tab
////////////////////////////////////////////////////////////////// SYNOPSYS INC ///////////////
// PLI LEARN FILE
// AUTOMATICALLY GENERATED BY VCS (TM) LEARN MODE
//////////////////////////////////////////////////////////////////
acc=rw:jkl
//SIGNAL string:rw
```

Now, you can use the new tab file during elaboration to achieve a better runtime performance. The use model is as shown below:

Step-2: Using the elaboration `+applylearn+[tabfile]` option in three-step flow.

## Elaboration

```
% vcs TB_TOP -P pli.tab +applylearn+pli_learn.tab pli.c
```

## Simulation

```
% simv
```

## Limitations

Following is the limitation:

- The `+applylearn` option is not supported with the Verilog function calls from UCLI.

## Enabling TAB File Capabilities in UCLI Using `-debug_access`

UCLI checks for the debug capability of a signal applied through a PLI table file (`pli.tab`), instance/signal based PLI(SIGPLI), PLI learn file, or config file. UCLI enables this capability with the `-debug_access` option, which is the minimum debug option required to enable UCLI.

This feature improves the runtime performance by allowing you to run your design with minimum debug capability.

## Use Model

Following is the use model to check for the debug capability of a signal applied through a tab file:

```
% vcs -debug_access -sverilog -P file.tab file.v
% ./simv -ucli
```

Where, `file.tab` is the tab file that specifies the debug capability for a signal.

## Example

Consider the following test case (`test.v`) and tab file (`test.tab`):

*Example 62 test.v*

```
module top;
 reg clk, a,b,c,d;
 dut d1(clk,a,b);
 dut1 d2(clk,c,d);
 initial begin
 clk=0;
 forever #1 clk =~clk;
 end
 initial begin
 #15 $finish;
 end
endmodule;

module dut(input clk,a,output b);
 initial begin
 $display("DUT B=%b\n",b);
 end
endmodule

module dut1(input clk,a,output b);
initial begin
 $display("DUT1 B=%b\n",b);
end
endmodule
```

*Example 63 test.tab*

```
acc+=frc:dut.a
```

Compile and run `test.v` as follows:

```
% vcs -nc -sverilog -debug_access -P test.tab test.v
% ./simv -ucli
```

Following is the output:

```
ucli% force top.d1.a 0
ucli% get top.d1.a
'b0
```

Although the above test case is compiled with minimum debug option `-debug_access`, the force capability enabled through the tab file is available in UCLI.

## Impact on Performance

Compiling with the `-debug_access` option disables VCS optimizations and this impacts the performance. The `-debug_access` option disables fewer optimizations, whereas the `-debug_access+all` option disables all the optimizations. The following table describes these options and their performance impact:

*Table 21 Performance Impact of -debug\_access*

| Options                        | Description                                                                                                                                              |
|--------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------|
| <code>-debug_access</code>     | Use this option to generate a dump file. You can also use this option to invoke UCLI and Verdi with some limitations. This has least performance impact. |
| <code>-debug_access+f</code>   | Use this option if you want to use the <code>force</code> command at the UCLI prompt, and for more debug capabilities.                                   |
| <code>-debug_access+all</code> | This option enables all debug capabilities, and therefore will have a huge performance impact.                                                           |

See the section [Compiling/Elaborating the Design in the Debug Mode](#) for more information.

Note that using extensive user interface commands, like `force` or `release` at runtime, will have a huge impact on the performance.

To improve the performance, Synopsys recommends you to convert these user interface commands to HDL files and to compile/elaborate and simulate them along with the design.

Contact Synopsys Support Center ([vcs\\_support@synopsys.com](mailto:vcs_support@synopsys.com)) or your Synopsys Application Consultant for further assistance.

## Obtaining VCS Consumption of CPU Resources

You can capture the CPU resource statistics for compilation and simulation using the `-reportstats` option.

- [Use Model](#)

### Use Model

You can specify this option at compile time as well as runtime or both depending on your requirement.

For example:

```
%vcs -reportstats
or
%simv -reportstats
```

#### Note:

This option is supported only on RHEL32, RHEL64, SUSE32, and SUSE64 platforms. If you attempt to use this option on other platforms, VCS issues a warning and then continues.

When you specify this option at compile time, VCS prints out the following information.

### Compile Time

```
Compilation Performance Summary
=====
vcs started at : Sat Nov 12 11:02:38 2011
Elapsed time : 4 sec
CPU Time : 3.0 sec
Virtual memory size : 361.7 MB
Resident set size : 141.7 MB
Shared memory size : 79.7 MB
Private memory size : 62.1 MB
Major page faults : 0
=====
```

The details of the above report are as follows:

- VCS start time
- Elapsed real time: wall clock time from VCS start to VCS end
- CPU time: Accumulated user time + system time from all processes spawned from VCS

- Peak virtual memory size summarized from all the contributing processes at specific time points
- Sum of resident set size from all the contributing processes at specific time points
- Sum of shared memory from all the contributing processes at specific time points
- Sum of private memory from all the contributing processes at specific time points
- Major fault accumulated from all processes spawned from VCS

## Simulation Time

Specifying this option at compile time and runtime, VCS prints out both the compile time and simulation time data:

Following is the simulation time sample report data:

```
Simulation Performance Summary
=====
Simulation started at : Sat Nov 12 11:02:43 2011
Elapsed Time : 1 sec
CPU Time : 0.1 sec
Virtual memory size : 152.2 MB
Resident set size : 106.5 MB
Shared memory size : 21.2 MB
Private memory size : 85.3 MB
Major page faults : 0
=====
```

If you specify the option only at runtime and not at compile time, VCS prints only runtime data at runtime.

**Note:**

The `-reportstats` option includes the compile time and compile memory information related to `-kdb` in the unified debug flow.

## Dumping Design Statistics

VCS generates design statistics covering various construct usages (such as number of modules, number of instances, number of signals and so on) and the design size. The information helps you to verify or view the design changes done over a period of time and even across releases.

- [Use Model](#)
- [Usage Example](#)
- [Limitations](#)

## Use Model

To dump user design statistics, use the following compile-time option:

```
-design_stats=<design|module|hier|filename>
[: (top_module_name) | (log_filename)]
```

Where,

<design>

Specify this option to dump statistics for the whole design.

<module>

Specify this option to dump statistics for the specified module in the design.

<hier: (top\_module\_name) >

Specify this option to dump statistics for the hierarchy top specified with <top\_module\_name>.

<filename: (log\_filename) >

Specify this option to dump statistics into a file whose name is specified by log\_filename. If the filename is not specified, VCS dumps the design statistics in the vcs\_design\_stats.log file by default.

## Usage Example

Consider the following test case (test.v):

*Example 64 Dumping design statistics*

```
module top;
 reg r;
 sub0 s0();
endmodule
module sub0;
 reg r;
 sub1 s1();
endmodule
module sub1();
 reg r;
endmodule
```

Compile the test case as follows:

```
% vcs -design_stats test.v
```

VCS generates the statistics for the whole design in the vcs\_design\_stats.log file.

Compile the test case as follows:

```
% vcs -design_stats=design,filename:my.log test.v
```

VCS generates the statistics for the whole design in the `my.log` file.

Compile the test case as follows:

```
% vcs -design_stats=module,filename:my.log test.v
```

VCS generates the statistics for each module of the design in the `my.log` file.

Compile the test case as follows:

```
% vcs -design_stats=hier:sub0,filename:my.log test.v
```

VCS generates the statistics for the part of hierarchy of the design starting from module `sub0` in the `my.log` file. In other words, VCS generates statistics only for `sub0` and `sub1` modules.

The content of the generated `vcs_design_stats.log` is as follows:

```
// =====
// DESIGN STATISTICS
// =====
//
// Static
//
// -----
// No. of all modules (module+interface+package+program) : 5
// No. of module instances:
// 2
// No. of all processes: 5
// No. of all nodes (variable+net): 452
// No. of all parameters: 44
// No. of class defns: 391
// No. of verilog task defns: 3

// =====
// MODULE "top" STATISTICS
// =====
// Syntax Features: sv, svtb
// No. of all modules (module+interface+package+program) :
// 1
// No. of module instances: 1
// No. of all processes: 1
// No. of all statements 5
//
// =====
// MODULE "sub" STATISTICS
// =====
// No. of all modules (module+interface+package+program) :
// 1
```

```
// No. of all nodes (variable+net) : 2
// No. of scalar nets: 2
```

---

## Limitations

The features have the following limitations:

- The design statistics are dumped only for Verilog portion of the design.
- VHDL and SystemC portions are not covered.

# 12

## Using X-Propagation

---

This chapter includes the following sections:

- [Introduction to X-Propagation](#)
  - [Using the X-Propagation Simulator](#)
  - [VHDL Two-State Objects in X-Propagation](#)
  - [X-Propagation Code Examples](#)
  - [Support for Active Drivers in X-Propagation](#)
  - [Support for Ternary Operator](#)
  - [Renaming xprop.log File](#)
  - [Limitations](#)
- 

### Introduction to X-Propagation

Designers use RTL constructs to describe hardware behaviors. However, certain RTL simulation semantics are insufficient to accurately model the hardware behaviors.

Therefore, simulation results are either too optimistic or pessimistic than the actual hardware behaviors.

The simulation semantics of conditional constructs in Verilog and the simulation semantics of the `STD_LOGIC` and `STD_LOGIC_VECTOR` types along with the Boolean equality and relational operators are insufficient to accurately model the ambiguity inherent in uninitialized registers and power-on reset values. This is particularly problematic when indeterminate states that are modeled as X values become control expressions.

Standard RTL simulations ignore the uncertainty of X-valued control signals and assign predictable output values. As a result, RTL simulations often fail to detect design problems related to the lack of X-Propagation. However, the same design problems can be detected in gate-level simulations. With X-Propagation support in RTL simulations, engineers can save time and effort in debugging differences between RTL and gate-level simulation results.

The simulation semantics of Verilog and VHDL control constructs is insufficient to account for the ambiguity of statements executed under X control. A more accurate simulation model to handle indeterminate control signals is to execute the design with a 0 and 1 control signal and then merge the results.

Gate-level simulations and pseudo-exhaustive 2-state simulations are techniques used to expose X-Propagation (Xprop) problems. However, as designs grow in size, these techniques become increasingly expensive and time consuming, often covering only a fraction of the overall design space.

The VCS Xprop simulator provides an effective simulation model that allows Xprop problems to be exposed by standard RTL simulations.

The VCS Xprop simulator provides two built-in merge modes that you can choose at either compile time or runtime:

- xmerge mode

This mode is more pessimistic than a standard gate-level simulation.

- tmerge mode

This mode is closer to actual hardware behavior and is the more commonly used mode.

In addition to these two merge modes, you can also select the vmerge mode at runtime to specify the standard RTL semantics, which effectively disables the enhanced Xprop semantics.

- vmerge mode

This mode is the classic Verilog and VHDL (optimistic) behavior.

## Guidelines for Running X-Propagation Simulations

Enabling Xprop on an entire design changes the simulation behavior, and may result in simulation failures. To facilitate deployment on existing designs, you can use the following divide-and-conquer approach to debug the failures:

- Enable Xprop on certain blocks at a time.
- Find and fix any design or testbench issues.
- Repeat the steps for the next set of blocks.

Debugging simulation failures is easier when only a small block is enabled for Xprop simulation at a time. However, resolving the Xprop simulation issues in all the small blocks independently does not guarantee that the entire design can simulate without any Xprop problems. Multiple iterations may be required to debug and fix all the issues.

One of the most common sources of simulation differences with Xprop enabled is incorrect initialization sequences. The behavior is typically caused by a reset or clock signal transitioning from 0 to x, 1 to x or vice versa.

If a flip-flop is sensitive to the rising edge of its clock signal, an x to 1 transition triggers the flip-flop and pass the value from input to output when coded using the Verilog `posedge` or VHDL `clk'` event type of usage. Effectively, the RTL constructs in these cases consider the x to 1 transition as `true`. However, in an Xprop simulation, the same clock transition causes the flip-flop to merge the input and output, possibly resulting in an unknown value. Therefore, to effectively load new values onto a flip-flop, you must ensure that clock signals have valid and stable values.

You can specify various Xprop behaviors using a configuration file. In an Xprop configuration file, you can specify the top-level module of a DUT (Design Under Test) and enable Xprop on the DUT instance tree. The Xprop technology is targeted for designs simulating actual hardware (synthesizable RTL code). Non-synthesizable or testbench blocks should be excluded from Xprop simulation using the configuration file.

Debugging a simulation mismatch is easier at the RTL level than at the gate level because RTL descriptions are closer to the actual functional intent of a circuit. There are different methods to debug RTL simulation failures.

A typical debug flow is:

- Identify a regression or test failure.
- Rerun the test with waveform dumping enabled.
- Go to point of test failure (assertion, monitor).
- Trace back a mismatching signal to its origin.
- Identify the root cause of the problem.

One method is to compare the dump file of a passing test and a failing test and search for differences near the point of failure.

Another debugging method is to compare a test when the simulation passes and when the simulation fails. The user identifies potential code modifications between the passing and the failing simulations that may cause simulation failure.

Traditional RTL debug techniques can be used to debug Xprop simulation failures.

However, you should generally not compare waveform files with and without Xprop enabled. This can lead to extraneous and wasted debug cycles. For example, resetting a device may take 10ms in normal RTL mode. In Xprop mode, the reset or clock may take 100ms due to an indeterminate reset signal. If you compare the waveform files of the two simulations, you can find that because the simulations are not cycle accurate with respect

to one another, the actual problem is at a point much farther away in the future than the first few simulation mismatches.

Most simulation debug tools automatically trace back signal changes across multiple logic levels to some origin that caused the signal changes. These debug tools are closely tied to RTL behaviors. Since VCS signal update is different when Xprop is enabled, these debug tools may not function accurately in Xprop simulations. Some manual interventions may be required to use these debug tools correctly.

The recommended debug methodology is to implement sufficient number of assertions or testbench monitors. With this methodology any deviation from the correct design functionality triggers one of these checkers and gives a runtime error message. You can debug the simulation problem starting with the error message.

## Using the X-Propagation Simulator

The Xprop usage model compiles the design in Xprop mode and execute the simv executable. The `-xprop` compile-time option is used to enable Xprop and to specify the merge mode at runtime. By default, VCS uses the `tmerge` merge mode. However, you can specify a different merge mode at either compile time or runtime.

Following is the syntax of the `-xprop` option:

```
vcs -xprop[=tmerge|xmerge|xprop_config_file]
 [-xprop=flowctrl]
 [-xprop=nestLimit=<limit>]
 [-xprop=xindex=<select_mode>]
 other_vcs_options
```

Where:

`-xprop`

Use this option without an argument to enable Xprop in the entire design. The default `tmerge` merge mode is used at runtime. This option is not recommended as Xprop instrumentation should be applied to RTL code only. This option would be deprecated in the upcoming releases.

`tmerge`

Use the `tmerge` merge mode in the entire design. The merge result yields `x` when all output values of logic 0 and logic 1 control signal are different, similar to a ternary operator. This mode is closer to actual hardware behavior and is more commonly used.

`xmerge`

Use the `xmerge` merge mode in the entire design. Merge result always yields `x` if there is any active driver. In case no active driver is present for the signal, the previous value is retained. This mode is more pessimistic than a standard gate-level simulation.

`xprop_config_file`

Specify a configuration file. You can define the scope of the Xprop instrumentation and select the merge mode in the configuration file. For more information on using the Xprop configuration file, see [X-Propagation Configuration File](#).

`xindex=<select_mode>`

Enables Xindex with the index resolution mode specified as `<select_mode>`. The `select_mode` can be one of resolution, dimensional, or random. For more details on Xindex, see [Support for XIndex Element Merging](#).

`flowctrl`

By default, Xprop does not instrument `for` loop containing `next/continue`, `exit/break` and `return` statements. The entire parent statement chain is also disabled for Xprop instrumentation. You can use the `-xprop=flowctrl` option at compile time if you want Xprop instrumentation on such constructs. Some runtime overhead is expected when instrumenting such blocks.

Consider the following case, where without `-xprop=flowctrl` Xprop skips instrumentation as captured in the following `xprop.log`:

```
test.v:41 NO "a disabled 'break', 'continue', or 'return' statement" (41)
test.v:42 NO "a disabled 'break', 'continue', or 'return' statement" (42)
test.v:43 NO "a disabled 'break', 'continue', or 'return' statement" (43)
test.v:44 NO "a disabled 'break', 'continue', or 'return' statement" (44)
=====

```

X P R O P S T A T I S T I C S

instrumentable assignments: 0

instrumented assignments: 0

instrumentation success rate: --

If you provide `-xprop=flowctrl` on VCS command line, you will see the following information on instrumentation in `xprop.log`:

```
test.v:41 YES
=====
```

```
X P R O P S T A T I S T I C S
instrumentable assignments: 10
instrumented assignments: 10
instrumentation success rate: 100%
nestLimit=<limit>
```

Specify the nesting limit for the `case` and `if` statements. Here, `<limit>` is any integral value. If you specify 0 or any negative value, Xprop is disabled completely. By default, the nesting limit for `case` and `if` statements is set to 128.

**Note:**

Simulation behavior is undefined for multiple specifications of the `-xprop` options, `tmerge`, `xmerge` and `xprop_config_file`. In the following command, the application of the option is undefined and an error is generated.

```
% vcs -xprop=xprop.cfg -xprop=unifiedInference design.v
```

You may specify the following options only once in the compilation command:

- Merge mode (`tmerge` or `xmerge`)
- Configuration file (`xprop_config_file`)

**Examples:**

```
vcs -xprop
vcs -xprop=xprop.cfg
vcs -xprop=tmerge top.v
```

**Note:**

The automatic hardware inference of flip-flops in Verilog simulations is enabled by default. Flip-flops with an active reset value of 0 are correctly simulated when the reset signal transition from X to 0. VCS generates a file named `unifiedInference.log` file to record a list of inferred flip-flops.

**Note:**

Using the `-xprop=mmsopt` compile-time option helps to improve the runtime performance by trading off the ability to change merge mode at runtime.

## Runtime Options

This section describes the VCS runtime options for Xprop.

```
-xprop=banner
```

The `-xprop=banner` option enables printing a message about Xprop merging state during the simulation. The message gets printed at time 0, and whenever there is a change in the merge mode.

If Xprop is not enabled at compile time and the `-xprop=banner` option is provided at runtime, the following message gets displayed

```
%simv -xprop=banner
Note-[XPROP_RT_BANNER] Xprop Runtime Banner
Xprop is off
```

Consider an example where Xprop is enabled at compile time with `tmerge` mode and at runtime, at time 10, the merge mode is changed to `xmerge`, then the following messages will be displayed at time 0 and time 10 respectively:

```
Note-[XPROP_RT_BANNER] Xprop Runtime Banner
Global merge mode is set to tmerge at time 0
```

and

```
Note-[XPROP_RT_BANNER] Xprop Runtime Banner
Global merge mode is set to xmerge at time 10
```

**Note:**

Changing the merge mode does not retrigger the statements or blocks. You can view the effective values from the new merge mode only on the next input change.

## Specifying X-Propagation Merge Mode

Merge mode can be mentioned at both compile time and runtime.

For example,

```
% vcs -xprop=xmerge -sverilog top.v
```

When the `-xprop=xmerge` option is specified, the design is compiled and simulation starts in the `xmerge` mode.

```
% vcs -xprop=tmerge -sverilog top.v
```

When the `-xprop=tmerge` option is specified, the design is compiled and simulation starts in the `tmerge` mode.

To change the merge mode at runtime you can invoke the `$set_x_prop` Verilog system task or VHDL `set_x_prop` procedural calls.

**Verilog examples:**

```
$set_x_prop("tmerge");
$set_x_prop("xmerge");
$set_x_prop("vmerge");
$set_x_prop("xprop");
```

**VHDL examples:**

```
set_x_prop("tmerge");
set_x_prop("xmerge");
set_x_prop("vmerge");
set_x_prop("xprop");
```

**Note:**

You cannot change the merge mode to `vmerge` during compile time. It can be done only by invoking `$set_x_prop` Verilog system task during runtime.

You must use the `use` clause in the file that calls the `set_x_prop` procedures.

```
use SYNOPSYS.XPROP_USER.ALL;
```

When only `-xprop` compile-time option is passed without specifying a merge mode and if you do not use `$set_x_prop` to specify the merge mode, the default `tmerge` mode is used.

The `vmerge` runtime merge mode enables standard RTL simulation behaviors, which effectively disables the Xprop semantics.

**Note:**

When multiple merge modes are specified using runtime options at the same delta time, the last merge mode always wins.

Another method to specify the Xprop merge mode is to use a Xprop configuration file. The `xprop` runtime merge mode reverts the merge scheme to use the merge scheme specified at the compile time. For more information on how to use the Xprop configuration file, see [X-Propagation Configuration File](#).

For example:

```
% vcs -xprop=xp_config cache.v alu.v
```

Use the merge mode and the scope of Xprop instrumentation specified in the Xprop configuration file `xp_config`.

## Compile Time Diagnostic Report

When you compile a design with Xprop enabled, VCS generate reports that record all the statements considered for Xprop instrumentation, whether or not the statements are instrumented, and the reason for statements not being instrumented. Reports are generated with the name `xprop.log` and `xprop_vhdl.log` for Verilog and VHDL respectively.

Report entries are created for the following HDL constructs:

- `if` statement in Verilog and VHDL
- `case` statement in Verilog and VHDL
- Body of an edge triggered `always` block in Verilog
- Edge sensitive expression in VHDL

Below is the format of a report entry in `xprop.log/xprop_vhdl.log` files:

`filename:line_number YES|NO ["reason" (primary_line)]`

Where:

`filename`

Name of the source file containing a statement being considered for Xprop instrumentation.

`line_number`

Line number that corresponds to the start of the statement.

`YES|NO`

Xprop instrumentation status of the statement.

`reason`

The reason for the statement not being instrumented. This is issued only when the Xprop instrumentation status is `NO`.

`primary_line`

The line number of the statement containing the actual construct not being instrumented. This is issued only when the Xprop instrumentation status is `NO`.

`xprop.log` example:

```
decode.v:3 YES
decode.v:7 NO "prevented by sub-statement" (12)
decode.v:16 NO "delay statement" (17)
```

```
decode.v:18 YES
decode.v:20 NO "a dynamic object" (22)
```

xprop\_vhdl.log example:

```
mux.vhd:22 YES
mux.vhd:35 YES
mux.vhd: 44 NO assignment has non-null delay (41)
```

The Xprop statistics are presented at the end of the *xprop.log* report. The statistics consist of the number of assignment statements considered for Xprop instrumentation, the number of statements instrumented, and the ratio of those two numbers (instrumented/instrumentable) that represents the percentage of the design instrumented for Xprop.

For example:

```
eth.v:31 YES
eth.v:45 NO "a dynamic type expression" (48)
eth.v:52 YES
=====
X P R O P S T A T I S T I C S
instrumentable assignments: 7
instrumented assignments: 5
instrumentation success rate: 71%
```

The Xprop instrumentation numbers reported in *xprop.log* are essentially the same between different compilation flows. In certain cases, subtle internal differences between the compilation flows may affect the calculation of the instrumentation numbers. Even though the instrumented code is same in both flows, the numbers in *xprop.log* may differ. You should not compare the Xprop instrumentation numbers between different compilation flows.

#### Note:

VCS does not record Xprop instrumentation information on modules that are excluded from Xprop via the configuration file. Instrumentation for instances excluded via the configuration file is recorded unless all instances of a particular module are excluded. The excluded modules do not appear in the *xprop.log* file.

The *xprop\_merged.log* file contains merged statistics from both VHDL and Verilog.

## Querying X-Propagation at Runtime

You can use the `$is_xprop_active` Verilog system function or the `is_xprop_active` VHDL procedure to query the X-prop status for a particular module or an entity instance. The function returns an 1 if Xprop is enabled in the current instance.

For example:

```
$set_x_prop("tmerge");
$display("%m: is Xprop active = %d", $is_xprop_active());

$set_x_prop("xmerge");
$display("%m: is Xprop active = %d", $is_xprop_active());

$set_x_prop("vmerge");
$display("%m: is Xprop active = %d", $is_xprop_active());
```

The Xprop configuration file:

```
tree {top}{xpropOn}
instance {top.dut2}{xpropOff}
```

The query result:

```
top.dut1: is Xprop active = 1
top.dut1: is Xprop active = 1
top.dut1: is Xprop active = 0
top.dut2: is Xprop active = 0
top.dut2: is Xprop active = 0
top.dut2: is Xprop active = 0
```

In the above example, Xprop instrumentation in the *top.dut2* instance is disabled at compile time using the configuration file. As a result, the instrumentation cannot be changed at runtime using the *\$set\_x\_prop* system tasks. The Xprop status is shown in the query result.

The example for querying in VHDL is:

```
report boolean'image(is_xprop_active);
```

## X-Propagation Instrumentation Report

You can generate an Xprop instrumentation report with the *-report* runtime option. The report displays the instrumentation status of every module instance in a design.

```
-report=xprop[+exit]
```

Where:

xprop

Generate an Xprop instrumentation report named *xprop\_config.report* and continue the simulation.

xprop+exit

Generate an Xprop instrumentation report named *xprop\_config.report* and terminate the simulation.

The following are the formats of the statements in the Xprop instrumentation report:

ON:`instance`  
OFF:`instance`

Where:

`instance`

A module instance in a design.

ON

The instance is included in Xprop instrumentation.

OFF

The instance is excluded from Xprop instrumentation.

For example:

`simv -report=xprop`

The `xprop_config.report` file:

ON: `top`  
ON: `top.il`  
ON: `top.p1`  
OFF: `top.p1.ul`

## Automatic Hardware Inference of Flip-Flops Enabled by Default

The automatic hardware inference of flip-flops in Verilog simulations is enabled by default. Flip-flops with an active reset value of 0 are correctly simulated when the reset signal transition from X to 0. VCS generates an `unifiedInference.log` file to record a list of inferred flip-flops. The unified inference statistics are presented at the end of the log file.

The following are the formats of the `unifiedInference.log` file entries:

`filename:line_number YES:SyncFF|AsyncFF`  
`filename:line_number NO "reason"`

Where:

`filename`

Name of the source file containing a flip-flop that is being considered for Xprop instrumentation.

`line_number`

Line number that corresponds to the start of a statement describing the flip-flop.

`YES`

The specified flip-flop is inferred.

SyncFF|AsyncFF

Type of the specified flip-flop. SyncFF indicates a synchronized flip-flop. AsyncFF indicates an asynchronous flip-flop.

NO

The specified flip-flop is not inferred.

reason

The reason for the specified flip-flop not being inferred.

For example:

```
lib.v:3 YES:AsyncFF
lib.v:10 YES:SyncFF
lib.v:17 NO "Unable to infer clock for flip-flop"
=====
Unified Inference Statistics
Number of always_ff: 0
Number of always_latch: 0
Number of always @*: 0
Number of always_comb: 0
Number of always: 4
Number of flip-flop candidates: 3
Number of synchronous flops inferred: 1
Number of Asynchronous flops inferred: 1
```

## X-Propagation Configuration File

The Xprop configuration file is used to define the scope of Xprop instrumentation in a design. This file allows you to specify the design hierarchies or modules to be excluded or included for Xprop instrumentation. You can also use the file to specify merge modes. Synopsys recommends that you use a Xprop configuration file when Xprop is enabled.

### Note:

If you use an Xprop configuration file, by default VCS does not perform Xprop instrumentation. You must use the *xpropOn* attribute to specify the design hierarchies or modules for Xprop instrumentation. For example:

```
module {*} {xpropOn};
```

The Xprop configuration file consists of the following types of statements:

- [X-Propagation Instrumentation Definition](#)
- [X-Propagation Merge Mode Specification](#)

The statements are processed in sequential order. Subsequent statements override the previously listed statements. If Xprop merge mode is specified multiple times in the configuration file, the merge mode from the last statement is enabled.

**Note:**

You can add comments to the file using the character types // and /\* \*/. For example:

```
// This is a single line comment /* This is a multi-line
comment*/
```

## X-Propagation Configuration File Syntax

The following is the BNF of the Xprop configuration file.

```
xprop_config_text ::= { xprop_config_item ; }

xprop_config_item ::=
 merge = merge_function
| xindex_select_method = select_mode
| disable_xindex = read_write_operation
| module_item
| instance_item
| tree_item

merge_function ::= tmerge | xmerge
select_mode ::= resolution | dimensional | random
read_write_operation ::= read | write

module_item ::= module { module_identifier_list } { xprop_mode }

tree_item ::= tree { module_identifier_list } { xprop_mode }

instance_item ::= instance [(depth)] { instance_path_list }
{ xprop_mode }

xprop_mode ::= xpropOn | xpropOff | tmerge | xmerge

module_identifier_list ::= module { , module }

module ::= module_identifier

instance_path_list ::= instance_path { , instance_path }
instance_path ::= { instance_identifier . } instance_identifier
```

**Note:**

If a merge mode is explicitly specified for a module, or a tree, or an instance, then that mode is used throughout the simulation, whenever Xprop is enabled on that module, or instance, or tree.

For the pure Verilog and the Verilog portion of a mixed HDL design, the following warning message is generated:

```
Warning-[UACF] Unrecognized attribute in Config file
Unrecognized attribute 'xmerge' found in config statement, it will be
ignored.
```

### X-Propagation Instrumentation Definition

The following are the BNF rules for the Xprop instrumentation definition in a configuration file:

```
module_item ::= module { module_identifier_list } { xprop_mode }

tree_item ::= tree { module_identifier_list } { xprop_mode }

instance_item ::= instance [(depth)]
{instance_path_list } { xprop_mode }
```

Where,

`module_item`

Apply the specified Xprop mode to all modules in the list.

`instance_item`

Apply the specified Xprop mode to all instances in the list and recursively to all the sub-instances. However, recursion varies with the `depth` option. An instance rule with `depth ::= 0` considers the entire sub-tree for Xprop instrumentation. Instance rule with `depth ::= 1`, considers only the specified instance, leaving the sub-tree's configuration status unchanged. If no depth or any other depth besides 0 or 1 is provided, it means the same as `depth ::= 0`, and indicates that the rule applies to the entire sub hierarchy.

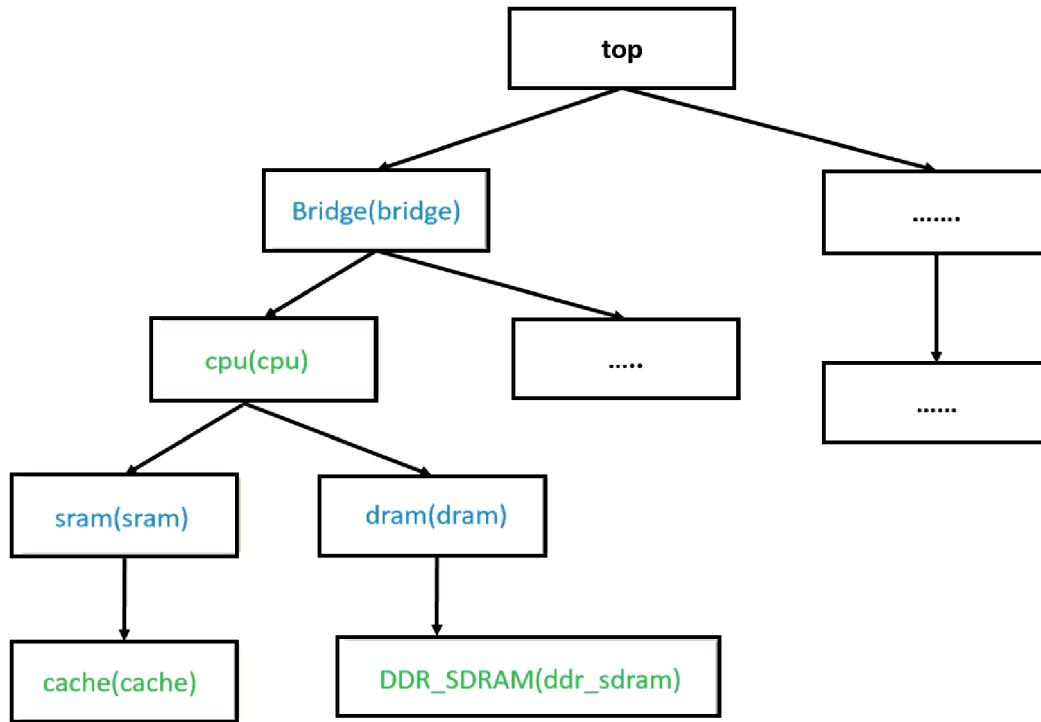
`tree_item`

Apply the specified Xprop mode to all modules in the list and recursively to all the sub-instances.

For example,

Consider the following figure, in which Xprop instrumentation is turned off using the `tree` and `module` directives, and turned on using `instance` directives.

Figure 130 Xprop Instrumentation Using Instance Directives



**Note:**

Green color indicates the X-prop instrumented nodes and blue color indicates the non X-prop instrumented nodes.

```
tree { bridge } { xpropOff } ;
```

Specifies that the sub-tree under every instance of the module `bridge` should be excluded from Xprop instrumentation.

```
instance (0) { top.bridge.cpu } { xpropOn } ;
```

Designates that the sub-tree under `top.bridge.cpu` should be included for X-propagation recursively.

```
module { sram } { xpropOff } ;
```

Specifies that all instances of the `sram` module are excluded from the Xprop instance, but not its sub-instance, `cache`.

```
instance (1) {top.bridge.cpu.dram} {xpropOff};
```

Specifies that only `dram` instance is excluded from Xprop instrumentation, but this directive does not affect the sub-instance `ddr_sram`. This instance retains its configuration setting from the tree directive above.

A module specification in a configuration file only affects the module instances, but not its sub-instances. For Verilog modules only, a module identifier may include asterisks (\*) to denote a wildcard string.

### X-Propagation Merge Mode Specification

The following are the BNF rules for the Xprop merge mode specification in a configuration file.

```
merge = merge_function
merge_function ::= tmerge | xmerge
```

merge\_function

Enable the specified Xprop merge mode.

For example:

```
merge = xmerge ;
```

In the above example, xmerge mode is enabled.

## Xprop Instrumentation Control

[Table 22](#) is a summary of the Xprop instrumentation control in a design hierarchy, when Xprop is enabled using various methods.

*Table 22      Xprop Instrumentation Control*

| Xprop Specification                                                                                                                                                                                             | Instrumentation Control              | Runtime Control                                                                                                                                                                                       |
|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Configuration file<br><b>example:vcs</b><br>-xprop=cfg                                                                                                                                                          | Instance/module specific control     | Verilog task call \$set_x_prop or VHDL procedure call set_x_prop can be used to change merge mode at runtime for instances that have been already compiled for Xprop<br><i>Caution: Use sparingly</i> |
| Process based<br><b>Verilog example:always</b><br>(*xprop_off*) @ (posedge clk)<br><b>VHDL example:process</b><br>(s, a, b)-- pragma xprop_off begin if (s = '1') then r <= a; else r <= b; end if;end process; | Disabled Xprop for this process only | Process is disabled and cannot be enabled at runtime                                                                                                                                                  |

**Table 22 Xprop Instrumentation Control (Continued)**

| Xprop Specification                                          | Instrumentation Control                              | Runtime Control                                                                                                                                                                                       |
|--------------------------------------------------------------|------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Compile-time option example:<br>example:vcs<br>-xprop=tmerge | Entire hierarchy is instrumented for Xprop semantics | Verilog task call \$set_x_prop or VHDL procedure call set_x_prop can be used to change merge mode at runtime for instances that have been already compiled for Xprop<br><i>Caution: Use sparingly</i> |

When a model uses Xprop configuration file, by default nothing is instrumented for Xprop.

To enable Xprop on all modules, packages, and \$unit, start the configuration with a wildcard \* (asterisk).

```
module {*} {xpropOn};
```

As an example, consider the following code that needs to be simulated:

```
%cat t.v:
`include "unit_functions.vh"
module top ();
 alldmtxd alldmtxd_xPropMe ();
 alldmtxd alldmtxd_FreeSpirit ();
endmodule

module alldmtxd ();
 import my_pkg::*;
 logic [6:0] a7,b7,c7;
 logic a,b;
 assign a7 = f_INC(b7,a ,b);
 assign c7 = f_PKG(b7,a ,b);
endmodule

%cat unit_functions.vh :
function logic [6:0] f_INC(logic [6:0] vec, logic enable45, logic
enable67);
 logic [6:0] tmp;
 unique casez({vec,enable45,enable67})
 9'b0000000_11 : tmp = 7'b1111111;
 9'b1111111_11 : tmp = 7'b1111110;
 9'b1111110_11 : tmp = 7'b1111100;
 default : tmp = 7'b1111111;
 endcase
 f_INC = tmp;
endfunction

%cat pkg_functions.vh :
package my_pkg;
```

```

function logic [6:0] f_PKG(logic [6:0] vec, logic enable45, logic
 enable67);
 logic [6:0] tmp;
 unique casez({vec,enable45,enable67})
 9'b0000000_11 : tmp = 7'b1111111;
 9'b1111111_11 : tmp = 7'b1111110;
 9'b1111110_11 : tmp = 7'b1111100;
 default : tmp = 7'b1111111;
 endcase
 f_PKG = tmp;
endfunction

endpackage

```

To enable Xprop instrumentation on just some hierarchy (without \$unit and without any packages), use the following rule:

```
instance {top.alldmtxd_xPropMe } {xpropOn};
```

To enable Xprop instrumentation on \$unit, use the following rule:

```
module {_vcs_unit* }{xpropOn};
```

To enable Xprop instrumentation on my\_pkg package, use the following option:

```
module {my_pkg }{xpropOn};
```

To turn on Xprop on alldmtxd\_xPropMe(), the configuration file looks like this.

```
%cat xprop.cfg
module {8} {xpropOn};
instance {top} {xpropOff};
instance {top.alldmtxd_xPropMe} {xpropOn};
```

## Limitation

The Xprop Instrumentation configuration has the following limitation:

- Does not support partial wildcard in Xprop configuration rules with pure-VHDL and MX designs.
- The instance depth support is for depths of 0 or 1 only. Any other depth is a semantic warning and the depth is suppressed to 0. An instance directive with a depth specification and any non-Xprop property (that is, anything except xpropOn, xpropOff, tmerge, xmerge) is also a semantic warning and the depth is ignored.

## Disabling Xprop on Library Cells

vcs Xprop provides an option -xprop=disableCellDefine for the Xprop instrumentation to be disabled on RTL modules that are specified under `celldefine. This enhances user's productivity and ease-of-use for efficiently deploying Xprop solutions.

This option only skips modules specified under `celldefine. Modules specified in Verilog files parsed with `-v/-y` are not skipped unless the module is under `celldefine.

If the module is specified in `celldefine and `-xprop=disableCellDefine` is provided, then the module/instance cannot be enabled for instrumentation using xprop configuration file.

## Process Based X-Propagation Exclusion

The Xprop configuration file allows exclusion of Xprop at module or entity level. For more information, see [X-Propagation Configuration File](#). If you need finer granularity, you can use the `xprop_off` attribute to disable Xprop on specific process.

Verilog example:

```
always (* xprop_off *) @ (posedge clk) begin
 if (we) begin
 q <= in;
 end
end
```

VHDL example:

```
process (s, a, b)
-- pragma xprop_off
begin
 if (s = '1') then
 r <= a;
 else
 r <= b;
end if;
end process;
```

### Note:

The pragma can be specified between the `process` keyword and the final semicolon after the `end process` statement. The pragma has no effect if it is specified outside of the `process` and `end process` statements. No message is issued in such cases.

The pragma is not required to start on a new line. However, the `pragma` keyword must be the first word of a comment followed by the `xprop_off` keyword with no intervening text.

You need to specify the `xprop_off` pragma before the `begin` keyword. Verilog and VHDL examples are mentioned above.

The pragma will not be effective if you specify the `xprop_off` pragma after the `begin` keyword. The examples are as follows:

**Verilog example:**

```
always begin (* xprop_off *) @ (posedge clk)
 if (we) begin
 q <= in;
 end
end
```

**VHDL example:**

```
process (s, a, b)
begin
-- pragma xprop_off
if (s = '1') then
 r <= a;
else
 r <= b;
end if;
end process;
```

## Support for XIndex Element Merging

VCS supports XIndex element merging in Verilog. This helps you to configure the simulator to propagate xs through array indexes. This improves RTL confidence and overall verification productivity by reducing debug effort during gate-level simulations.

The following two operations are considered for XIndex element merging in Verilog:

- Write element merging semantics
- Read element merging semantics

A write operation merges the RHS expression into each potentially written element. A read operation merges values of all the elements potentially read.

When dealing with X-indexes, there are two distinct operations to consider: Index selection and Merge mode.

The Index selection is the mechanism that determines the array locations involved in the indexing operation.

The Merge mode is the semantic associated with the array locations identified by the index selection. There are three different index selection mechanisms and two merge behaviors, which leads to six distinct new behaviors, in addition to the default HDL behavior.

The following are the index selection mechanisms:

- Dimensional index
- Random index
- Index resolution

The three index selection mechanisms exhibit different accuracy and performance characteristics.

Aside from these different index selection mechanisms, Xprop merge modes (`tmerge`, `xmerge`, `vmerge`) is applied. The merge mode is not independently controllable. Instead, the merge mode of the local context is applied.

XIndex considers all select expressions within an Xprop region as specified by the configuration file, if any, including select expressions in `always` blocks, processes and continuous or concurrent assignments. However, XIndex does not consider select expressions that are in instance port connections.

XIndex does not result in any additional `YES` entries in the log file. All select expressions within Xprop-enabled regions are considered. However, instance port expressions and XIndex expressions on function calls associated with output or inout expressions are not instrumented. Such unsupported expressions are logged in the `xprop.log` file with a `NO` entry.

## X-Index

An indexing expression that contains one or more X bits (Verilog) or the X value (VHDL).

## Index BSpace

Index BSpace is the set of values that are possible in the addressable space of the designated array given a particular X-Index pattern. The addressable space of the array are the index values within the zero-based power-of-2 space that encompasses the bounds of dimensions that are indexed.

For example, the BSpace for the `x0x` pattern in range `[6:1]` is the set of minterms `{001, 100, 101}`. In this case, the zero-based power-of-2 space for the range `6:1` is `7:0` ( $2^{**3}-1:0$ ). For read operations, the index BSpace of an X pattern includes all minterms in the set. For write operations, the index BSpace is further constrained to the subset of minterms that are in-bounds.

A multidimensional BSpace is comprised of a set of n-tuples, where n is the number of dimensions addressed in the multidimensional array select expression. For example, consider the following code snippet:

```
logic array[6:1][7:0][3:0];
logic [2:0] a, b;
logic [1:0] c;
```

```
logic d;
...
a = 3'bx0x;
b = 1;
c = 2'b0x
array[a][b][c] = d;
```

For the assignment in the last statement, BSpace is the set of 3-tuples:

```
{(001, 1, 00), (001, 1, 01), (100, 1, 00), {100, 1,01), (101, 1, 00),
(101, 1, 01)}
```

For a read operation, BSpace also includes the out-of-bounds tuples within the power-of-2 space:

```
{(000, 1, 00), (000, 1, 01), (001, 1, 00), (001, 1, 01}, (100, 1, 00), {100, 1,01), (101, 1, 00},
(101, 1, 01)}
```

## Addressing Models

When considering dimensional indexing in the context of XIndex, the array addressing models the actual hardware.

XIndex element merging considers every dimension to exist within a power-of-2 sized space. High-order bits that lie outside an array's addressing space are ignored.

For ranges that are not a power-of-2, some minterms may be out-of-bounds. For write operations, these out-of-bounds minterms are always ignored, regardless of the element merging method. For read operations, these out-of-bounds minterms result in the undefined value, which is currently x.

The section consists of the following subsections:

- [Unsigned Indexing](#)
- [Signed indexing](#)

### Unsigned Indexing

Dimensions where both bounds are positive (including 0) are considered unsigned. For unsigned indexing, the power-of-2 space is  $2^{**n-1}:0$ , where n is the smallest integer such that  $2^{**n}$  is greater than the high bound of the dimension. For example, the range 6:1 results in n=3.

### Signed indexing

Dimensions where either bounds are negative are considered signed indexing. For signed indexing the power-of-2 space is  $2^{** (n-1)-1}:- (2^{** (n-1)})$ , where n is a smallest integer such that  $2^{** (n-1)}$  is greater than the high bound and  $- (2^{** (n-1)})$  is less than or equal to the low bound. For example, the ranges 8:-8 and 7:-9 result in n=5, while range 7:-8 results in n=4.

## Merge Modes

The runtime merge mode for XIIndex Element Merging is the same as for the Xprop merge mode. The merge mode is applied to specific instances when specified in the configuration file. Alternatively, the merge mode can be set globally in either the configuration file or via the compile-time command line option.

The merge mode can also be modified at runtime via the `$set_x_prop()` system task.

**Note:**

The X-Index Element Merging merge mode is not independently controllable.

### TMerge

For write operations, this mode merges the value being written (RHS) with the value of each element in the BSpace and storing the result of the merge into the corresponding array location. For read operations, the value read is the merge of all the values obtained from each index in the BSpace.

### XMerge

When the BSpace has more than one minterm this merge mode always results in an X being read or written. For a write operation, all elements selected by the BSpace are written.

### VMerge

The default X-Index merge mode follows standard Verilog semantics for read and write operations. VHDL also follows these same semantics when in Xprop mode.

- X-Index write behavior: The write is ignored.
- X-Index read behavior: The result is always X.

**Note:**

This merge mode disables both the index selection as well as the Xprop merge mode.

## Index Selection Methods

Element merging method is applied within an Xprop instrumentation region and are not implemented in regions where Xprop is disabled.

The section consists of the following subsections:

- [Dimensional Index](#)
- [Random Index](#)
- [Index Resolution](#)

## Dimensional Index

The dimensional index method computes the BSpace by first converting an X-Index into all xs in the corresponding dimension. An index that does not contain an x is unaffected. Therefore, an index with value `3'b0x` is converted to `3'bxxx` prior to computing BSpace minterms. Once the BSpace is computed, the RHS of the assignment is merged with the element at each index in the X-Index BSpace, similar to the resolution model. For read operation, the elements of the BSpace are merged with each other to derive the result.

For VHDL, no conversion is necessary for BSpace computation as an index is either known or it is X.

This index-selection mechanism is specified via the following configuration file directive:

```
xindex_select_method=dimensional
```

## Random Index

For the random index method, an in-bound random index is selected from the X-Index BSpace. For a write operation, the selected array cell is written with a value computed according to the merge mode. For a read operation, the value at the (randomly) selected index is returned without merging.

For VHDL, a random index is generated in-bounds of the dimension being indexed.

This index-selection mechanism is specified via the following configuration file directive:

```
xindex_select_method=random
```

## Index Resolution

For write operations, the index resolution mode merges the value being written (RHS) with each element addressed by the minterms of X-Index BSpace.

If a BSpace minterm is out-of-bounds for that dimension, the element write is ignored.

For read operations, the values of all the elements in the BSpace are merged to derive the result. If any BSpace index at a dimension is out-of-bounds, the value merged for the element is the default value x.

VHDL does not support the Index Resolution selection mechanism. If the resolution mechanism is specified in VHDL, it is downgraded to the dimensional model.

This index-selection mechanism is specified via the following configuration file directive:

```
xindex_select_method=resolution
```

## Disabling XIndex Merging for Read or Write Operations

The index-selection mechanism and the merge mode are applied consistently to all indexing expressions for read and write operations. However, there may be situations where the XIndex element merging might cause undue performance degradation

on certain operations. In particular, read operations employing the index resolution mechanism along with the `tmerge` merge mode can incur a high performance cost for relatively small accuracy gain. As read operations are much more common than write operations, and a single XIndex read operation may require the merging of many cells (potentially the entire array), the added accuracy might be unwarranted. To control such situations, you can provide two configuration directives to independently enable or disable the XIndex element merging of either a read or a write operation.

To disable the XIndex element merging of a read operation, use the following configuration directive:

```
disable_xindex = read
```

To disable the XIndex element merging of a write operation, use the following configuration directive:

```
disable_xindex = write
```

When you specify read or write to this directive, it disables the corresponding operation. By default, you can enable both of these operations by specifying the `xindex_select_method` directive.

## Use Model

You can enable XIndex element merging by using the configuration file directive as follows:

```
% vcs -xprop=<xprop_config_file> testcase.v <other_vcs_options>
```

Where, `xprop_config_file` specifies the Xprop configuration file.

The configuration file contains the following content:

```
merge = merge_mode
module {memory_rtl} {xpropOn};
xindex_select_method=select_method;
```

Where:

`merge_mode`

Specifies the merge mode. The merge mode can be `tmerge`, `xmerge`, or `xpropOff` (`vmerge`).

`select_method`

Specifies the element merging method. The element merging method are `dimensional`, `random`, or `resolution`. If you specify more than one element merging method, the last method specified in the configuration file is used.

## Examples

Consider the following examples:

```
//example.v
=====
module memory_rtl (clk,reset,wr,addr,data_in,data_out);
input clk,reset;
input wr;
input [1:0] addr;
input [7:0] data_in;
output [7:0] data_out;
wire [7:0] data_out;
reg [7:0] mem [0:3];
reg [7:0] rdata;
reg out_enable;
assign data_out = out_enable ? rdata : 'bz;
always @(posedge clk or posedge reset)
begin
 if (reset) begin
 mem[0] <= 8'b11000011;
 mem[1] <= 8'b11010100;
 mem[2] <= 8'b11110110;
 mem[3] <= 8'b00110011;
 end else if(wr)
 mem[addr] <= data_in;
end
always @(posedge clk)
begin
 if(wr==0)
 begin
 rdata <= mem[addr];
 out_enable <= 1'b1;
 end
 else out_enable <=1'b0;
end
endmodule

//testcase.sv
=====
module top;
bit clk;
reg reset,wr;
reg [1:0] addr;
reg [7:0] data_in;
wire [7:0] data_out;
always #5 clk++;
memory_rtl inst(clk,reset,wr,addr,data_in,data_out);
initial
begin
 reset = 1'b1;
 #3;
 reset = 1'b0;
```

```
$display("*****reading from memory location*****");
wr = 1'b0;
addr = 2'b0x;
@(negedge clk);
$display("addr : %b data_out : %b",addr,data_out);
addr = 2'b1x;
@(negedge clk);
$display("addr : %b data_out : %b",addr,data_out);
$display("*****writing to memory location*****");
wr = 1'b1;
addr = 2'b0x;
data_in = 8'b11010100;
@(negedge clk);
$display("addr : %b mem[0] : %b mem[1] : %b mem[2] :
%b mem[3] : %b",addr,top.inst.mem[0],top.inst.mem[1],
top.inst.mem[2],top.inst.mem[3]);
addr = 2'b1x;
data_in = 8'b11000100;
@(negedge clk);
$display("addr : %b mem[0] : %b mem[1] : %b mem[2] :
%b mem[3] : %b",addr,top.inst.mem[0],top.inst.mem[1],
top.inst.mem[2],top.inst.mem[3]);
$finish;
end
endmodule
```

### Dimensional Mode

To run the examples in the dimensional merging mode, use the following compile-time and runtime commands:

```
% vcs -sverilog example.v testcase.sv -xprop=testcase.cfg
% ./simv
```

Where, `testcase.cfg` has the following entries:

```
merge=tmerge;
module {memory_rtl} {xpropOn};
xindex_select_method=dimensional;
```

It generates the following output:

```
*****reading from memory location*****
addr : 0x data_out : xxxx0xxx
addr : 1x data_out : xxxx0xxx
*****writing to memory location*****
addr : 0x mem[0] : 110x0xxx mem[1] : 11010100 mem[2] : 11x101x0 mem[3] :
xxx10xxx
addr : 1x mem[0] : 110x0xxx mem[1] : 110x0100 mem[2] : 11xx01x0 mem[3] :
xxxx0xxx
```

## Random Mode

To run the examples in the random merging mode, use the following compile-time and runtime commands:

```
% vcs -sverilog example.v testcase.sv -xprop=testcase.cfg
% ./simv
```

Where, *testcase.cfg* has the following entries:

```
merge=tmerge;
module {memory_rtl} {xpropOn};
xindex_select_method=random;
```

It generates the following output:

```
*****reading from memory location*****
addr : 0x data_out : 11000011
addr : 1x data_out : 00110011
*****writing to memory location*****
addr : 0x mem[0] : 11000011 mem[1] : 11010100 mem[2] : 11110110 mem[3] :
00110011
addr : 1x mem[0] : 11000011 mem[1] : 11010100 mem[2] : 11110110 mem[3] :
xxxx0xxx
```

## Resolution Mode

To run examples in the resolution merging mode, use the following compile-time and runtime commands:

```
% vcs -sverilog example.v testcase.sv -xprop=testcase.cfg
% ./simv
```

Where, *testcase.cfg* file has the following entries:

```
merge=tmerge;
module {memory_rtl} {xpropOn};
xindex_select_method=resolution;
```

It generates the following output:

```
*****reading from memory location*****
addr : 0x data_out : 110x0xxx
addr : 1x data_out : xx110x1x
*****writing to memory location*****
addr : 0x mem[0] : 110x0xxx mem[1] : 11010100 mem[2] : 11110110 mem[3] :
00110011
addr : 1x mem[0] : 110x0xxx mem[1] : 11010100 mem[2] : 11xx01x0 mem[3] :
xxxx0xxx
```

## Limitations

The feature has the following limitations:

- Only memory of type `reg`, `wire`, and `logic` declared in the module scope are supported.
- Memory declared inside a class and dynamic array types are not supported.

## Bounds Checking

In Verilog, the `-boundscheck` compile-time option can be used to catch invalid indices. When the option is specified, the four types of assertions listed in [Table 23](#) are enabled:

*Table 23 Assertions Enabled In Bounds Checking*

| Assertion Type | Description                                                | Default Behavior |
|----------------|------------------------------------------------------------|------------------|
| xindex-wr      | Write with indeterminate index (index containing X values) | Warning          |
| xindex-rd      | Read with indeterminate index (index containing X values)  | Warning          |
| index-wr       | Write with out-of-bounds index                             | Warning          |
| index-rd       | Read with out-of-bounds index                              | Warning          |

In Xprop simulations, the behavior and severity of the above four types of assertions can be controlled by a set of runtime Xprop assertion control Verilog system tasks. The system tasks operate on all assertions of the specified type in a design.

### Note:

For VHDL, these assertion controls are synonymous to each other as VHDL does not distinguish between an out-of-bounds index and an X index (`xindex`). It also does not distinguish between an indexed-read and an indexed-write.

- `$xprop_assert_on(assertion_type)`

Enable the specified assertion type.

- `$xprop_assert_off(assertion_type)`

Disable the specified assertion type. Assertion check of the specified assertion type is stopped until a subsequent call of the `$xprop_assert_on` task with the same assertion type is executed.

- `$xprop_assert_fatal(assertion_type)`

Set the severity of the specified assertion type to fatal. When an assertion of the specified type is triggered, the simulation terminates.

- `$xprop_assert_warn(assertion_type)`

Set the severity of the specified assertion type to warning. When an assertion of the specified type is triggered, the simulation continues.

**Note:**

The Xprop assertion control system tasks have no effect on Verilog behavior if the `-boundscheck` option is not used. The assertion control system tasks are always effective in VHDL.

## Detecting Unknown Values in Type Conversion Functions

VHDL provides type conversion functions to convert different types. For example, the `conv_integer` function is used to convert `std_logic_vector` to `integer` types. If the `std_logic_vector` contains an unknown value, then the type conversion results in a value of 0. This kind of conversion is considered as a lossy conversion because a 4-state value is converted to a 2-state value. Lossy conversion can mask potential design problems due to the lack of Xprop. VCS provide an assertion to detect lossy conversions. You can use the following methods to enable or disable the assertion:

- Specify an `ASSERT_IGNORE_XPROP_LOSSY_CONVERSION` declaration in the `synopsys_sim.setup` file as shown:

```
ASSERT_IGNORE_XPROP_LOSSY_CONVERSION<=[TRUE, FALSE]>
```

- Specify the `set_assert_off(LOSSY_CONV)` procedure as shown:

```
set_assert_on(LOSSY_CONV)
```

## Time Zero Initialization

When Xprop is enabled, initialization of a latch or flip-flop at time zero may result in an `x` output instead of the initialized value. At time zero, the SystemVerilog `always_latch` and `always_comb` processes are executed. An `x` value on the clock signal propagates through the device and may cause an indeterminate output.

## Enabling X-propagation on RTL Block Containing Asserts Having System Task Usage

Xprop does not instrument blocks containing the system tasks. The tasks like `$fatal` may be used inside `assert` block and would lead to Xprop not instrumenting an RTL logic

containing such asserts. The option `-unifiedInference=skipAssert` specifies Xprop to skip checking for constructs inside `assert` block for disabling the instrumentation.

## Handling Non-pure Functions Due to Static Lifetime

VCS provides an easy way to denote the lifetime of all user-defined functions that do not specify an explicit lifetime as automatic. Functions with a static lifetime (default) often create side-effects that require the compiler to consider those functions as non-pure. The side-effects due to static lifetime sometimes leads to simulation-synthesis mismatches. Moreover, they prevent the code that calls those functions from being instrumented by hardware-accurate simulation features, such as Xprop. To eliminate the side-effects due to static lifetime, VCS provides the `-fauto` compile-time option. When this option is specified, all Verilog functions that do not specify an explicit lifetime are automatically converted to automatic functions.

The default lifetime of Verilog functions defined within modules or interfaces is static (note that functions in program blocks or class methods are already automatic by default). This means that all the arguments, the return value, and all the variables declared within those functions are static and retain their values in between calls. The retention of values across calls may result in side-effects such that the behavior of the function depends not only on the current argument values, but also on the previous invocations. By definition, such functions are considered as non-pure functions.

For example:

```
module foo;
 function crc (input [31:0] data);
 reg tmp;
 tmp = tmp ^ data;
 crc = tmp;
 endfunction
endmodule
```

In the above example, the `crc` function is static by default. Therefore, the state of the `tmp` variable is retained through each invocation of the `crc` function. The result of each `crc` function call does not depend solely on the input argument `data`. Therefore, the `crc` function is a non-pure function.

You can use the `-fauto` compile-time option to change the lifetime of all functions that do not specify an explicit lifetime to automatic. The automatic lifetime eliminates the potential simulation-synthesis mismatches and enable the instrumentation of code that calls such functions in Xprop simulations. The behavior of the `-fauto` option is similar to declaring an automatic lifetime for the functions.

For example:

```
function automatic crc (input [31:0] data);
```

**Note:**

Functions that do indeed rely on the value retention side-effect for correct simulation need to be modified to specify the intended lifetime.

For example:

```
function static crc (input [31:0] data);
```

## Supporting UCLI Commands for X-Propagation Control Tasks

X-Propagation (Xprop) supports UCLI/TCL commands. The commands allow you to control the Xprop behavior at runtime. The commands return a success value or return a resultant value for a query, such as `$is_xprop_active()`.

### Use Model

This section describes how to use the UCLI commands.

- [UCLI Command to Specify the Merge Mode](#)
- [UCLI Command to Control Error Messages or Warning Messages](#)

### UCLI Command to Specify the Merge Mode

The syntax of the new UCLI command to specify the merge mode is as follows:

```
xprop {-is_active [inst_name]
| -merge_mode {vmerge|tmerge|xmerge|xprop}}
```

This command is equivalent to the Verilog `$set_x_prop()` and `$is_xprop_active()` system task calls, as well as the VHDL built-in package sub-programs `XPROPUSER.set_x_prop()` and `XPROP_USER.is_xprop_active()`.

For example,

```
xprop -is_active top.dut.core0dff
xprop -merge_mode vmerge
xprop -merge_mode xprop
```

**Note:**

- For a non-Xprop simulation, the command returns `False` and generates a warning message if the `-merge_mode` option is present.
- You must use either `-is_active` option or `-merge_mode` option. If neither or both options are provided, or if the value of the `-merge_mode` option is not valid, a help message is generated.

- The UCLI command allows you to provide both relative (to the current scope) instance name and absolute instance name. If no instance name is provided for the `-is_active` option, the command uses the current scope.
- If the `[inst_name]` option does not exist, a warning message is generated and the UCLI command returns `False`.

## UCLI Command to Control Error Messages or Warning Messages

The syntax of the new UCLI command to control error messages or warning messages is as follows:

```
report_violations -type {oob_index_rd | oob_index_wr| x_index_rd |
x_index_wr | lossy_conversion| enum_cast | ffdcheck} {-severity {warn |
error}| -on | -off}
```

The following command is equivalent to `$xprop_assert_{on,off,warn,fatal}()` or `XPROP_USER.xprop_assert_{on,off,warn,fatal}()` Verilog system task calls and VHDL `XPROP_USER` built-in package sub-programs.

**Note:**

- Multiple options are allowed, however, at least one option must be provided. If no option is provided, or illegal options or option values are provided, a help message is generated.
- If both `-on` and `-off` options are provided, a warning message is generated. The command returns `False` and the violation reporting state is not changed.
- Multiple options are allowed to the (singular) `-type` option, if presented in a TCL list (enclosed in braces and separated by spaces).
- For pure VHDL in non-Xprop mode, this command is not relevant under any circumstances. Hence, this command always generates a warning message and returns `False`.
- For Verilog and VHDL in non-Xprop mode, this command generates a warning message and returns `False` for `lossy_conversion`, `enum_cast` and `ffdcheck` violation types. The VHDL portion of the design is not affected by this command while running in non-Xprop mode.

---

## VHDL Two-State Objects in X-Propagation

VCS supports following two-state objects in Xprop:

- Integer (`XInteger`)
- Boolean (`XBoolean`)
- User-defined enumerations less than 256 values (`XEnum`)

In VHDL, integers are frequently used for array indexes as an array constraint is always of discrete type. As these integer values cannot represent `x` values during Xprop simulation, the `x` value is lost.

To propagate the `x` value, VCS promotes objects and object elements from 2-state types to X-state types. These types have a particular value that is defined to be the XValue.

This section discusses the following topics:

- [Supported XValues](#)
- [Supported Objects](#)
- [Supported Expressions](#)
- [Supported Attributes](#)
- [Limitations](#)

---

### Supported XValues

The following XValues are supported:

- `XInteger`: 0x7fffffed

You can override this XValue using the `vhdl_intx_override=<value>` directive in the Xprop configuration file. If the RTL simulation computes integer value that matches this `XInteger` value, then a runtime error (`VHDL_XPROP_INT_X`) is issued to notify this overlap.

- For the type `XEnum` and `XBoolean`, `X` value is an out-of-range value.

---

### Supported Objects

All the objects that are declared in the Xprop enabled entity are promoted.

## Supported Expressions

Expressions are promoted based on the operand type. The following list shows the list of operators and the corresponding operands:

- LRM Operators

XBoolean overload functions are created for the following Boolean operators:

- Logicals -- not, and, or, xor, nand, nor, xnor

XInteger overload functions are created for the following Integer operators:

- Arithmetic: +, -, /, \*, mod, rem
- Miscellaneous: \*\*, abs
- Shift: sll, srl, sla, sra, rol, ror

**Note:**

The shift operators are only available on unidimensional arrays of bit and Boolean types. An x value on shift or rotate count of a Boolean vector results in all xs on the Boolean vector. The bit vector values do not change with a indeterminate shift or rotate count.

- Relational: =, /=, <, <=, >, >=
- Std\_logic\_arith Operators
  - For the arithmetic + operator:
    - Function + (L: Xinteger, R signed) return signed
    - Function + (L: signed, R Xinteger) return signed
    - Function + (L: Xinteger, R unsigned) return unsigned
    - Function + (L: unsigned, R Xinteger) return unsigned
    - Function + (L: Xinteger, R signed) return std\_logic\_vector
    - Function + (L: Xinteger, R signed) return std\_logic\_vector
    - Function + (L: signed, R Xinteger) return std\_logic\_vector
    - Function + (L: Xinteger, R unsigned) return std\_logic\_vector
    - Function + (L: unsigned, R Xinteger) return std\_logic\_vector

For arithmetic + operator, the function returns a vector with all elements set to x when the integer input is the x value.

For the other arithmetic operators, – and \*, the function overload is analogous.

- For the relational = operator:
  - Function = (L: Xinteger, R: signed) return Xboolean
  - Function = (L: signed, R: Xinteger) return Xboolean
  - Function = (L: Xinteger, R: unsigned) return Xboolean
  - Function = (L: unsigned, R: XInteger) return Xboolean

For other relational operators /=, <, <=, >, >= the function returns a Xboolean x value when either the XInteger argument is x or when the signed or unsigned vector has any x element.

- For the conversion functions:
  - Function conv\_integer (signed) return XInteger
  - Function conv\_integer (unsigned) return XInteger
  - Function conv\_signed (XInteger) return signed
  - Function conv\_unsigned (XInteger) return unsigned
  - Function conv\_std\_logic\_vector (XInteger) return std\_logic\_vector

Conversion functions return all “x” when the argument is x.

- Std\_logic\_signed Operators
  - For the arithmetic + and – operator:
    - Function + (L: std\_logic\_vector, R XInteger) return std\_logic\_vector
    - Function + (L: XInteger, R std\_logic\_vector) return std\_logic\_vector
    - Function – (L: std\_logic\_vector, R XInteger) return std\_logic\_vector
    - Function – (L: XInteger, R std\_logic\_vector) return std\_logic\_vector

- For the relational = operator:

- Function = (L: std\_logic\_vector, R XInteger) return Xboolean
- Function = (L: XInteger, R std\_logic\_vector) return Xboolean

For other relational operators /=, <, <=, >, >= the function overload is analogous.

- For the conversion functions:

- Function conv\_integer (std\_logic\_vector) return XInteger

- Std\_logic\_unsigned Operators

- For the arithmetic + and – operator:

- Function + (L: std\_logic\_vector, R XInteger) return std\_logic\_vector
- Function + (L: XInteger, R std\_logic\_vector) return std\_logic\_vector
- Function – (L: std\_logic\_vector, R XInteger) return std\_logic\_vector
- Function – (L: XInteger, R std\_logic\_vector) return std\_logic\_vector

- For the relational = operator:

- Function = (L: std\_logic\_vector, R XInteger) return Xboolean
- Function = (L: XInteger, R std\_logic\_vector) return Xboolean

For other relational operators /=, <, <=, >, >= the function overload is analogous.

- For the conversion functions:

- Function conv\_integer (std\_logic\_vector) return XInteger

- Numeric\_std Operators

- For the arithmetic + and – operator:

- Function + (L: unsigned, R XNatural) return unsigned
- Function + (L: XNatural, R Unsigned) return unsigned
- Function – (L: signed, R XInteger) return signed
- Function – (L: XInteger, R signed) return signed

For other arithmetic operators –, \*, /, rem, mod the function overload is analogous.

- For the relational = operator:
  - Function = (L: XNatural, R unsigned) return Xboolean
  - Function = (L: unsigned, R XNatural) return Xboolean
  - Function = (L: XInteger, R signed) return Xboolean
  - Function = (L: signed, R XInteger) return Xboolean
- For other relational operators /=, <, <=, >, >= the function overload is analogous.
- For the shift or rotate operators:
  - Function sll (L: signed, Count XInteger) return signed
  - Function sll (L: unsigned, Count XInteger) return unsigned
- For other shift or rotate operators srl, rol, ror the function overload is analogous.
- For the shift or rotate functions:
  - Function shift\_left (L: signed, Count XNatural) return signed
  - Function shift\_left (L: unsigned, Count XNatural) return unsigned
- For other shift or rotate functions shift\_right(), rotate\_left(), rotate\_right() the function overload is analogous.
- For the conversion functions:
  - Function to\_integer (unsigned) return XNatural
  - Function to\_integer (signed) return XInteger
  - Function to\_unsigned (Arg: XNatural; Size: natural) return unsigned
  - Function to\_signed (Arg: XInteger; Size: natural) return signed
- For the resize functions:
  - Function resize (Arg: unsigned; new\_size: XNatural) return unsigned
  - Function resize (Arg: signed; new\_size: XNatural) return signed

## Supported Attributes

Attributes need to be extended to support the XValue for XInteger, XEnum and XBoolean:

The following supported attributes result in an `X` if the input parameter is `X`:

- POS
- VAL
- SUCC
- PRED
- LEFTOF
- RIGHTOF

The following supported attributes propagate `X`, if the prefix is promoted to X-type:

- LAST\_VALUE
- DRIVING\_VALUE
- DELAYED

In addition, the following attributes are also supported:

- IMAGE: For `X`, it returns the string `<basetype name>-X`. For example, for Natural, it returns INTEGER-X.
- VALUE: It returns `X` value when the input is the string returned by image on `X` input.

## Limitations

The feature has the following limitations:

- The supported two-state objects for promotion to x-state types are `enums` with less than 256 values, `integer` and `boolean`.
- `enums` in standard packages, with the exception of `boolean`, are not promoted.
- The `case` statements support only `enums` with less than 256 values. `enums` ranging 256 values or more disable the indeterminate execution of the `case` statements.
- By default, the initial value of two-state is '`LEFT`'.

- The two-state `x` value that crosses an Xprop -> Non-Xprop boundary or an VHDL Xprop -> Verilog boundary takes the integer value representing `x`. For `enums`, the value is 255 and for integer it is the customized value.
  - The corruption value in NLP is the `x` value, if the object is declared in an Xprop scope. For objects declared in a non-xprop scope, the corruption value is '`LEFT`'.

## X-Propagation Code Examples

X-Propagation (Xprop) changes the simulation semantics of the standard HDL conditional constructs. This section describes the Xprop simulation behavior of the following code examples.

- [If Statement](#)
- [Case Statement](#)
- [Edge Sensitive Expression](#)
- [Latch](#)

### If Statement

#### Verilog Example

The following Verilog code example of an `if` statement represents a simple multiplexer:

```
always@*
 if(s)
 r = a;
 else
 r = b;
```

Table 24     Xprop Merge Mode Truth Table for if Statement

| s | a | b | vmerge | tmerge | xmerge |
|---|---|---|--------|--------|--------|
| X | 0 | 0 | 0      | 0      | X      |
| X | 0 | 1 | 1      | X      | X      |
| X | 1 | 0 | 0      | X      | X      |
| X | 1 | 1 | 1      | 1      | X      |

**Table 24** describes the truth table of the above code example when the control signal  $s$  of the if statement is unknown with a value of  $X$ .

Under the vmerge mode, the standard HDL simulation semantics is used. When the control signal  $s$  is unknown, the output signal  $r$  is always assigned the value of the else statement. In this case, the value of  $r$  is the same as signal  $b$ .

Under the tmerge mode, the simulation semantics is modified when the control signal  $s$  is unknown. In this scenario, two of the case statements are executed, one considering  $s=0$  and one considering  $s=1$ . The output values that result from both statements are then merged. If the values of the output signal  $r$  are same in both conditions, then the merged value of  $r$  is same as in either condition  $s=0$  and  $s=1$ . If the values of the output signal  $r$  are different then the merged value of  $r$  is  $X$ . Thus, if the input signals  $a$  and  $b$  are same, the value of  $r$  is same as  $a$  (or  $b$ ). If  $a$  and  $b$  are different, the value of  $r$  is  $X$ .

In the xmerge mode, the value of the output signal  $r$  is always  $X$  when the control signal  $s$  is unknown and there is an active driver. In case no active driver is present for the signal, the previous value is retained.

## VHDL Example

The following VHDL code example of an if statement represents a simple multiplexer:

```
entity MUX is
 Port (s : IN std_logic;
 a : IN std_logic;
 b : IN std_logic;
 r : OUT std_logic);
end MUX;

architecture BEHAVIORAL of MUX is

begin
 process (s, a, b)
 begin

 if (s = '1') then
 r <= a;
 else
 r <= b;
 end if;
 end process;
end BEHAVIORAL;
```

**Table 25** Xprop Merge Mode Truth Table for if Statement

| <b>s</b> | <b>a</b> | <b>b</b> | <b>r in vmerge</b> | <b>r in tmerge</b> | <b>r in xmerge</b> |
|----------|----------|----------|--------------------|--------------------|--------------------|
| X        | 0        | 0        | 0                  | 0                  | X                  |

*Table 25 Xprop Merge Mode Truth Table for if Statement (Continued)*

| s | a | b | r in vmerge | r in tmerge | r in xmerge |
|---|---|---|-------------|-------------|-------------|
| X | 0 | 1 | 1           | X           | X           |
| X | 1 | 0 | 0           | X           | X           |
| X | 1 | 1 | 1           | 1           | X           |

**Table 25** describes the truth table of the above code example when the control signal *s* of the *if* statement is unknown with a value of *X*.

Under the *vmerge* mode, the standard HDL simulation semantics is used. When the control signal *s* is unknown, the output signal *r* is always assigned the value of the *else* statement. In this case, the value of *r* is the same as signal *b*.

Under the *tmerge* mode, the simulation semantics is modified when the control signal *s* is unknown. In this scenario, two of the case statements are executed, one considering *s=0* and one considering *s=1*. The output values that result from both statements are then merged. If the values of the output signal *r* are same in both conditions, then the merged value of *r* is same as in either condition *s=0* and *s=1*. If the values of the output signal *r* are different, then the merged value of *r* is *X*. Thus, if the input signals *a* and *b* are same, the value of *r* is same as *a* (or *b*). If *a* and *b* are different, the value of *r* is *X*.

In the *xmerge* mode, the value of the output signal *r* is always *X* when the control signal *s* is unknown and there is an active driver. In case no active driver is present for the signal, the previous value is retained.

## Case Statement

### Verilog Example

The following Verilog code example of a *case* statement represents a simple multiplexer:

```
case (s)
 1'b0: r = a;
 1'b1: r = b;
endcase
```

*Table 26 Xprop Merge Mode Truth Table for Case Statement*

| s | a | b | vmerge | tmerge | xmerge |
|---|---|---|--------|--------|--------|
| X | 0 | 0 | r(t-1) | 0      | X      |

**Table 26      Xprop Merge Mode Truth Table for Case Statement  
(Continued)**

| s | a | b | vmerge | tmerge | xmerge |
|---|---|---|--------|--------|--------|
| X | 0 | 1 | r(t-1) | X      | X      |
| X | 1 | 0 | r(t-1) | X      | X      |
| X | 1 | 1 | r(t-1) | 1      | X      |

**Table 26** describes the truth table of the above code example when the control signal *s* of the case statement is unknown with a value of *X*.

In the vmerge mode, the standard HDL simulation semantics is used. When the control signal *s* is unknown, the value of the output signal *r* remains the same as before the case statement is executed.

In the tmerge mode, when the control signal *s* is unknown, the case statement is executed twice, once with *s=0* and once with *s=1*. The output values from both conditions are computed and merged. If the values of the output signal *r* are same in both conditions, then the merged and the final value of *r* is same as in both conditions *s=0* and *s=1*. If the values of the output signal *r* are different in both conditions, then the merged and the final value of *r* is determined by the values of the input signals *a* and *b*. If *a* and *b* are same, the merged and the final value of *r* is the same as *a* and *b*. If *a* and *b* are different, the merged and the final value of *r* is *X*.

In the xmerge mode, the value of the output signal *r* is always *X* when the control signal *s* is unknown and there is an active driver. In case no active driver is present for the signal, the previous value is retained.

## VHDL Example

The following VHDL code example of a `case` statement represents a simple multiplexer:

```

library IEEE;
use IEEE.std_logic_1164.all;
use IEEE.std_logic_arith.all;

entity SIMPLECASE is
 Port (s : IN std_logic;
 a : IN std_logic;
 b : IN std_logic;
 r : OUT std_logic);
end SIMPLECASE;

architecture BEHAVIORAL of SIMPLECASE is

```

```

begin
 process (s, a, b)
begin
 case s is
 when '0' => r <= a;
 when '1' => r <= b;
 when others => null;
 end case;
end process;

end BEHAVIORAL;

```

*Table 27 Xprop Merge Mode Truth Table for Case Statement*

| s | a | b | r in vmerge | r in tmerge | r in xmerge |
|---|---|---|-------------|-------------|-------------|
| X | 0 | 0 | r(t-1)      | 0           | X           |
| X | 0 | 1 | r(t-1)      | X           | X           |
| X | 1 | 0 | r(t-1)      | X           | X           |
| X | 1 | 1 | r(t-1)      | 1           | X           |

**Table 27** describes the truth table of the above code example when the control signal `s` of the `case` statement is unknown, with a value of `X`.

In the vmerge mode, the standard HDL simulation semantics is used. When the control signal `s` is unknown, the value of the output signal `r` remains the same as before the `case` statement is executed.

In the tmerge mode, when the control signal `s` is unknown, the `case` statement is executed twice, once with `s=0` and once with `s=1`. The output values from both conditions are computed and merged. If the values of the output signal `r` are same in both conditions, then the merged and the final value of `r` is same as in both conditions `s=0` and `s=1`. If the values of the output signal `r` are different in both conditions, then the merged and the final value of `r` is determined by the values of the input signals `a` and `b`. If `a` and `b` are same, the merged and the final value of `r` is same as `a` and `b`. If `a` and `b` are different, the merged and the final value of `r` is `X`.

In the xmerge mode, the value of the output signal `r` is always `X` when the control signal `s` is unknown and there is an active driver. In case no active driver is present for the signal, the previous value is retained.

## Unique/Priority Case Variants

SystemVerilog allows `unique` or `priority` directives to be applied to a `case` statement. Synthesis also allows a `case` to be specified as `full_case` or `parallel_case`.

Handling of these case variants requires that indeterminate bits in the case selector can cause more than one branch to match ambiguously, which in turn alter the condition under which `unique/priority` assertions are triggered. The ambiguity due to multiple matches is naturally handled by executing all the matching branches and merging the resulting outputs. This behavior is not modified by the triggering of an assertion. These assertions are triggered as follows:

**Priority:** Assertion is triggered when none of the branches matches deterministically. Therefore, zero matches or multiple ambiguous matches (due to a comparison resulting in X) that do not completely cover the Boolean space of the indeterminate bits involved does not constitute a deterministic match and triggers the assertion.

**Unique:** Assertion is triggered when either none of the branches matches deterministically (as for `priority` above), or when more than one branch matches non-ambiguously. Therefore, multiple exact matches trigger the assertion, whereas a single exact match does not trigger the assertion.

To demonstrate the assertion semantics, consider the following two examples:

```
priority case (sel)
 2'b00 : a = 1;
 2'b01 : a = 2;
 3'b11 : a = 3;
endcase

unique case (1'b1)
 b + c : d = 1;
 b - c : d = 2;
endcase
```

When the `sel` condition is `2'b0X`, the first example (`priority case`) ambiguously matches the first and the second branches. Therefore, the output is `a=merge(1, 2)`. Also, since the Boolean space of the indeterminate selector is completely covered by the two matches, no assertion is triggered. If, however, the `sel` condition is `2'b1X`, then a priority assertion is triggered and the output is `a=merge(3, a)`.

If `b` and `c` have the values 1 and 0 respectively, the second example (`unique case`) unambiguously matches both branches, thereby triggering an unique assertion. Conversely, if `b` or `c` are X, then the example ambiguously matches the same two branches and triggers a (no match) unique assertion.

## Edge Sensitive Expression

### Verilog Example

In standard Verilog RTL simulations, a positive edge transition is triggered for the following value changes in a clocking signal:

```
0 -> 1
0 -> X
0 -> Z
X -> 1
Z -> 1
```

If X is considered as either a 0 or a 1 value, then in the 0->X transition, X may represent a value of 0, which denotes no transition. And, X may represent a value of 1, which denotes a positive edge transition. An Xprop simulation considers both these behaviors and merges the outcome.

The following Verilog code example of an edge sensitive expression represents a simple D flip-flop with an inactive reset:

```
always@(posedge clk, negedge rst)
 if (!rst)
 q <= 1'b0;
 else
 q <= d;
```

*Table 28 Xprop Merge Mode Truth Table for D Flip-Flop*

| clk    | vmerge | tmerge          | xmerge |
|--------|--------|-----------------|--------|
| 0 -> 1 | d      | d               | d      |
| 0 -> X | d      | merge(d,q(t-1)) | X      |
| 0 -> Z | d      | merge(d,q(t-1)) | X      |
| X -> 1 | d      | merge(d,q(t-1)) | X      |
| Z -> 1 | d      | merge(d,q(t-1)) | X      |

**Table 28** describes the truth table of the above code example with different value transitions of the clocking signal *clk*.

In all merge modes, if the clocking signal *clk* is changing from 0 to 1, then a positive edge transition is triggered. The output signal of the D flip-flop *q* is assigned the value of the input signal *d*.

For all other clocking signal transitions, the output signal  $q$  is assigned the value of the input signal  $d$  in the vmerge mode. The output signal  $q$  is assigned the value of  $x$  in the xmerge mode. In the tmerge mode, the current value of the output signal  $q$  is merged with the input signal  $d$ , as described in the tmerge column of the truth table [Table 24](#). Then, the merged value is assigned to  $q$ .

## VHDL Example

In standard VHDL RTL simulations, a positive edge transition is triggered for the following value changes in a clocking signal:

```
0 -> 1
0 -> X
0 -> Z
X -> 1
Z -> 1
```

If  $X$  is considered as either a  $0$  or a  $1$  value, then in the  $0->X$  transition,  $X$  may represent a value of  $0$ , which denotes no transition. And,  $X$  may represent a value of  $1$ , which denotes a positive edge transition. An Xprop simulation considers both these behaviors and merges the outcome.

The following VHDL code example of an edge sensitive expression represents a simple D flip-flop with an inactive reset:

```
library ieee;
use ieee.std_logic_1164.all;

entity FLOP is

 generic (
 width : integer := 4);

 port (
 d : in std_logic_vector(width-1 downto 0);
 rst : in std_logic;
 clk : in std_logic;
 q : out std_logic_vector(width-1 downto 0));
end FLOP;

architecture arc of FLOP is

begin
 process (clk, rst)
 begin
 if rst = '1' then
 q <= "0000";
 elsif (rising_edge (clk)) then
 q <= d;
 end if;
 end process;
```

```
end arc;
```

*Table 29 Xprop Merge Mode Truth Table for D Flip-Flop*

| clk    | q in vmerge | q in tmerge     | q in xmerge |
|--------|-------------|-----------------|-------------|
| 0 -> 1 | d           | d               | d           |
| 0 -> X | d           | merge(d,q(t-1)) | X           |
| 0 -> Z | d           | merge(d,q(t-1)) | X           |
| X -> 1 | d           | merge(d,q(t-1)) | X           |
| Z -> 1 | d           | merge(d,q(t-1)) | X           |

**Table 29** describes the truth table of the above code example with different value transitions of the clocking signal `clk`.

In all merge modes, if the clocking signal `clk` is changing from 0 to 1, then a positive edge transition is triggered. The output signal of the D flip-flop `q` is assigned the value of the input signal `d`.

For all other clocking signal transitions, the output signal `q` is assigned the value of the input signal `d` in the vmerge mode. The output signal `q` is assigned the value of `x` in the xmerge mode. In the tmerge mode, the current value of the output signal `q` is merged with the input signal `d`, as described in the tmerge column of the truth table [Table 25](#). Then, the merged value is assigned to `q`.

#### Note:

The standard VHDL behavior for the above flip-flop is to not store a new value when the clock transitions from either `0->x` or `x-> 1`, whereas the standard behavior of the corresponding Verilog model shown above is to store the new value. Xprop semantics, therefore, normalize the behavior of the two languages.

## Latch

### Verilog Example

The following Verilog code example of an `if` statement without an `else` branch represents a simple latch.

```
always@(*)
 if(g)
 q <= d ;
```

*Table 30 Xprop Merge Mode Truth Table for Latch*

| g | d | vmerge | tmerge          | xmerge |
|---|---|--------|-----------------|--------|
| X | 0 | q(t-1) | merge(d,q(t-1)) | X      |
| X | 1 | q(t-1) | merge(d,q(t-1)) | X      |

**Table 30** describes the truth table of the above code example when the control signal *g* of the if statement is unknown with a value of *x*.

In the vmerge mode, when the control signal *g* is unknown, the value of the output signal *q* is unchanged. In the tmerge mode, the current value of the output signal *q* is merged with the input signal *d*, as described in the tmerge column of the truth table [Table 24](#). The merged value assigned to *q* when the control signal *g* is unknown thus depends on the values of both *q* and *d*.

In the xmerge mode, the value of the output signal *r* is always *x* when the control signal *s* is unknown and there is an active driver. In case no active driver is present for the signal, the previous value is retained.

## VHDL Example

The following VHDL code example represents a simple latch:

```
library ieee;
use ieee.std_logic_1164.all;

entity LATCH is

 generic (
 width : integer := 4);

 port (
 d: in std_logic;
 g: in std_logic;
 q: out std_logic);
end LATCH;

architecture arc of LATCH is
begin
 process (g, d)
 begin
 if g = '1' then
 q <= d;
```

```

 end if;
end process;

end arc;
```

*Table 31 Xprop Merge Mode Truth Table for Latch*

| <b>g</b> | <b>d</b> | <b>q in vmerge</b> | <b>q in tmerge</b> | <b>q in xmerge</b> |
|----------|----------|--------------------|--------------------|--------------------|
| X        | 0        | q(t-1)             | merge(d,q(t-1))    | X                  |
| X        | 1        | q(t-1)             | merge(d,q(t-1))    | X                  |

**Table 31** describes the truth table of the above code example when the control signal  $g$  of the `if` statement is unknown with a value of X.

In the vmerge mode, when the control signal  $g$  is unknown, the value of the output signal  $q$  is unchanged. In the tmerge mode, the current value of the output signal  $q$  is merged with the input signal  $d$ , as described in the tmerge column of the truth table **Table 25**. The merged value assigned to  $q$  when the control signal  $g$  is unknown thus depends on the values of both  $q$  and  $d$ .

In the xmerge mode, the value of the output signal  $x$  is always X when the control signal  $s$  is unknown and there is an active driver. In case no active driver is present for the signal, the previous value is retained.

## Support for Active Drivers in X-Propagation

Verdi supports active driver tracing functionality for the designs compiled with X-Propagation (using the `-xprop` option). Verdi supports active drivers tracing for X-Propagation (Xprop) in combinational logic, latches, and flip-flops. This section describes the following topics:

- [Combinational Logic](#)
- [Latches](#)
- [Flip-flops](#)
- [Key points to Note](#)

### Combinational Logic

Verdi supports active drivers tracing for Xprop in combinational logic. For tmerge, all drivers that are used for Xprop merge function are shown as active. For xmerge, all drivers from the branches of control structure with unknown condition value are shown as active.

For example, consider the following test case (`comb_logic.sv`) that contains the combinational logic:

**Example 65 comb\_logic.sv**

```
module top();
 reg op;
 reg [2:0] out;
 reg [1:0] a,b;
 dut dut1 (op,a,b,out);
 initial begin
 op = 1'b0; a = 2'b11; b = 2'b10;
 #1 op = 1; a = 2'b00;
 #3 a = 1; b = 2'b01; op = 1'bx;
 #2 op = 1; b = 2'b11;
 #1 op = 1'bx;
 #2 op = 0;
 end
endmodule

module dut(input op,a,b, output out);
 logic op;
 logic [1:0] a,b;
 logic [2:0] out;
 always_comb begin
 if (op)
 out = a + b;
 else
 out = a - b;
 end
endmodule
```

Perform the following steps:

1. Compile the `comb_logic.sv` code shown in [Example 65](#) as follows:

```
% vcs -sverilog -debug_access+drivers -kdb comb_logic.sv
```

2. Invoke the Verdi GUI using the following command:

```
% simv -gui
```

3. In console pane, use the following command:

```
verdi> fsdbDumpvars 0 "top" +all +trace_process
```

```
verdi> run
```

4. Perform **Trace Drivers**.

Verdi displays active drivers for the combinational logic signals that cause Xprop in the *DriverLoad* Pane, as shown in [Figure 131](#).

*Figure 131 Viewing Active Drivers of the Combinational Logic Signals*

| Signals/Drivers/Loads |       |      |                   |          |
|-----------------------|-------|------|-------------------|----------|
|                       | Value | Time | File(Line)        | Scope    |
| out                   | 2->x  | 4    | comb_logic.sv(19) | top.dut1 |
| out = a + b;          |       | 4    | comb_logic.sv(22) | top.dut1 |
| out = a - b;          |       | 4    | comb_logic.sv(24) | top.dut1 |

Verdi displays both active and inactive contributor signals for the driver. Active contributor is a contributor that has a value change and impacts the value of the traced signal. You can expand the active contributor signal, as shown in [Figure 132](#), to further trace the origin of X.

*Figure 132 Tracing Origin of X*

| Signals/Drivers/Loads               |       |      |                   |          |
|-------------------------------------|-------|------|-------------------|----------|
|                                     | Value | Time | File(Line)        | Scope    |
| out                                 | 2->x  | 4    | comb_logic.sv(19) | top.dut1 |
| out = a + b;                        |       | 4    | comb_logic.sv(22) | top.dut1 |
| out = a - b;                        |       | 4    | comb_logic.sv(24) | top.dut1 |
| a                                   | 0->1  | 4    | comb_logic.sv(18) | top.dut1 |
| b                                   | 2->1  | 4    | comb_logic.sv(18) | top.dut1 |
| op                                  | 1->x  | 4    | comb_logic.sv(17) | top.dut1 |
| #1 op = 1'b0; a = 2'b11; b = 2'b10; |       | 4    | comb_logic.sv(8)  | top      |
| #1 op = 1; a = 2'b00;               |       | 4    | comb_logic.sv(9)  | top      |
| #3 a = 1; b = 2'b01; op = 1'bx;     |       | 4    | comb_logic.sv(10) | top      |
| #2 op = 1; b = 2'b11;               |       | 4    | comb_logic.sv(11) | top      |
| #1 op = 1'bx;                       |       | 4    | comb_logic.sv(12) | top      |
| #2 op = 0;                          |       | 4    | comb_logic.sv(13) | top      |

#### Note:

In the xmerge mode, all RHS contributors are displayed as inactive. In tmerge mode, the contributors with value change that can impact RHS value, are displayed as active. Otherwise, they are displayed as inactive.

## Latches

Verdi supports active drivers tracing for Xprop in latches. Verdi displays the control signals as contributors for the drivers.

For example, consider the following test case (`latch.sv`) that contains the latch signals:

**Example 66** `latch.sv`

```
module top;
 reg clk,y,a;
 dut dut1 (clk,a,y);
 initial begin
 clk = 1'b1;
 a = 1'b0;
 #1 clk = 1'b0;
 #1 a = 1'b1;
 #1 clk = 1'bx;
 #1 clk = 1'b1;
 end
endmodule

module dut(input clk,a, output y);
 logic clk,a,y;
 always_latch begin
 if (clk)
 y = a;
 end
endmodule
```

Perform the following steps:

1. Compile the `latch.sv` code shown in [Example 66](#) as follows:

```
% vcs -sverilog -debug_access+drivers -kdb latch.sv
```

2. Invoke the Verdi GUI using the following command:

```
% simv -gui
```

3. In console pane, use the following command:

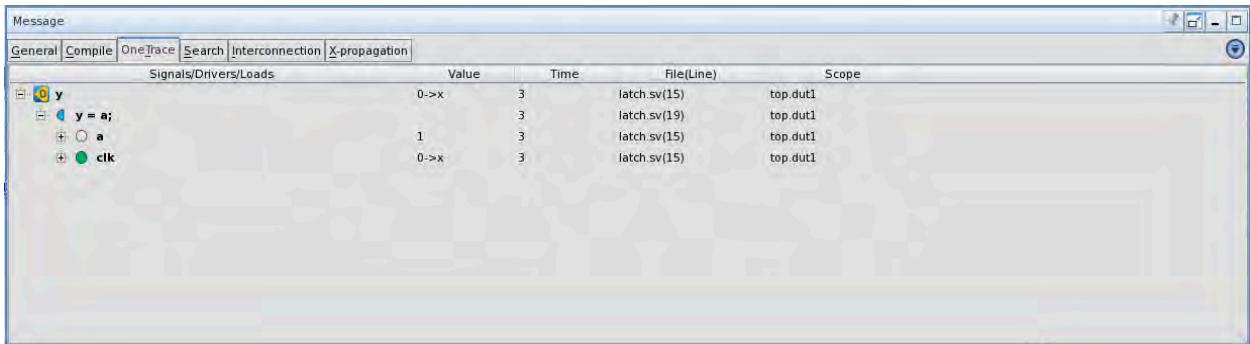
```
verdi> fsdbDumpvars 0 "top" +all +trace_process
```

```
verdi> run
```

4. Perform **Trace Drivers**.

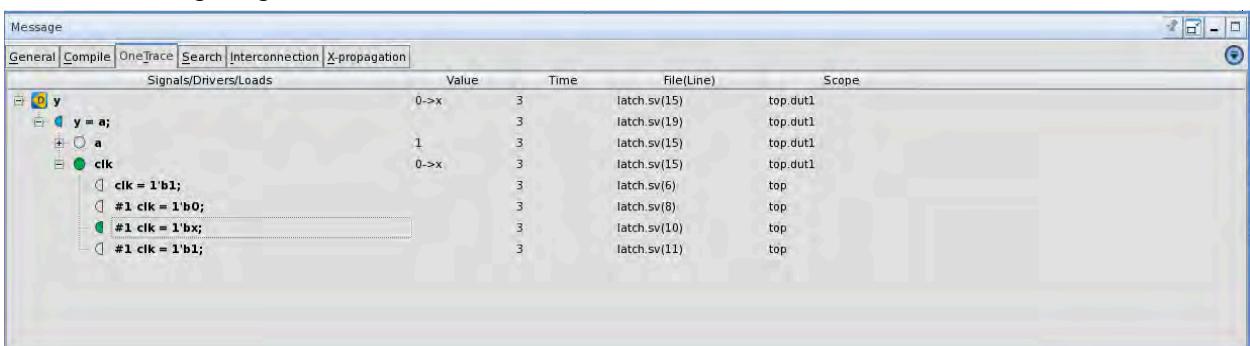
Verdi displays active drivers for the latch signals that cause Xprop in the *DriverLoad* Pane, as shown in [Figure 133](#).

*Figure 133 Viewing Active Drivers of the Latch Signals*



Verdi displays both active and inactive contributor signals for the driver. You can expand the active contributor signal, as shown in [Figure 134](#), to further trace the origin of X.

*Figure 134 Tracing Origin of X*



## Flip-flops

Verdi supports active drivers tracing for Xprop in flip-flops. Verdi also displays signals from the edge-sensitivity list, which has transition to X, as contributors for the drivers.

For example, consider the following test case (`flip_flop.sv`):

*Example 67 flip\_flop.sv*

```
module top;
 reg clk;
 reg q,d;
 dut dut1 (clk,d,q);
 always #2 clk = ~clk;
 initial begin
 clk = 1'b0;
 d = 1'b1;
```

## Chapter 12: Using X-Propagation Support for Active Drivers in X-Propagation

```

#4 d = 1'b0;
#1 clk = 1'bx;
#1 d = 1'b0;
#1 clk = 1'b1;
#1 d = 1'b1;
#10 $finish();
end
endmodule

module dut(input clk,d,output q);
 logic clk,q,d;
 always_ff@(posedge clk) q <= d;
endmodule

```

Perform the following steps:

1. Compile the `flip_flop.sv` code shown in [Example 67](#) as follows:

```
% vcs -svverilog -debug_access+drivers -kdb flip_flop.sv
```

2. Invoke the Verdi GUI using the following command:

```
% simv -gui
```

3. In console pane, use the following command:

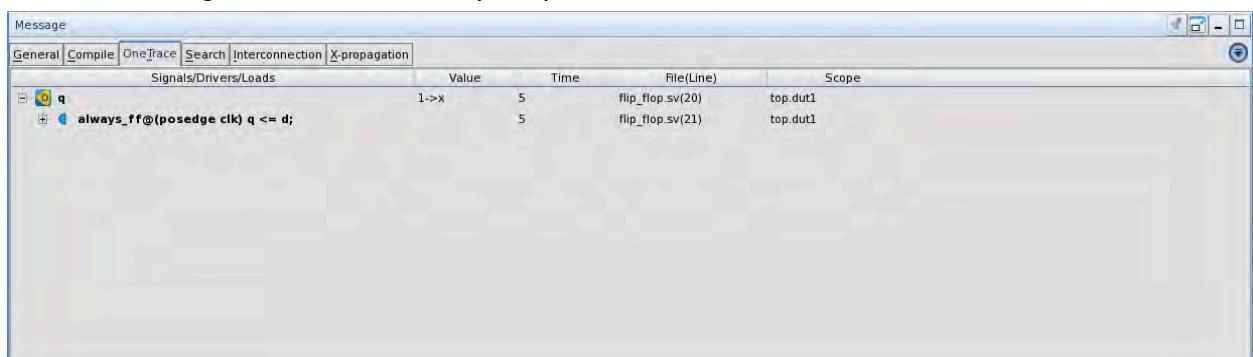
```
verdi> fsdbDumpvars 0 "top" +all +trace_process
```

```
verdi> run
```

4. Perform **Trace Drivers**.

Verdi displays active drivers for the flip-flops that cause Xprop in the *DriverLoad* Pane, as shown in [Figure 135](#).

**Figure 135 Viewing Active Drivers of Flip-Flops**



Verdi does not display signals from always block sensitivity list as contributor except in this case, where a signal from the process sensitivity list (for example, clock) has

an unclean edge and causes Xprop. You can expand the active contributor signal, as shown in [Figure 136](#), to further trace the origin of X.

*Figure 136 Tracing Origin of X*

| Signals/Drivers/Loads           | Value | Time | File(Line)       | Scope    |
|---------------------------------|-------|------|------------------|----------|
| clk                             | 0->x  | 5    | flip_flop.sv(20) | top.dut1 |
| always #2 clk = ~clk;           |       | 5    | flip_flop.sv(5)  | top      |
| clk = 1'b0;                     |       | 5    | flip_flop.sv(7)  | top      |
| #1 clk = 1'bx;                  |       | 5    | flip_flop.sv(10) | top      |
| #1 clk = 1'b1;                  |       | 5    | flip_flop.sv(12) | top      |
| q                               | 1->x  | 5    | flip_flop.sv(3)  | top      |
| always_ff@(posedge clk) q <= d; |       | 5    | flip_flop.sv(21) | top.dut1 |
| clk                             | 0->x  | 5    | flip_flop.sv(20) | top.dut1 |
| d                               | 1->0  | 4    | flip_flop.sv(20) | top.dut1 |

## Key points to Note

- The change of the merge function at runtime is ignored. That is, analysis is done using the merge function specified at compile time.

---

## Support for Ternary Operator

The behavior of the ternary operator may be more optimistic than some of its possible gate-level implementations. Therefore, when the `xmerge` function is in effect, the behavior of the ternary operator is modified to be more pessimistic.

Specifically, when `xmerge` is in effect, `(cond ? a : b)` becomes:

```
(|cond === 'X) ? 'X : (cond ? a : b)
```

When the `tmerge` function is in effect, the ternary operator remains unchanged.

---

## Renaming `xprop.log` File

VCS Xprop supports the following option to provide a user-defined file name or file path, so that the `xprop.log` file gets renamed and directed to the given location.

```
-xprop=logfilename=<file>
```

Both relative and absolute paths are supported.

In the example `-xprop=logfilename=file1.log` file, the `file1.log` file is created in the compile directory itself.

In `-xprop=logfilename=temp/file1.log` file, (path relative to the compile directory), the `file1.log` file is created inside the `temp` directory. If the `temp` directory does not exist or does not have write permission, the error message below is issued, and compilation terminates.

```
Error-[NOWRITE_PERM] No write permission on file
You don't have write permission for log file 'temp/file2.log'.
Please make sure permission exist
```

In `-xprop=logfilename=/remote/path/to/directory/file1.log` file, (absolute path is provided), the `file1.log` file is created at the path mentioned with the option name.

If the provided file path does not exist or does not have proper write permission, the error message below is issued, and compilation terminates.

```
Error-[NOWRITE_PERM] No write permission on file
You don't have write permission for log file
 '/remote/scratch28/28July/file1.log'.
Please make sure permission exist
```

## Limitations

Following are the limitations of this feature:

- This feature is only supported for pure Verilog designs. VHDL and MX designs are not supported.
- The following NYI message is issued if a VHDL or Mixed design with this Xprop option is encountered.

```
Warning-[XPROPLOG-RENAMING-NYI] Not yet implemented
Xprop Log file renaming is not supported for VHDL and MX designs.
```

## Limitations

1. X-Propagation is not supported with the following VCS feature:
  - `+vcst+initreg` option described in the section [Initializing Verilog Variables, Registers, and Memories](#).
2. X-Propagation support for the following VCS features have limitations:
  - Code coverage in Verilog
 

Code coverage does not exclude branches that are executed ambiguously, that is, under control of an X. Therefore, code coverage results may appear to overestimate coverage when Xprop is enabled.
  - A loop with non-constant bounds

The loops with variable bounds are not synthesizable. Xprop requires loop bounds to be constant or constant expressions for the loops to be instrumented. Along with the use of explicit variables of 4 state types, functions returning non-constant values are also treated as non-constant expressions.

- VHDL specific limitations

Delays are not supported in VHDL code.

3. Limitations that disable Xprop at the call site and setup the parent chain. These limitations at the call site do not prevent instrumentation of the procedure body. If a call is determined as non-xpropable, all the internal structural statements are also disabled for Xprop.

- wait() Statements

The `wait()` statement cannot appear in the Xprop region because it is not possible to merge delayed values and normal values

- Signal output parameters

Signal output parameters indicate a signal assignment within the procedure. Merging and signal final assignment are completed at the end of the last conditional statement within the sub-program. As merging is not completed at a call site, the call site cannot be within an Xprop region.

- Up-level references and external names

The references can be made to objects that are undergoing merging in different contexts. It is not possible to know which call site context the merging should be utilized from within the sub-program. Thus, the call site to the sub-program with up-level or external references must exist in a non-xprop region, where no merging is possible.

- Side Effects

It is not possible to merge side effects. Thus, the call must occur in a non-xprop region.

- Un-instrumented sub-program with output arguments

A sub-program that is not instrumented cannot be executed in a non-deterministic method. So, its call site should not be in an Xprop region.

4. Limitations that disables Xprop within the sub-program body:

- Return statements

Return statements execute immediately irrespective of whether the branch is executed in a deterministic way or not.

**Note:**

This limitation only applies to Verilog methods. For VHDL, return statement is supported when the switch `-xprop=flowctrl` is specified.

- Dynamic for loop bounds

Unlike in a process where the loop bounds as the loop cannot be dynamic, the rule is relaxed for sub-programs because unconstrained types are statically constrained at the call site. The loop can then iterate over the static bounds, however, the bounds appear dynamic from within a sub-program.

- Edges

Clock edges are not detected within a sub-program. Flip-flops in sub-programs are not inferred. A clock edge in a sub-program is treated as a normal condition within the sub-programs.

- Unlike in a process, `translate_on` or `translate_off` around a sub-program does not suppress Xprop.
- The `xprop_on` or `xprop_off` pragma does not suppress Xprop in a sub-program.

5. For composite types, only `integral`, `std_logic`, `enum`, `boolean` and `record` types are supported.

**Note:**

For Integer, enum and Boolean types, instrumentation only happens if the `synopsys_sim.setup` file contains an entry `XPROP_ANALYSIS_CHECK=true`.

6. For memories, MDA of `std_logic` base types and vector of vectors are supported. The MDA of `std_logic` include `std_logic`, `std_ulogic`, `std_logic_vector`, and `std_ulogic_vector`.

**Note:**

For Integer, enum and boolean types, instrumentation only happens if the `synopsys_sim.setup` file contains an entry `XPROP_ANALYSIS_CHECK=true`.

7. Other `non-std_[u]logic` based multidimensional arrays or arrays of arrays are not supported.
8. Record types that contain field of unsupported types are not supported.
9. The `$set_x_prop` Verilog system task or VHDL `set_x_prop` does not trigger process blocks to re-evaluate. Therefore, unless there is a change in the input data, output does not reflect the behavior of the new merge scheme.
10. For unique/priority case or case with default branch, instrumentation is not done if bit-length of non-constant case expressions exceeds 8 bits.

# 13

## Gate-Level Simulation

---

This chapter contains the following sections:

- [SDF Annotation](#)
  - [Precompiling an SDF File](#)
  - [SDF Configuration File](#)
  - [Delays and Timing](#)
  - [Using the Configuration File to Disable Timing](#)
  - [Using the timopt Timing Optimizer](#)
  - [Using Scan Simulation Optimizer](#)
  - [Negative Timing Checks](#)
  - [Using VITAL Models and Netlists](#)
  - [Additional Diagnostics](#)
- 

### SDF Annotation

The Open Verilog International Standard Delay File (OVI SDF) specification provides a standard ASCII file format for representing and applying delay information. VCS supports OVI versions 1.0, 1.1, 2.0, 2.1, and 3.0 of this specification.

In the SDF format, a tool can specify intrinsic delays, interconnect delays, port delays, timing checks, timing constraints, and pulse control (PATHPULSE).

When VCS reads an SDF file, it “back-annotates” delay values to the design, that is, it adds delay values or changes the delay values specified in the source files.

- [Using the Unified SDF Feature](#)
- [Using the \\$sdf\\_annotate System Task](#)

- Using the -xlm Option for SDF Retain, Gate Pulse Propagation, and Gate Pulse Detection Warning
- Enhancing SDF Annotation to Support Nets Through SPICE

---

## Using the Unified SDF Feature

The Unified SDF feature allows you to back-annotate SDF delays using the following compilation/elaboration option:

```
-sdf min|typ|max:instance_name:file.sdf
```

For two-step flow:

### Compilation

```
% vcs -sdf min|typ|max:instance_name:file.sdf \
[compile_options]
```

### Simulation

```
% simv [run_options]
```

For three-step flow:

### Analysis

```
% vlogan [vlogan_options] file2.v file3.v
% vhdlan [vhdlan_options] file4.vhd file5.vhd
```

#### Note:

The VHDL bottommost entity first, then move up in order.

### Elaboration

```
% vcs -sdf min|typ|max:instance_name:file.sdf \
[elab_options] top_cfg/entity/module
```

### Simulation

```
% simv [run_options]
```

For more information on specifying delays and SDF files, see [Options for Specifying Delays and SDF Files](#).

For an example, see the `$VCS_HOME/doc/examples/timing/ mx_unified_sdf` directory.

---

## Using the `$sdf_annotate` System Task

You can use the `$sdf_annotate` system task to back-annotate delay values from an SDF file to your Verilog design.

The syntax for the `$sdf_annotate` system task is as follows:

```
$sdf_annotate ("sdf_file" [, module_instance]
 [, "sdf_configfile"] [, "sdf_logfile"] [, "mtm_spec"]
 [, "scale_factors"] [, "scale_type"]);
```

Where:

`"sdf_file"`

Specifies the path to an SDF file.

`module_instance`

Specifies the scope where back-annotation starts. The default is the scope of the module instance that calls `$sdf_annotate`.

`"sdf_configfile"`

Specifies the SDF configuration file. For more information on the SDF configuration file, see the [SDF Configuration File](#) section.

`"sdf_logfile"`

Specifies an SDF log file to which VCS sends error messages and warnings. By default, VCS displays no more than ten warnings and ten error messages about back-annotation and writes no more than that in the log file you specify with the `-l` option. However, if you specify the SDF log file with this argument, the SDF log file receives all messages about back-annotation. You can also use the `+sdfverbose` runtime option to enable the display of all back-annotation messages.

`"mtm_spec"`

Specifies which delay values of `min:typ:max` triplets VCS back-annotates. Its possible values are "MINIMUM", "TYPICAL", "MAXIMUM", or "TOOL\_CONTROL" (default).

`"scale_factors"`

Specifies the multiplier for the minimum, typical, and maximum components of delay triplets. It is a colon separated string of three positive, real numbers "`1.0:1.0:1.0`" by default.

`"scale_type"`

Specifies the delay value from each triplet in the SDF file for use before scaling.  
Its possible values are "FROM\_TYPICAL", "FROM\_MIMINUM", "FROM\_MAXIMUM", and  
"FROM\_MTM" (default).

The usage model to simulate a design using `$sdf_annotation` is same as the basic usage model as shown below:

For two-step flow:

### Compilation

```
% vcs [elab_options] top_cfg/entity/module
```

### Simulation

```
% simv [run_options]
```

For three-step flow:

### Analysis

```
% vlogan [vlogan_options] file2.v file3.v
% vhdlan [vhdlan_options] file4.vhd file5.vhd
```

### Note:

The VHDL bottommost entity first, then move up in order.

### Elaboration

```
% vcs [elab_options] top_cfg/entity/module
```

### Simulation

```
% simv [run_options]
```

For more details, see [Options for Specifying Delays and SDF Files](#).

## Using the `-xIrm` Option for SDF Retain, Gate Pulse Propagation, and Gate Pulse Detection Warning

The following sections explain how to use the new features added under the `-xIrm` option:

- [Using the Optimistic Mode in SDF](#)
- [Using Gate Pulse Propagation](#)
- [Generating Warnings During Gate Pulses](#)

## Using the Optimistic Mode in SDF

Currently, when you use the `-sdfretain` option, SDF retain is visible whenever there is a change in related inputs.

When you specify the `-sdfretain` option with `-xlm alt_retain`, SDF retain is visible only when there is a change in the output. This new behavior is called optimistic mode. For example, consider the following Verilog code:

```
and u(qout,d1,d2);

specify
 (d1 => qout) = (10); //RETAIN (6)
 (d2 => qout) = (10);
endspecify
```

The corresponding SDF entry is:

```
(IOPATH d1 qout (RETAIN (6)) (10))
(IOPATH d2 qout (10))
```

The default output for the above example is:

```
time= 10 , d1=0,d2=0, qout=0
time= 100 , d1=1,d2=0, qout=0
time= 106 , d1=1,d2=0, qout=x // since input d1 change at /
 //100, VCS propagate "x" to qout
time= 110 , d1=1,d2=0, qout=0
= 200 , d1=0,d2=0, qout=0
time= 206 , d1=0,d2=0, qout=x // since input d1 change at
 //200, VCS propagate "x" to qout
time= 210 , d1=0,d2=0, qout=0
time= 300 , d1=0,d2=1, qout=0
time= 400 , d1=1,d2=1, qout=0
time= 406 , d1=1,d2=1, qout=x
time= 410 , d1=1,d2=1, qout=1
```

The output using the `-xlm alt_retain` option (new behavior) is as follows:

```
time= 10 , d1=0,d2=0, qout=0
time= 100 , d1=1,d2=0, qout=0 // since there is no logic
 //change on "qout", no retain "x" seen
time= 200 , d1=0,d2=0, qout=0
time= 300 , d1=0,d2=1, qout=0
time= 400 , d1=1,d2=1, qout=0
time= 406 , d1=1,d2=1, qout=x // since there is logic change
 //on "qout", retain "x" propagated
time= 410 , d1=1,d2=1, qout=1
```

## Using Gate Pulse Propagation

Using the `-x1rm gd_pulseprop` option, VCS always propagates a gate pulse, even when the pulse width is equal to the gate delay. For example, consider the following Verilog code:

```
module dut(qout,dinA,dinB);
output qout;
input dinA;
input dinB;

xor #10 inst(qout,dinA,dinB);

endmodule
```

Under the `-x1rm gd_pulseprop` option, if the pulse width on a gate is equal to the gate delay, VCS always propagates the pulse as shown below:

```
0 qout=x, dinA=1 dinB=1
10 qout=0, dinA=0 dinB=1
20 qout=1, dinA=0 dinB=0
30 qout=0, dinA=0 dinB=1
40 qout=1, dinA=0 dinB=0
50 qout=0, dinA=0 dinB=0
```

## Generating Warnings During Gate Pulses

Using the `-x1rm gd_pulsegarn` option, VCS generates a warning when it detects that the width of a pulse is identical to the gate delay. For example, consider the following Verilog code:

```
module dut(qout,dinA,dinB);
output qout;
input dinA;
input dinB;

xor #10 inst(qout,dinA,dinB);

endmodule
```

Under the `-x1rm gd_pulsegarn` option, if the pulse width on a gate is equal to the gate delay, VCS generates the following warning message:

```
0 qout=x, dinA=1 dinB=1

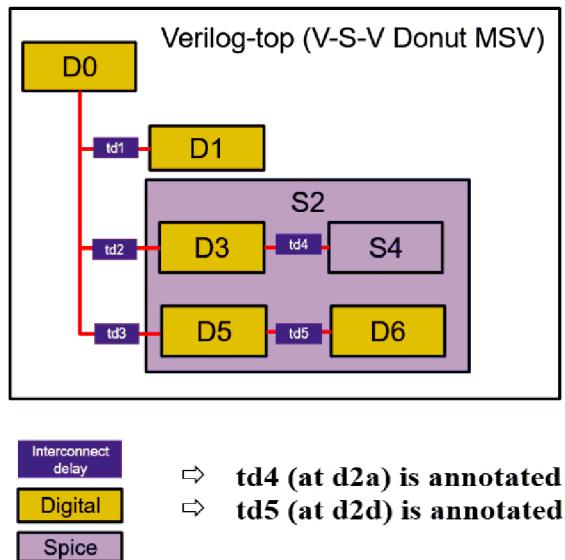
Warning-[PWIWGD] Pulse Width Identical With Gate Delay
verilogfile.v, 42
top.mid_inst.dut_inst
At time 10, pulse width identical with gate delay "10" is detected

10 qout=0, dinA=0 dinB=1
20 qout=1, dinA=0 dinB=0
```

## Enhancing SDF Annotation to Support Nets Through SPICE

VCS supports SDF in Mixed Signal Verification (MSV) that covers the INTERCONNECT delay between digital port and analog port. VCS also annotates INTERCONNECT delay when SPICE region is parent (such as a V-S-V donut configuration). Therefore, you can use the compile time option `+msvsdfext`, to enable SDF annotation with MSV to include digital nets passing through SPICE.

Figure 137 V-S-V Donut Configuration



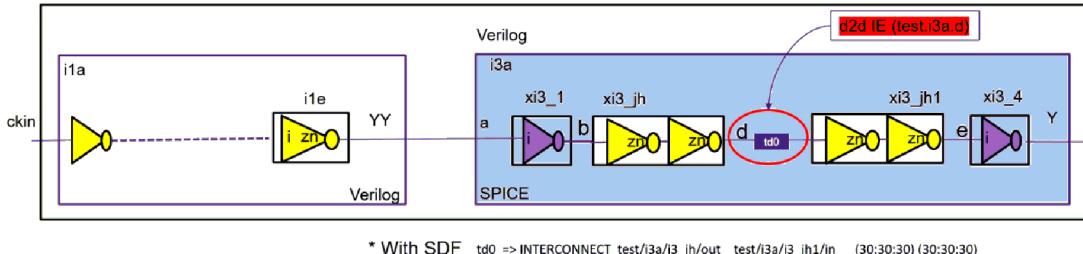
## Use Model

VCS annotates SDF delays in the input, inout and output ports using the `+msvsdfext` option for the following scenarios:

- d2d under V-S-V donut configuration

Figure 138 shows a d2d connection where `td0` is the interconnect delay. SDF is annotated in input, output scenarios.

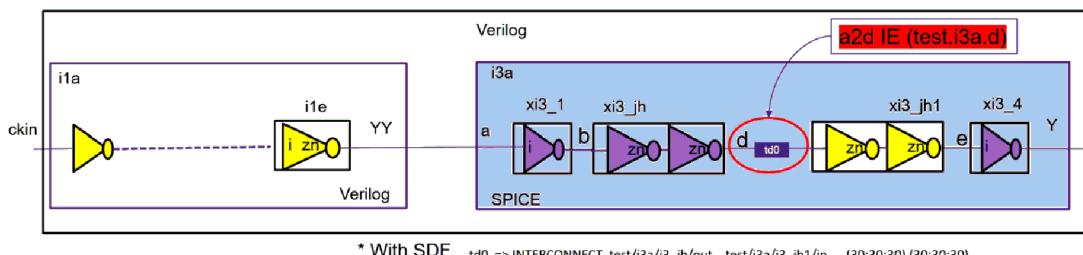
Figure 138 d2d under V-S-V Donut Configuration



- a2d under V-S-V donut configuration

Figure 139 shows a d2d connection where td0 is the interconnect delay. SDF is annotated in input, output scenarios.

Figure 139 a2d under V-S-V Donut Configuration



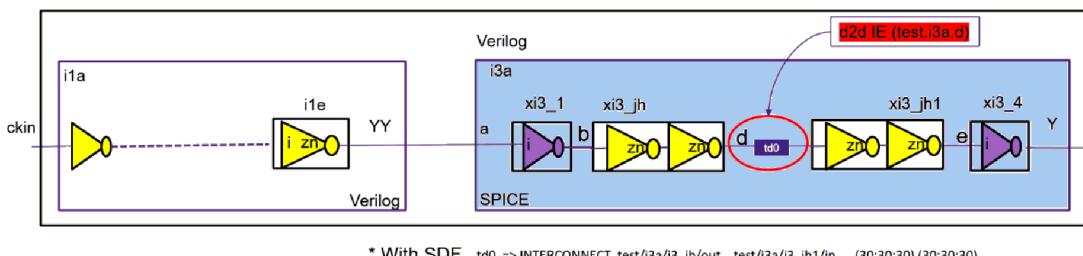
## Use Model

VCS annotates SDF delays in the input, inout and output ports using the `+msvsdfext` option for the following scenarios:

- d2d under V-S-V donut configuration

Figure 138 shows a d2d connection where td0 is the interconnect delay. SDF is annotated in input, output scenarios.

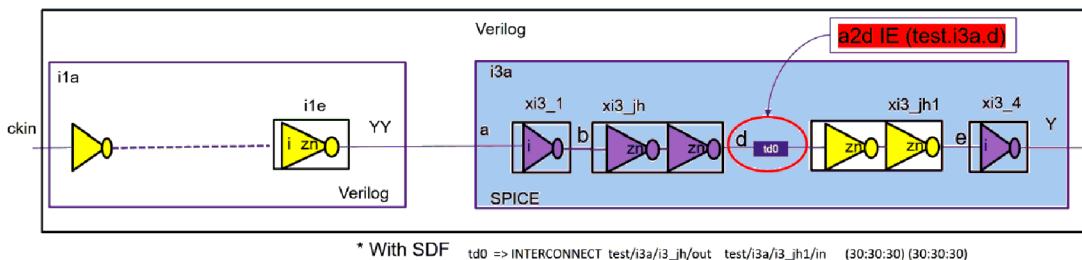
Figure 140 d2d under V-S-V Donut Configuration



- a2d under V-S-V donut configuration

Figure 139 shows a d2d connection where td0 is the interconnect delay. SDF is annotated in input, output scenarios.

Figure 141 a2d under V-S-V Donut Configuration



## Limitations

The following are the limitations of the current implementation of SDF annotation through SPICE:

- Supports only INTERCONNECT delay.
- VCS identifies the d2a and inout scenario under V-S-V donut configuration and generates a warning message.
- Shows limitation of multiple drivers that are running at the same time. This is a design issue and the designer must resolve this issue.
- Does not support if the destination signal is connected to tran-gate.
- Behavior of other delay types such as “PORT” (`PORT test/i2a/a (10)`) is not determined.
- Does not support VHDL SPICE boundary in IC delays.
- Does not support real type and user defined type in SPICE boundary with IC delays.

## Precompiling an SDF File

Whenever you compile your design, if your design back-annotates SDF data, VCS parses either the ASCII-text SDF file or the precompiled version of the ASCII-text SDF file that VCS can make from the original ASCII-text SDF file. VCS does this even if the SDF file is unchanged and already compiled into a binary version by a previous compilation. In addition, VCS parses even when you are using incremental compilation and the parts of the design back-annotated by the SDF file are unchanged.

VCS can parse the precompiled SDF file much faster than it can parse the ASCII-text SDF file. Therefore, for large SDF files, it is good to have VCS create a precompiled version of the SDF file.

## **Creating the Precompiled Version of the SDF File**

To create the precompiled version of the SDF file, include the `+csdf+precompile` option on the `vcs` command line.

By default, the `+csdf+precompile` option creates the precompiled SDF file in the same directory as the ASCII-text SDF file and differentiates the precompiled version by appending `_c` to its extension. For example, if the `/u/design/sdf` directory contains a `design1.sdf` file, the `+csdf+precompile` option creates the precompiled version of the file named `design1.sdf_c` in the `/u/design/sdf` directory.

After you have created the precompiled version of the SDF file, you no longer need to include the `+csdf+precompile` option on the `vcs` command line, unless there is a change in the SDF file. Continuing to include it, however, such as in a script that you run every time you compile your design, has no effect when the precompiled version is newer than the ASCII-text SDF file. However, it creates a new precompiled version of the SDF file whenever the ASCII-text SDF file changes. Therefore, this option is intended to be used in scripts for compiling your design.

When you recompile your design, VCS finds the precompiled SDF file in the same directory as the SDF file specified in the `$sdf_annotate` system task. You can also specify the precompiled SDF file in the `$sdf_annotate` system task. The `+csdf+precompile` option also supports zipped SDFs.

## **Precompiling SDF Without Compiling Design Files**

You can also precompile the SDF files without compiling the entire set of design files. For this, use the following command option:

```
+csdf+precomp+file+<sdf file>
```

The following is the use model for this option:

```
vcs +csdf+precomp+file+<sdf file>
```

For example:

```
vcs +csdf+precomp+file+test1.sdf
```

## **Writing Precompiled SDF to a Different Directory**

You can write the precompiled SDF to a different directory. To do this, use the following command option:

```
+csdf+precomp+dir+PRE_COMP_SDF/
```

The following is the use model for this option:

```
vcs +csdf+precomp+file+<sdf file> +csdf+precomp+dir+<DIR>
```

For example:

```
mkdir PRE_COMP_SDF vcs +csdf+precomp+file+./test.sdf +csdf+precomp+dir+PRE_COMP_SDF/
```

**Note:**

The precompiled SDF file is generated in the `PRE_COMP_SDF` directory.

## SDF Configuration File

You can use the configuration file to control the following on a module-type basis as well as on a global basis:

- The `min:typ:max` selection
- Scaling
- The Module-Input-Port-Delay (MIPD) approximation policy for cases of ‘overlapping’ annotations to the same input port

Additionally, there is a mapping command you can use to redirect the target of `IOPATH` and `TIMINGCHECK` statements from the scope of `INSTANCE` to a specific `IOPATH` or `TIMINGCHECK` in its sub hierarchy for all instances of a specified module type.

## Delay Objects and Constructs

The mapping from SDF statements to simulation objects in VCS is fixed, as shown in [Table 32](#).

*Table 32      VCS Simulation Delay Objects/Constructs*

| SDF Constructs  | VCS Simulation Object                |
|-----------------|--------------------------------------|
| Delays          |                                      |
| PATHPULSE       | module path pulse delay              |
| GLOBALPATHPULSE | module path pulse reject/error delay |
| IOPATH          | module path delay                    |
| PORT            | module input port delay              |

Table 32 VCS Simulation Delay Objects/Constructs (Continued)

| SDF Constructs | VCS Simulation Object                                                                                        |
|----------------|--------------------------------------------------------------------------------------------------------------|
| INTERCONNECT   | module input port delay or, intermodule path delay<br>when <code>+multisource_int_delays</code> is specified |
| NETDELAY       | module input port delay                                                                                      |
| DEVICE         | primitive and module path delay                                                                              |
| Timing-Checks  |                                                                                                              |
| SETUP          | <code>\$setup timing-check limit</code>                                                                      |
| HOLD           | <code>\$hold timing-check limit</code>                                                                       |
| SETUPHOLD      | <code>\$setup and \$hold timing-check limit</code>                                                           |
| RECOVERY       | <code>\$recovery timing-check limit</code>                                                                   |
| SKEW           | <code>\$skew timing-check limit</code>                                                                       |
| WIDTH          | <code>\$width timing-check limit</code>                                                                      |
| PERIOD         | <code>\$period timing-check limit</code>                                                                     |
| NOCHANGE       | ignored                                                                                                      |
| PATHCONSTRAINT | ignored                                                                                                      |
| SUM            | ignored                                                                                                      |
| DIFF           | ignored                                                                                                      |
| SKEWCONSTRAINT | ignored                                                                                                      |

## SDF Configuration File Commands

This section explains the following commands used in SDF configuration files with their syntax and examples:

- [The INTERCONNECT\\_MIPD Command](#)
- [The MTM Command](#)
- [The SCALE Commands](#)

## The INTERCONNECT\_MIPD Command

The `INTERCONNECT_MIPD` command selects INTERCONNECT delays in the SDF file that are mapped to Module Input Port Delays (MIPDs) in VCS. By default, interconnect delay is modeled as MIPD. It means that even if a destination has multiple sources, only one port delay is selected.

`MINIMUM` Selects the shortest delay from all INTERCONNECT delay value entries in the SDF file to MIPD for the input or the inout port instance. The delay specifies the connection to the input or inout port.

`MAXIMUM` Selects the longest delay from all INTERCONNECT delay value entries in the SDF file to MIPD for the input or the inout port instance. The delay specifies the connection to the input or inout port. By default, VCS annotates the `MAXIMUM` delay.

`AVERAGE` Selects the average delay of all INTERCONNECT delay value entries in the SDF file to MIPD for the input or the inout port instance. The delay specifies the connection to the input or inout port.

`LAST` Selects the delay in the last INTERCONNECT delay value entries in the SDF file to MIPD for the input or the inout port instance. The delay specifies the connection to the input or inout port.

The default value of the `INTERCONNECT_MIPD` command is `MAXIMUM` and its syntax is as follows:

```
INTERCONNECT_MIPD = MINIMUM | MAXIMUM | AVERAGE | LAST;
```

For example:

```
INTERCONNECT_MIPD=LAST;
```

## The MTM Command

The command annotates the minimum, typical, or maximum delay value. You can specify one of the following keywords:

`MINIMUM` Annotates the minimum delay value.

`TYPICAL` Annotates the typical delay value.

`MAXIMUM` Annotates the maximum delay value.

`TOOL_CONTROL` Delay value is determined by the command-line options of the Verilog tool (`+mindelays`, `+typdelays`, or `+maxdelays`).

The default for the `MTM` command is `TOOL_CONTROL` and its syntax is as follows:

```
MTM = MINIMUM | TYPICAL | MAXIMUM | TOOL_CONTROL;
```

For example:

```
MTM=MAXIMUM;
```

## The SCALE Commands

There are the following two types of SCALE commands:

- **SCALE\_FACTORS** - Set of three real number multipliers that scale the timing information in the SDF file to the minimum, typical, and maximum timing information that is back-annotated to the Verilog tool. Each multiplier represents a positive real number, for example 1.6:1.4:1.2.
- **SCALE\_TYPE** - Selects one of the following keywords to scale the timing specification in the SDF file to the minimum, typical, and maximum timing that is back-annotated to the Verilog tool.

**FROM\_MINIMUM** Scales from the minimum timing specification in the SDF file.

**FROM\_TYPICAL** Scales from the typical timing specification in the SDF file.

**FROM\_MAXIMUM** Scales from the maximum timing specification in the SDF file.

**FROM\_MTM** Scales directly from the minimum, typical, and maximum timing specifications in the SDF file.

The syntax of **SCALE\_FACTORS** and **SCALE\_TYPE** is as follows:

```
SCALE_FACTORS = number : number : number;
SCALE_TYPE = FROM_MINIMUM | FROM_TYPICAL | FROM_MAXIMUM | FROM_MTM;
```

For example:

```
SCALE_FACTORS=100:0:9;
SCALE_TYPE=FROM_MTM;
SCALE_FACTORS=1.1:2.1:3.1;
SCALE_TYPE=FROM_MINIMUM;
```

## An SDF Example With Configuration File

The following example uses the VCS SDF configuration file, `sdf.cfg`:

The content of the `test.v` file is as follows:

```
// test.v - test sdf annotation
`timescale 1ns/1ps
module test;
initial begin
 $sdf_annotation("./test.sdf", test, "./sdf.cfg", , , ,);
end
```

```

wire out1,out2;
wire w1,w2;
reg in;
reg ctrl,ctrlw;
sub Y (w1,w2,in,in,ctrl,ctrl);
sub W (out1,out2,w1,w2,ctrlw,ctrlw);
initial begin
 $display(" i c ww oo");
 $display("ttt n t 12 12");
 $monitor($realtime,,,in,,ctrl,,w1,w2,,out1,out2);
end
initial begin
 ctrl = 0;// enable
 ctrlw = 0;
 in = 1'bx; //stabilize at x;
 #100 in = 1; // x-1
 #100 ctrl = 1; // 1-z
 #100 ctrl = 0; // z-1
 #100 in = 0; // 1-0
 #100 ctrl = 1; // 0-z
 #100 ctrl = 0; // z-0
 #100 in = 1'bx; // 0-x
 #100 ctrl = 1; // x-z
 #100 ctrl = 0; // z-x
 #100 in = 0; // x-0
 #100 in = 1; // 0-1
 #100 in = 1'bx; // 1-x
end
endmodule
`celldefine
module sub(o1,o2,i1,i2,c1,c2);
output o1,o2;
input i1,i2;
input c1,c2;
bufif0 Z(o1,i1,c1);
bufif0 (o2,i2,c2);
specify
 (i1,c1 *> o1) = (1,2,3,4,5,6);
 // 01 = 1, 10 = 2, 0z = 3, z1 = 4, 1z = 5, z0 = 6
 if (i2==1'b1) (i2,c2 *> o2) = (7,8,9,10,11,12);
 // 01 = 7, 10 = 8, z1 = 10, 1z = 11, z0 = 12
endspecify
subsub X ();
endmodule
`endcelldefine
module subsub(oa,ob,ib,ia);
input ia,ib;output oa,ob;
specify
 (ia *> oa) = 99.99;
 (ib *> ob) = 2.99;
endspecify
endmodule

```

**The content of the test.sdf file is as follows:**

```
(DELAYFILE
(SDFVERSION "3.0")
(DESIGN "sdftest")
(DATE "July 14, 1997")
(VENDOR "Synopsys")
(PROGRAM "manual")
(VERSION "4.0")
(DIVIDER .)
(VOLTAGE)
(PROCESS "")
(TEMPERATURE)
(TIMESCALE 1 ns)
(CELL (CELLTYPE "sub")
(INSTANCE *)
(DELAY (ABSOLUTE
(IOPATH i1 o1
(10:11:12) (13:14:15) (16:17:18) (19:20:21) (22:23:24) (25:26:27))
(COND (i2==1) (IOPATH i2 o2
(10:11:12) (13:14:15) (16:17:18) (19:20:21) (22:23:24) (25:26:27)))
)))
)
```

**The content of the sdf.cfg file is as follows:**

```
PATHPULSE=IGNORE;
INTERCONNECT_MIPD=MAXIMUM;
MTM=TOOL_CONTROL;
SCALE_FACTORS=100:0:9;
SCALE_TYPE=FROM_MTM;
MTM = TYPICAL;
SCALE_TYPE=FROM_MINIMUM;
SCALE_FACTORS=1.1:2.1:3.1;
```

Run the file using the following command:

- % vcs test.v
- % simv

**The following output is generated:**

```
i c ww oo
ttt n t 12 12
0 x 0 xx xx
100 1 0 xx xx
139.9 1 0 11 xx
.....
1100 1 0 00 x0
1105 1 0 00 00
```

```
1121 1 0 11 00
1142 1 0 11 11
```

## Delays and Timing

This section describes the following topics:

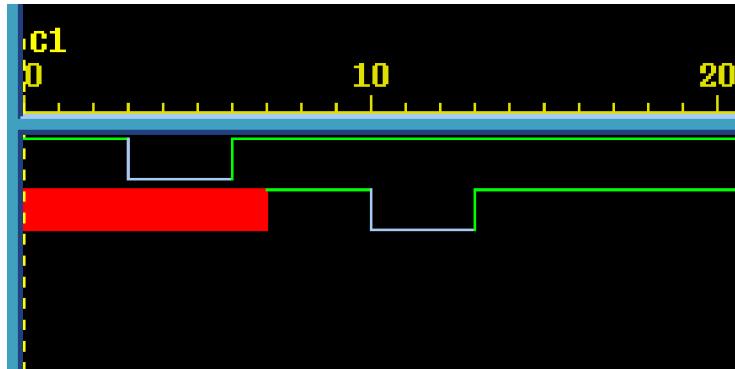
- [Transport and Inertial Delays](#)
- [Pulse Control](#)
- [Specifying the Delay Mode](#)
- [Support for Delayed Annotation When No Conditions Have Matched](#)
- [Support for Delay Selection Options for Recent Active Paths by Ignoring Condition Checking](#)
- [Specifying Timing Control Attributes in the +optconfigfile File](#)

## Transport and Inertial Delays

Delays can be categorized into transport and inertial delays.

Transport delays allow all pulses that are narrower than the delay to propagate. For example, [Figure 142](#) shows the waveforms for an input and output port of a module that models a buffer with a module path delay of seven-time units between these ports. The waveform on top is that of the input port and the waveform underneath is that of the output port. In this example, you have enabled transport delays for module path delays and specified that a pulse three-time units wide can propagate. For an explanation on how this is done, see [Enabling Transport Delays](#) and [Pulse Control](#).

*Figure 142 Transport Delay Waveforms*

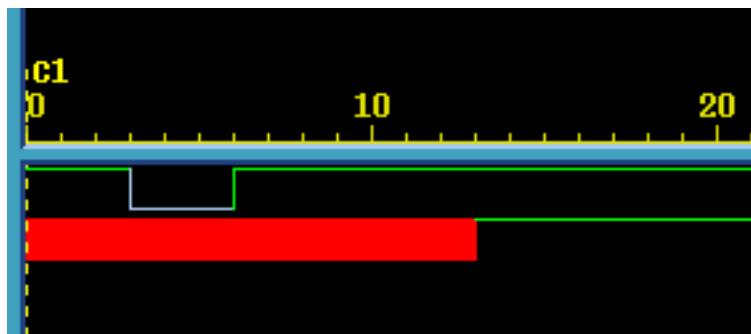


At time 0, a pulse three-time unit wide begins on the input port. This pulse is narrower than the module path delay of seven-time units, but this pulse propagates through the module and appears on the output port after seven-time units. Similarly, another narrow pulse begins on the input port at time 3 and it also appears on the output port seven-time units later.

You can apply transport delays on all module path delays and all SDF INTERCONNECT delays back-annotated to a net from an SDF file. For more information on SDF back-annotation, see [SDF Annotation](#).

Inertial delays, in contrast, filter out all pulses that are narrower than the delay. [Figure 143](#) shows the waveforms for the same input and output ports when you have not enabled transport delays for module path delays.

*Figure 143 Inertial Delay Waveforms*



The pulse that begins at time 0 that is three-time units wide does not propagate to the output port because it is narrower than the seven-time unit module path delay. Also, the pulse that begins at time 3 does not propagate. Note that the wide pulse that begins at time 6 does propagate to the output port.

Gates, switches, MIPDs, and continuous assignments only have inertial delays, which are the default type of delay for module path delays and INTERCONNECT delays back-annotated from an SDF file to a net.

## The Inertial Delay Implementation

The inertial delay implementation is the same for primitives [gates, switches, and User-Defined Primitives (UDP)], continuous assignments, MIPDs, module path delays, and INTERCONNECT delays back-annotated from an SDF file to a net. For more details on SDF back-annotation, see [SDF Annotation](#). There is also a third implementation that is for module path and INTERCONNECT delays and pulse control, see [Pulse Control](#).

The implementation of inertial delays is as follows:

Consider an event that is scheduled by the leading edge of a pulse and is either scheduled for a later simulation time or has not yet occurred. This event is replaced by the event that

is scheduled by the trailing edge at the end of the specified delay and at a new simulation time. All narrow pulses are filtered out.

**Note:**

VCS enables more complex and flexible pulse control processing when you include the `+pulse_e/number` and `+pulse_r/number` options. For details on these options, see [Pulse Control](#).

## Enabling Transport Delays

Transport delays are not the default delays. You can specify transport delays on module-path delays with the `+transport_path_delays` compile-time option. For this option to work, you must also include the `+pulse_e/number` and `+pulse_r/number` compile-time options. For details on these options, see [Pulse Control](#).

You can specify transport delays on a net to which you back-annotate SDF INTERCONNECT delays with the `+transport_int_delays` compile-time option. For this option to work, you must also include the `+pulse_int_e/number` and `+pulse_int_r/number` compile-time options. For details on these options, see [Pulse Control](#).

The `+pulse_e/number`, `+pulse_r/number`, `+pulse_int_e/number`, and `+pulse_int_r/number` options define specific thresholds for pulse width, which allow you to tell VCS to filter out only some of the pulses and let the other pulses propagate. For details on these options, see [Pulse Control](#).

## Pulse Control

As discussed in previous sections, for pulses narrower than a module path or INTERCONNECT delay, you have two options. One is to filter all pulses by using the default inertial delay. Another is to allow all pulses to propagate by specifying transport delays. VCS also provides a third option - pulse control. Pulse control allows you to do the following:

- Allow pulses that are slightly narrower than the delay to propagate.
- Have VCS replace even narrower pulses with an x value pulse on the output and display a warning message.
- Have VCS then filter out and ignore pulses that are even narrower than the ones for which it propagates an x value pulse and displays an error message.

For module path delays, specify pulse control with the `+pulse_e/number` and `+pulse_r/number` compile-time options. For INTERCONNECT delays, specify pulse control with the `+pulse_int_e/number` and `+pulse_int_r/number` compile-time options.

The `number` argument of the `+pulse_e/number` option specifies a percentage of the module path delay. VCS replaces pulses whose widths that are narrower than the

specified percentage of the delay with an `x` value pulse on the output or inout port and displays a warning message.

Similarly, the `number` argument of the `+pulse_int_e/number` option specifies a percentage of the INTERCONNECT delay. VCS replaces pulses whose widths are narrower than the specified percentage of the delay with an `x` value pulse on the inout or output port instance that is the load of the net to which you back-annotated the INTERCONNECT delay. It also displays a warning message.

The `number` argument of the `+pulse_r/number` option also specifies a percentage of the module path delay. VCS filters out the pulses whose widths are narrower than the specified percentage of the delay. With these pulses, there is no warning message and VCS ignores these pulses.

Similarly, the `number` argument of the `+pulse_int_r/number` option specifies a percentage of the INTERCONNECT delay. VCS filters out pulses whose widths are narrower than the specified percentage of the delay. There is no warning message with these pulses.

You can use pulse control with transport delays (see [Pulse Control With Transport Delays](#)) or inertial delays (see [Pulse Control With Inertial Delays](#)).

When a pulse is narrow enough for VCS to display a warning message and propagate an `x` value pulse, you can set VCS to do one of the following:

- Place the starting edge of the `x` value pulse on the output, as soon as it detects that the pulse is sufficiently narrow, by including the `+pulse_on_detect` compile-time option.
- Place the starting edge on the output at the time when the rising or falling edge of the narrow pulse would propagate to the output. This is the default behavior.

For more details, see [Specifying Pulse on Event or Detect Behavior](#).

Also, when a pulse is sufficiently narrow to display a warning message and propagate an `x` value pulse, you can have VCS propagate the `x` value pulse. However, you can disable the display of the warning message with the `+no_pulse_msg` runtime option.

## Pulse Control With Transport Delays

You can specify transport delays for module path delays with the `+transport_path_delays`, `+pulse_e/number`, and `+pulse_r/number` options. You must include all three of these options.

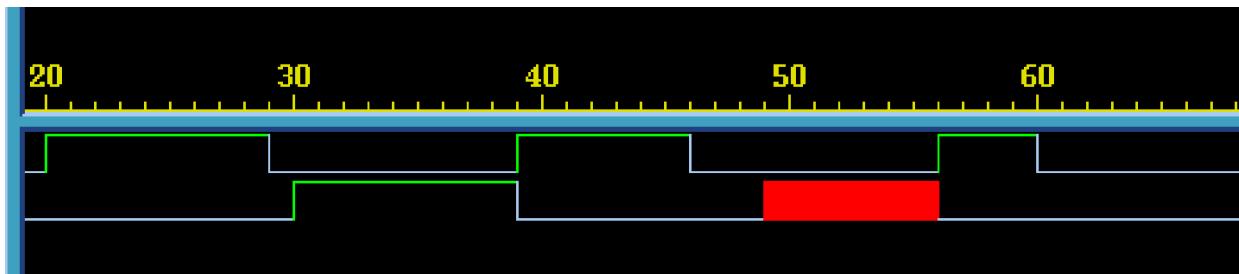
You can specify transport delays for INTERCONNECT delays on nets with the `+transport_int_delays`, `+pulse_int_e/number`, and `+pulse_int_r/number` options. You must include all three of these options.

If you want VCS to propagate all pulses, no matter how narrow, specify a 0 percentage. For example, if you want VCS to replace pulses that are narrower than 80% of the delay with an x value pulse (and display a warning message) and filter out pulses that are narrower than 50% of the delay, enter the `+pulse_e/80` and `+pulse_r/50` or `+pulse_int_e/80` and `+pulse_int_r/50` compile-time options.

[Figure 144](#) shows the waveforms for input and output ports for an instance of a module that models a buffer with a ten-time unit module path delay. For this, the `vcs` command contains the following compile-time options:

```
+transport_path_delays +pulse_e/80 +pulse_r/50
```

*Figure 144 Pulse Control With Transport Delays*



In the example illustrated in [Figure 144](#), the following occurs:

1. At time 20, the input port toggles to 1.
2. At time 29, the input port toggles to 0 ending a nine-time unit wide value 1 pulse on the input port.
3. At time 30, the output port toggles to 1. The nine-time unit wide value 1 pulse that began at time 20 on the input port is propagating to the output port. This is because transport delays are enabled and the nine-time unit is more than 80% of the ten-time unit module path delay.
4. At time 39, the input port toggles to 1 ending a ten-time unit wide value 0 pulse. Also, at time 39 the output port toggles to 0. The ten-time unit wide value 0 pulse that began at time 29 on the input port is propagating to the output port.
5. At time 46, the input port toggles to 0 ending a seven-time unit wide value 1 pulse.
6. At time 49, the output port transitions to x. The seven-time unit wide value 1 pulse that began at time 39 on the input port has propagated to the output port. However, VCS has replaced it with an x value pulse because seven-time units is less than 80% of the module path delay. VCS issues a warning message in this case.

7. At time 56, the input port toggles to 1 ending a ten-time unit wide value 0 pulse. Also, at time 56, the output port toggles to 0. The ten-time unit wide value 0 pulse that began at time 46 on the input port is propagating to the output port.
8. At time 60, the input port toggles to 0 ending a four-time unit wide value 1 pulse. Four-time units is less than 50% of the module path delay. Therefore, VCS filters out this pulse and no indication of it appears on the output port.

## Pulse Control With Inertial Delays

You can enter the `+pulse_e/number` and `+pulse_r/number` or `+pulse_int_e/number` and `+pulse_int_r/number` options without the `+transport_path_delays` or `+transport_int_delays` options. If you do this, you are specifying pulse control for inertial delays on module path delays and INTERCONNECT delays.

There is a special implementation of inertial delays with pulse control for module path delays and INTERCONNECT delays. In this implementation, value changes on the input can schedule two events on the output.

The first of these two scheduled events always causes a change on the output. The type of value changes on the output is determined by the following:

- The first event is scheduled by the leading edge of a pulse whose width is equal to or wider than the percentage specified by the `+pulse_e/number` option. Then, the value change on the input propagates to the output.
- The pulse is not wider than the percentage specified by the `+pulse_e/number` option, but is wider than the percentage specified by the `+pulse_r/number` option. Then, the value change is replaced by an x value.
- The pulse is not wider than the percentage specified by the `+pulse_r/number` option and the pulse is filtered out.

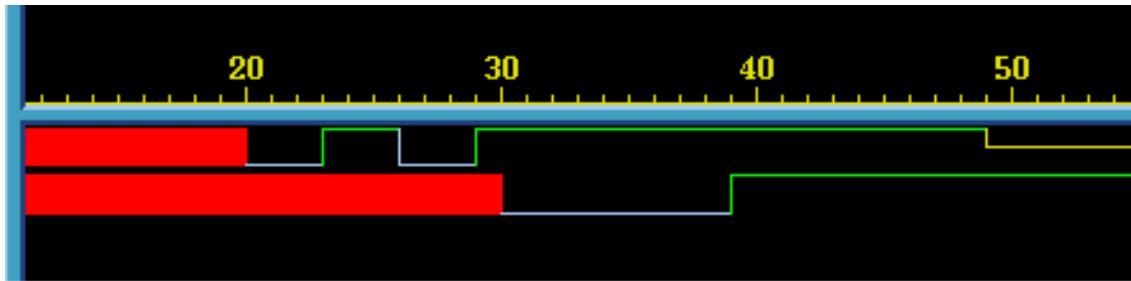
The second scheduled event is always tentative. If another event occurs on the input before the first event occurs on the output, that additional event on the input cancels the second scheduled event and schedules a new second event.

[Figure 145](#) shows the waveforms for input and output ports for an instance of a module that models a buffer with a ten-time unit module path delay. The `vcs` command contains the following compile-time options:

```
+pulse_e/0 +pulse_r/0
```

In this example, specifying 0 percentage means that the trailing edge of all pulses can change the second scheduled event on the output. Specifying 0 does not mean that all pulses propagate to the output because this implementation has its own way of filtering out short pulses.

**Figure 145 Pulse Control With Inertial Delays**



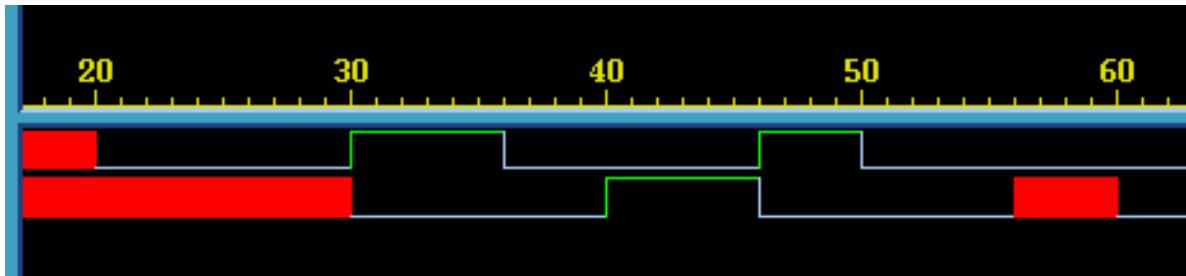
In the example illustrated in [Figure 145](#), the following occurs:

1. At time 20, the input port transitions to 0. This schedules a transition to 0 on the output port at time 30, ten-time units later as specified by the module path delay. This is the first scheduled event on the output port. This event is not tentative and it occurs.
2. At time 23, the input port toggles to 1. This schedules a transition to 1 on the output port at time 33. This is the second scheduled event on the output port. This event is tentative.
3. At time 26, the input port toggles to 0. This cancels the current scheduled second event and replaces it by scheduling a transition to 0 at time 36. The first scheduled event is a transition to 0 at time 30, so the new second scheduled event is not really a transition on the output port. This is how this implementation filters out narrow pulses.
4. At time 29, the input port toggles to 1. This cancels the current scheduled second event and replaces it by scheduling a transition to 1 at time 39.
5. At time 30, the output port transitions to 0. The second scheduled event on the output becomes the first scheduled event and is therefore, no longer tentative.
6. At time 39, the output port toggles to 1.

[Figure 146](#) shows the waveforms for input and output ports for an instance of the same module with a ten-time unit module path delay. The `vcs` command contains the following compile-time options:

```
+pulse_e/60 +pulse_r/40
```

Figure 146 Pulse Control With Inertial Delays and Narrow Pulses



In the example illustrated in [Figure 146](#), the following occurs:

1. At simulation time 20, the input port transitions to 0. This schedules the first event on the output port, a transition to 0 at time 30.
2. At simulation time 30, the input port toggles to 1. This schedules the output port to toggle to 1 at time 40. Also, at simulation time 30, the output port transitions to 0. It does not matter which of these events happened first. At the end of this time, there is only one scheduled event on the output.
3. At simulation time 36, the input port toggles to 0. This is the trailing edge of a six-time unit wide value 1 pulse. The pulse is equal to the width specified with the `+pulse_e/60` option so VCS schedules a second event on the output, a value change to 0 on the output at time 46.
4. At simulation time 40, the output toggles to 1 so now there is only one event scheduled on the output, the value change to 0 at time 46.
5. At simulation time 46, the input toggles to 1 scheduling a transition to 1 at time 56 on the output. Also at time 46, the output toggles to 0. There is now only one event scheduled on the output.
6. At time 50, input port toggles to 0. This is the trailing edge of a four time unit wide value 1 pulse. The pulse is not equal to the width specified with the `+pulse_e/60` option. However, it is equal to the width specified with the `+pulse_r/40` option, therefore, VCS changes the first scheduled event from a change to 1 to a change to x at time 56 and schedules a second event on the output, a transition to 0 at time 60.
7. At time 56, the output transitions to x and VCS issues a warning message.
8. At time 60, the output transitions to 0.

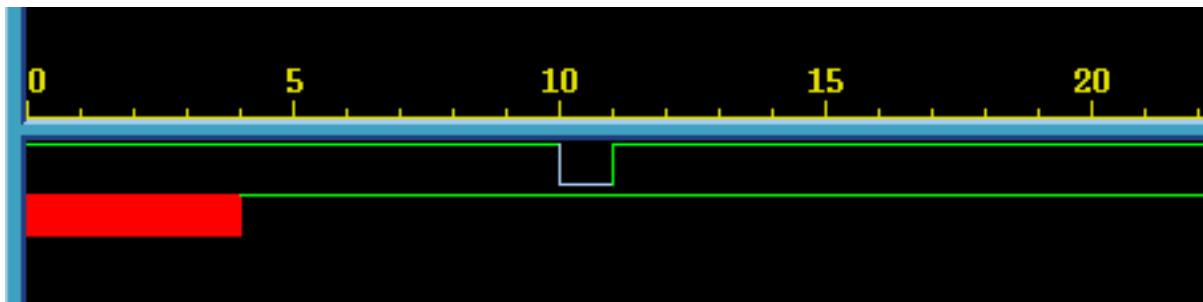
Pulse control sometimes blurs the distinction between inertial and transport delays. In this example, the results are the same if you also included the `+transport_path_delays` option.

## Specifying Pulse on Event or Detect Behavior

Asymmetric delays, such as different rise and fall times for a module path delay, can cause schedule cancellation problems for pulses. These problems persist when you specify transport delays and can persist for a wide range of percentages that you specify for pulse control options.

For example, for a module that models a buffer, if you specify a rise time of 4 and a fall time of 6 for a module path delay, a narrow value 0 pulse can cause scheduling problems, as illustrated in [Figure 147](#).

*Figure 147 Asymmetric Delays and Scheduling Problems*



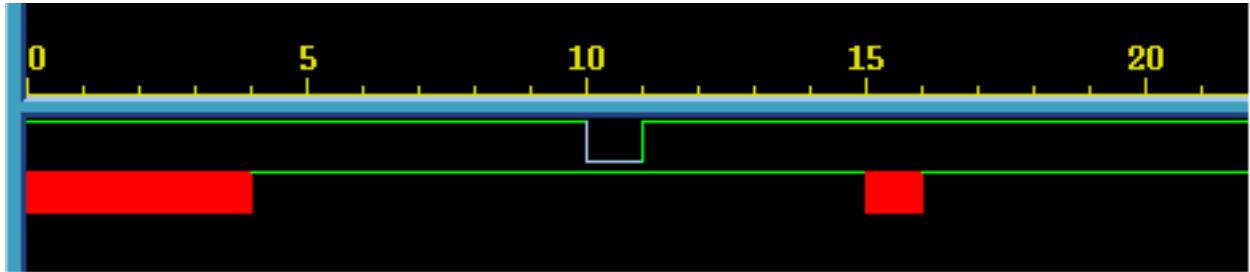
In this example, you include the `+pulse_e/100` and `+pulse_r/0` options. The scheduling problem is that the leading edge of the pulse on the input, at time 10, schedules a transition to 0 on the output at time 16; but the trailing edge, at time 11, schedules a transition to 1 on the output at time 15.

Obviously, the output has to end up with a value of 1 so VCS cannot allow the events scheduled at time 15 and 16 to occur in sequence; if it did, the output ends up with a value of 0. This problem persists when you enable transport delays and whenever the percentage specified in the `+pulse_r/number` option is low enough to enable the pulse to propagate through the module.

To circumvent this problem, when a later event on the input schedules an event on the output that is earlier than the event scheduled by the previous event on the input, VCS cancels both events on the output.

This ensures that the output ends up with the proper value, but what it does not do is indicate that something happened on the output between times 15 and 16. You might want to see an error message and an X value pulse on the output indicating there was an undefined event on the output between these simulation times. You see this message and the X value pulse, if you include the `+pulse_on_event` compile-time option, specifying pulse on event behavior, as illustrated in [Figure 148](#). Pulse on event behavior calls for an X value pulse on the output after the delay and when there are asymmetrical delays scheduling events on the output that would be canceled by VCS, to output an X value pulse between those events instead.

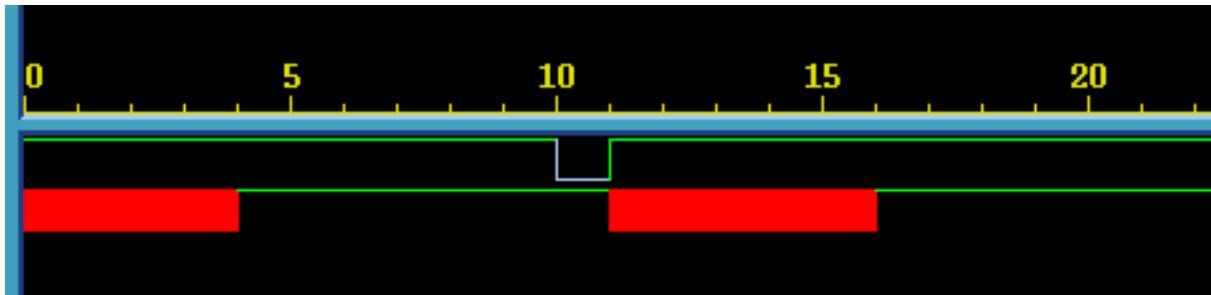
*Figure 148 Using +pulse\_on\_event*



In most cases where the `+pulse_e/number` and `+pulse_r/number` options already create X value pulses on the output, also including the `+pulse_on_event` option to specify pulse on event behavior makes no change on the output.

Pulse on detect behavior, specified by the `+pulse_on_detect` compile-time option, displays the leading edge of the X value pulse on the output. This is done as soon as events on the input, controlled by the `+pulse_e/number` and `+pulse_r/number` options, schedule an X value pulse to appear on the output. Pulse on detect behavior differs from pulse on event behavior in that it calls for the X value pulse to begin before the delay elapses. [Figure 149](#) illustrates pulse on detect behavior.

*Figure 149 Using +pulse\_on\_detect*



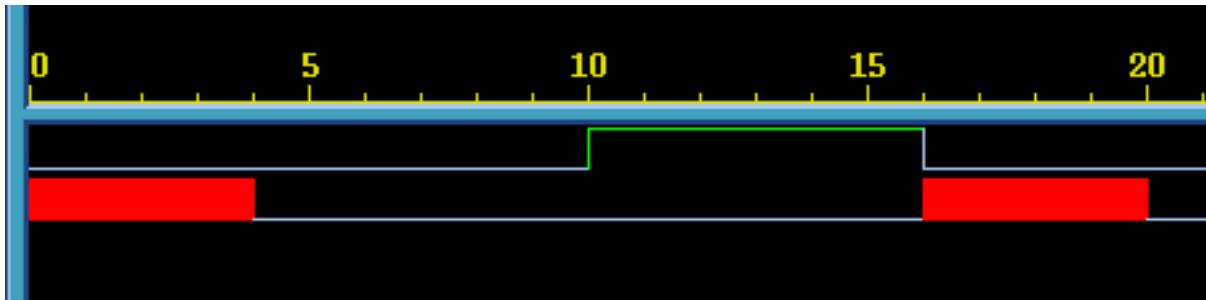
In this example, by including the `+pulse_on_detect` option, VCS causes the leading edge of the X value pulse on the output to begin at time 11. This is because of an unusual event that occurred on the output between times 15 and 16 because of the rise at simulation time 11.

Using pulse on detect behavior can also show you when VCS has scheduled multiple events for the same simulation time on the output. This is done by starting the leading edge of an X value pulse on the output as soon as VCS has scheduled the second event.

For example, a module that models a buffer has a rise time module path delay of 10 time units and a fall time module path delay of 4 time units.

[Figure 150](#) shows the waveforms for the input and output port when you include the `+pulse_on_detect` option.

*Figure 150 Pulse on Detect Behavior Showing Multiple Transitions*



In the example illustrated in [Figure 150](#), the following occurs:

1. At simulation time 0, the input port transitions to 0 scheduling the first event on the output, a transition to 0 at time 4.
2. At time 4, the output transitions to 0.
3. At time 10, the input transitions to 1 scheduling a transition to 1 on the output at time 20.
4. At time 16, the input toggles to 0 scheduling a second event on the output at time 20, a transition to 0. This event also is the trailing edge of a six-time unit wide value 1 pulse so the first event changes to a transition to X. There is more than one event for different value changes on the output at time 20, so VCS begins the leading edge of the X value pulse on the output at this time.
5. At time 20, the output toggles to 0, the second scheduled event at this time.

If you did not include the `+pulse_on_detect` option, or substituted the `+pulse_on_event` option, you would not see the X value pulse on the output between times 16 and 20.

Pulse on detect behavior does not show you when asymmetrical delays schedule multiple events on the output. Other kinds of events can cause multiple events on the output at the same simulation time, such as different transition times on two input ports and different module path delays from these input ports to the output port. Pulse on detect behavior shows you an X value pulse on the output starting when the second event was scheduled on the output port.

## Specifying the Delay Mode

It is possible for a module definition to include module path delay that does not equal the cumulative delay specifications in primitive instances and continuous assignment statements in that path. [Example 68](#) shows such a conflict.

### *Example 68 Conflicting Delay Modes*

```
'timescale 1 ns / 1 ns
module design (out,in);
output out;
input in;
wire int1,int2;

assign #4 out=int2;

buf #3 buf2 (int2,int1),
 buf1 (int1,in);

specify
 (in => out) = 7;
endspecify
endmodule
```

In [Example 68](#), the module path delay is seven-time units, but the delay specifications distributed along that path add up to ten-time units.

If you include the `+delay_mode_path` compile-time/analysis option, VCS ignores the delay specifications in the primitive instantiation and continuous assignment statements and uses only the module path delay. In [Example 68](#), it uses the seven-time unit delay for propagating signal values through the module.

If you include the `+delay_mode_distributed` compile-time/analysis option, VCS ignores the module path delays and uses the delay in the delay specifications in the primitive instantiation and continuous assignment statements. In [Example 68](#), it uses the ten-time unit delay for propagating signal values through the module.

There are other modes that you can specify:

- If you include the `+delay_mode_unit` compile-time/analysis option, VCS ignores the module path delays and changes the delay specification in all primitive instantiation and continuous assignment statements to the shortest time precision argument of all the `'timescale` compiler directives in the source code. (The default time unit and time precision argument of the `'timescale` compiler directive is 1 s). In [Example 68](#) the `'timescale` compiler directive has a precision argument of 1 ns. VCS might use this 1 ns as the delay, but if the module definition is used in a larger design and there is another `'timescale` compiler directive in the source code with a finer precision argument, then VCS uses the finer precision argument.
- If you include the `+delay_mode_zero` compile-time/analysis option, VCS changes all delay specifications and module path delays to zero.
- If you include none of the compile-time options described in this section, when, as in [Example 68](#), the module path delay does not equal the distributed delays along the path, VCS uses the longer of the two.

---

## Support for Delayed Annotation When No Conditions Have Matched

VCS supports delayed annotations by ignoring condition checking and inserting the least delay from all applicable delays.

When an input changes, it creates an impact on a specific output signal. If no matching conditional arcs extend from inputs (that are toggling) to outputs, the `+ignoreempcond` option ignores conditional checking while applying the least delay from all applicable delays. This can be extended to ignoring condition checking when multiple inputs change simultaneously.

You can enable delayed annotations using the `+ignoreempcond` runtime option.

### Usage Example

Consider that at any given instance, inputs `in1` and `in2` change from 1 to 0 simultaneously. There are no matching arcs for either `in1==0` or `in2==0`.

#### *Example 69 Support for Delayed Annotation*

```
assign out =in1& in2;
specify
 if(in1==1'b1) (in2=>out)=3;
 if(in2==1'b1) (in1=>out)=4;
```

With default behavior, the output changes from 1 to 0 without any delay. Using the `+ignoreempcond` option, the output changes from 1 to 0 with a delay of 3 units.

### Note:

#### Without an Active Specify Path

`+ignoreempcond` is effective only when there is no active specify path.

According to LRM, active specify paths are those where the input has transitioned most recently in time. These paths either have no condition or have their conditions as true.

#### Example:

```
initial begin
 in1 = 1;
 #10 in2 = 1;
 #10 in2 = 0;
end

assign out =in1& in2;
specify
 if (in1==1'b1 && in2==1'b1) (in2=>out)=3;
 if (in2==1'b0 && in1==1'b1) (in1=>out)=4;
```

```
endspecify
```

At time 20, the output changes from 1 to 0. The `+ignorempcond` switch does not annotate the delayed annotation in this case because of the active specify path `if (in2==1'b0 && in1==1'b1) (in1=>out)=4;` that exists here.

### With an Active Specify Path

To enable delayed annotation, use `+ignorempcond=match_min` or `+ignorempcond=match_max` when there is an active specify path.

## Support for Delay Selection Options for Recent Active Paths by Ignoring Condition Checking

VCS supports delayed annotations for recent active paths by ignoring condition checking and inserting applicable delays based on delay selection options.

### Use Model

You can use the following runtime options for delay selections:

- `+ignorempcond=match_max`

VCS applies the maximum delay from the active paths.

- `+ignorempcond=match_min`

VCS applies the minimum delay from the active paths.

### Example

In the following example, both inputs A1 and A2 change simultaneously at time 500ns.

```
`timescale 1ns/1ps
module test_d ;
reg A1,A2,B1;
wire Z;
AO22 u4 (.A1(A1),.A2(A2),.B1(B1),.Z(Z));

initial begin
 $monitor("Time:%t A1=%b, A2=%b, Z=%b",$realtime, A1,A2,Z);
end

initial
 fork
 $sdf_annotate ("dut.sdf",test_d,,, "MAXIMUM");
 A1 = 1;
 A2 = 1;
 B1 = 1;
 #500 A1 = 0; //Both A1 and A2 are changing simultaneously at time 500.
```

```

#500 A2 = 0;
#10000 $finish;
join
endmodule

`timescale 1ns / 1ps
module AO22 (A1,A2,B1,Z);
 input A1,A2,B1;
 output Z;
 and (Z,A1,A2);

 specify
 if (A1 == 1'b1 && A2 == 1'b1 && B1 == 1'b0) (A1 => Z) = (1,1);
 if (A1 == 1'b1 && A2 == 1'b0 && B1 == 1'b0) (A1 => Z) = (1,1);
 if (A1 == 1'b0 && A2 == 1'b0 && B1 == 1'b0) (A2 => Z) = (1,1);
 if (A1 == 1'b0 && A2 == 1'b1 && B1 == 1'b0) (A2 => Z) = (1,1);
 if (A1 == 1'b0 && A2 == 1'b0 && B1 == 1'b1) (B1 => Z) = (1,1);
 endspecify
endmodule

```

### Example Delay File

```

CELL

(CELLTYPE "AO22")

(INSTANCE u4)

(DELAY

(ABSOLUTE

(COND A1==1'b1 && A2==1'b1 && B1==1'b0 (IOPATH A1 Z (2.198::2.798)
(2.299::2.82)))

(COND A1==1'b1 && A2==1'b0 && B1==1'b0 (IOPATH A1 Z (2.018::2.122)
(2.499::2.871)))

(COND A1==1'b0 && A2==1'b0 && B1==1'b0 (IOPATH A2 Z (2.231::2.352)
(2.597::2.87)))

(COND A1==1'b0 && A2==1'b1 && B1==1'b0 (IOPATH A2 Z (2.598::2.798)
(2.235::2.309)))

(COND A1==1'b0 && A2==1'b0 && B1==1'b1 (IOPATH B1 Z (2.018::2.398)
(2.082::2.389)))

)
)
)
```

### Without Delay Selection Options

Delay values are chosen from the following arc where A1, A2 and B1 are matching conditions. However, it is not using the most recent active path, that is, the time of last value change of B1(0) + rise/fall delay (2.398/2.389) < current time stamp (500). Therefore, 0 delay is applied.

```
(COND A1==1'b0 && A2==1'b0 && B1==1'b1 (IOPATH B1 Z (2.398::2.018)
(2.389::2.082)))
```

### With Delay Selection Options

VCS applies the minimum delay from all applicable recent active paths or the maximum delay from all applicable recent active paths. This is done based on user input after ignoring condition checking.

Maximum delay chosen from the following runtime option:

```
+ignorempcond=match_max: max(2.82,2.871,2.87,2.309) = 2.871ns
```

Minimum delay chosen from the following runtime option:

```
+ignorempcond=match_min: min(2.82,2.871,2.87,2.309) = 2.309ns
```

## Specifying Timing Control Attributes in the `+optconfigfile` File

You can specify timing control attributes in the `+optconfigfile` file statements as follows:

- `module {list_of_module_identifiers} {list_of_attributes}`
- `instance {list_of_module_identifiers_and_hierarchical_instance_names} {list_of_attributes}`
- `tree {list_of_module_identifiers} {list_of_attributes}`

The supported attributes are:

- `noSpecify`
- `noIopath`
- `noTiming`

#### Note:

Only the keyword `tree` is recursive and impacts all sub-modules of the specified module, whereas keywords `instance` and `module` do not have recursive effect.

For example,

```
module {mod1}{noSpecify};

instance {mod1.mod2_inst.mod3_inst}{noIopath};

tree {mod1,mod2}{noTiming};
```

There is no limitation prohibiting multiple `+optconfigfile` entries on the `vcs` command line, for example:

```
% vcs +optconfigfile+configfile1_specifying_a_partition \
+optconfigfile+configfile2_specifying_a_partition
```

This example used multiple `+optconfigfile` entries and configuration files to specify different partitions.

## Using the Configuration File to Disable Timing

You can use the VCS configuration file to disable module path delays, specify blocks, and timing checks for module instances that you specify as well as all instances of module definitions that you specify. You use the instance, module, and tree statements to do this just as you do for applying Radiant Technology. The attribute keywords for timing are as follows:

`noIopath`

Disables module path delays in the specified module instances.

`Iopath`

Enables module path delays in the specified module instances.

`noSpecify`

Disables the specify blocks in the specified module instances.

`Specify`

Enables specify blocks in the specified module instances.

`noTiming`

Disables timing checks in the specified module instances.

`Timing`

Enables timing checks in the specified module instances.

## Using the `timopt` Timing Optimizer

The `timopt` timing optimizer can yield large speedups for full-timing gate-level designs. The `timopt` timing optimizer makes its optimizations based on clock signals and sequential devices that it identifies in the design. `timopt` is particularly useful when you use SDF files because SDF files cannot be used with Radiant Technology (+rad).

You enable `timopt` with the `+timopt+clock_period` compile-time option, where the argument is the shortest clock period (or clock cycle) of the clock signals in your design. For example:

```
+timopt+100ns
```

This options specifies that the shortest clock period is 100ns.

`timopt` first displays the number of sequential devices that it finds in the design and the number of these sequential devices to which it might be able to apply optimizations. For example:

```
Total Sequential Elements : 2001
Total Sequential Elements 2001, Optimizable 2001
```

`timopt` then displays the percentage of identified sequential devices to which it can actually apply optimizations followed by messages about the optimization process.

```
TIMOPT optimized 75 percent of the design
Starting TIMOPT Delay optimizations
Done TIMOPT Delay Optimizations
DONE TIMOPT
```

The next step is to simulate the design and see if the optimizations applied by `timopt` produce a satisfactory increase in performance. If you are not satisfied, there are additional steps that you can take to get more optimizations from `timopt`.

If `timopt` is able to identify all the clock signals and all the sequential devices with an absolute certainty, it simply applies its optimizations. If `timopt` is uncertain about the number of clock signals and sequential devices, you can use the following process to maximize `timopt` optimizations:

1. `timopt` writes a configuration file named `timopt.cfg` in the current directory that lists signals and sequential devices that it finds questionable.
2. You review and edit this file, validate that the signals in the file are, or are not, clock signals and that the module definitions in it are, or are not, sequential devices. If you do not need to make any changes in the file, go to Step 5. If you do make changes, go to Step 3.
3. Compile your design again with the `+timopt+clock_period` compile-time option.

`timopt` makes additional optimizations that it did not make, because it was unsure of the signals and sequential devices in the `timopt.cfg` file that it wrote during the first compilation.

4. Look at the `timopt.cfg` file again:

- If `timopt` wrote no new entries for potential clock signals or sequential devices, go to step 5.
- If `timopt` wrote new entries, but you make no changes to the new entries, go to step 5.
- If you make modifications to the new entries, return to step 3.

5. `timopt` does not need to look for any more clock signals and it can assume that the `timopt.cfg` file correctly specifies clock signal and sequential devices. At this point, it just needs to apply the latest optimizations. Compile your design one more time, including the `+timopt` compile-time option, but without its `+clock_period` argument.

6. You now simulate your design using `timopt` optimizations. `timopt` monitors the simulation and makes its optimizations based on its analysis of the design and information in the `timopt.cfg` file. During simulation, if it finds that its assumptions are incorrect, for example, the clock period for a clock signal is incorrect, or there is a port for asynchronous control on a module for a sequential device, `timopt` displays a warning message similar to the following:

```
+ Timopt Warning: for clock testbench.clockgen..clk: TimePeriod
50ns Expected 100ns
```

## Editing the `timopt.cfg` File

When editing the `timopt.cfg` file, first edit the potential sequential device entries. Edit the potential clock signal only when you have made no changes to the entries for sequential devices.

## Editing Potential Sequential Device Entries

The following is an example of potential sequential devices:

```
// POTENTIAL SEQUENTIAL CELLS
// flop {jknpn} {},;
// flop {jknpc} {},;
// flop {tfnpc} {},;
```

You can remove the comment marks for the module definitions that are, in fact, model sequential devices and which provide the clock port, clock polarity, and optionally asynchronous ports.

A modified list might look like the following:

```
flop { jknpn } { CP, true};
flop { jknpc } { CP, true, CLN};
flop { tfnpc } { CP, true, CLN};
```

In this example, `CP` is the clock port and the `true` keyword indicates that the sequential device is triggered on the posedge of the clock port and `CLN` is an asynchronous port.

If you uncomment any of these module definitions, then `timopt` might identify additional clock signals that drive these sequential devices. To enable `timopt` to do this:

1. Remove the clock signal entries from the `timopt.cfg` file.
2. Recompile the design with the same `+timopt+clock_period` compile-time option.

`timopt` writes new clock signal entries in the `timopt.cfg` file.

## Editing Clock Signal Entries

The following is an example of the clock signal entries:

```
clock {
 // test.badClock , // 1
 test.goodClock // 2000
} {100ns};
```

These clock signals have a period of `100ns` or longer. This time value comes from the `+clock_period` argument that you added to the `+timopt` compile-time option when you first compiled the design. The entry for the signal `test.badClock` is commented out because it connects to a small percentage of the sequential devices in the design. In this instance, it is only 1 of the 2001 sequential devices that it identified in the design. The entry for the signal `test.goodClock` is not commented out because it connects to a large percentage of the sequential devices. In this instance, it is 2000 of the 2001 sequential devices in the design.

To make `timopt` use a commented out clock signal when it optimizes the design in a subsequent compilation, remove the comment characters preceding the signal's hierarchical name.

## Using Scan Simulation Optimizer

Scan Simulation Optimizer (`scanopt`) yields large speed-ups when used with Serial Scan DFT simulations. The optimizations are done based on the scan cells that are identified in the design. This optimization is applicable only on the Serial Scan DFT designs, using scan flops built with the MUX-FLOP combination.

The optimizer applies its optimization after scan flops in the design are identified. There is an option for providing all the scan flops in the design through a configuration file,

`scanopt.cfg`, in the current directory. This can be used if the optimizer fails to identify the scan flops, thereby, not producing a satisfactory performance improvement.

For example, for a design with shortest clock period of 100ns, use the following compile-time option:

```
-scanopt=100ns
```

You can supply the list of scan flops in the file, `scanopt.cfg` using the format specified in the following section, and then use the following compile-time option.

```
-scanopt=cfg
```

This enables the optimizer to pick up the scan flops specified in the configuration file and use for its optimization.

The optimizer also determines the length of the scan chain(s) on its own. If there are multiple scan chains, the minimal scan length is chosen for optimizations.

This chapter discusses the following topics:

- [ScanOpt Configuration File Format](#)
- [ScanOpt Assumptions](#)
- [Improving the ScanOpt for Debug Support](#)

## ScanOpt Configuration File Format

The following format must be used for specifying a scan flop:

```
BEGIN_FLOP <scan_cell_name>
 BEGIN_PORT
 Q_PORT <q_port_name>
 [QN_PORT <qn_port_name>]
 D_PORT <d_port_name>
 TI_PORT <ti_port_name>
 TE_PORT <te_port_name>
 END_PORT
END_FLOP
```

The section between `BEGIN_FLOP` and `END_FLOP` corresponds to one particular scan flop. The `<scan_cell_name>` field corresponds to the name of scan flop (scan cell). Multiple sections can be used to specify multiple scan flops.

The section between `BEGIN_PORT` and `END_PORT` also corresponds to ports of the scan flop. Specifying `Q_PORT`, `D_PORT`, `TI_PORT`, and `TE_PORT` are mandatory, whereas `QN_PORT` could be optional.

---

## ScanOpt Assumptions

### Combinational Path Delays

By default, the optimizer assumes that the worst case delay for any combinational path in the design is not more than five times the shortest clock period and applies the optimizations. The following banner is printed at the compile time to indicate this assumption to you:

*"ScanOpt assumes that no combinational path has worst-case delay more than 5 clock period. Please use, "-scanopt=cdel=" to override the assumed value"*

For example, for a design with shortest clock period of 100ns, if the default value of 5 is to be overridden with a value of 10, you can use the following compile-time option.

```
-scanopt=100ns,cdel=10
```

### Length of Test Cycles

The optimizer assumes that the simulation remains in the test mode for at least the scan chain length times the shortest clock period. Any violation of this assumption is automatically detected during the simulation, and the following error message is displayed quitting the simulation.

*"Error: Simulation has been aborted due to fatal violation of ScanOpt assumptions.  
Please refer to the documentation for more details. To get around this error, please rerun simulation with "-noscanopt" switch"*

For example, if the inferred length of scan chain in the design is 5000 and the short clock period is 100ns, then the Test enable signal(s) should remain in test mode for at least 500000ns (that is,  $5000 * 100\text{ns}$ ).

**Note:**

The `-noscanopt` option can be used at runtime, thereby avoiding re-compilation of the design.

---

## Improving the ScanOpt for Debug Support

VCS enhances the `scanopt` performance even in the presence of debug capabilities using the `-scanopt=nodumpcombo` option and speeds up the design execution when you run the design in `scanopt` mode with debug capabilities.

## Use Model

The feature has the following use model:

```
% vcs <file> <other options> -scanopt=relaxdbg,nodumpcombo (with or without
the -hsopt=gates option)
```

```
% simv
```

The following table describes the options:

|                      |                                                                                                                                                                                                          |
|----------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
|                      | -scanopt=nodumpcombo                                                                                                                                                                                     |
| With -hsopt=gates    | Displays a warning message if you access any signal in scan optimized hierarchy using the PLI/VPI/UCLI call for read/write/force values or for callback. Skips the signals from the generated FSDB file. |
| Without -hsopt=gates | Displays a warning message during compilation.                                                                                                                                                           |

### Note:

The `-scanopt=nodumpcombo` option is useful only when the FSDB file is dumped along with the `-hsopt=gates` option. When the `-scanopt=nodumpcombo` option is specified, VCS generates the FSDB file that do not contain any signal in scan optimized hierarchy. Therefore, the nCompare Waveform Comparison module does not display any difference between the FSDB file generated with and without the `scanopt` optimization.

## Usage Example

Consider the following examples:

```
test.v

module top;
 reg E,SE,CP,SI,SI_1,D;
 reg CDN;
 wire Q, Q_1, QN;
 wire w_1_4 = Q ^ 1;
 wire w_2_4 = Q_1 ^ 1;

 initial begin
 #100;
 SE = 1'b0;
 $display("===== Performing FORCE, READ and CALLBACK when SE = 0 =====");
 $forceData();
 #1000;
 SE = 1'b1;
```

```

#300;
$display("===== Performing FORCE, READ and CALLBACK when SE = 1
=====");
$forceData();
#2000;
$finish;
end
always
begin
 SE <= 1;
 D <= generateLogicValue();
.....
#10;
end

always
begin
 SI <= generateLogicValue();
 SI_1 <= generateLogicValue();
 #100;
end
scan_cell
inst_1(.E(E), .SE(SE), .CP(CP), .SI(SI_1), .D(w_1_4), .CDN(CDN), .Q(Q_1)
, .QN(QN));
endmodule

`celldefine
module scan_cell(E, SE, CP, SI, D, CDN, Q, QN);
 input E,SE,CP,SI;
 input D;
 input CDN;
 output Q;
 output QN;
 mux_udp (D1, Q_buf, D, E);
 dff_udp (Q_buf, D2, CP, CDN_i, SDN, notifier);
 buf_udp (xCP_check, CP_check, 1'b1);
 buf (CDN_i, CDN);
endmodule
`endcelldefine

test.c

#include "stdio.h"
#include "sv_vpi_user.h"

int forceData (char *data)
{
 vpiHandle frch,frcl;
 vpiHandle GH;
 s_vpi_value valG;
 GH = vpi_handle(vpiSystfCall,NULL);
 frcl =vpi_iterate(vpiArgument,GH);
 frch = vpi_handle_by_name("top.inst_1.D",NULL);

```

```

 valG.format = vpiScalarVal;
 valG.value.scalar = 0;
 vpi_put_value(frcH,&valG,NULL,vpiForceFlag);
}

test.tab

$forceData call=forceData check=forceData acc+=frc:*

```

Compile and simulate the examples as follows:

```

% vcs -sverilog test.v -P test.tab test.c -debug_access+all
-scanopt=relaxdbg,nodumpcombo -hsopt=gates

% simv

```

VCS generates the following warning message with the `-scanopt=relaxdbg,nodumpcombo` option:

```

Warning-[VPI-FORCE-SCAN] VPI force on scanopt hierarchy
At time 1410, in PLI routine called from test.v, 20
VPI force on scan optimized signal 'top.inst_1.D' may not have any
effect.

```

**Note:**

The above message indicates that, at time 1410, the signal from the combo cloud connected to `D` pin of the scan cell is accessed through PLI/VPI/UCLI call when the Scan Enable (SE) is high. When you use the `nodumpcombo` option, a warning message is generated and the simulation continues as scan optimized hierarchy is accessed. The values that are dumped after SE is high and are valid values.

## Negative Timing Checks

Negative timing checks are either `$setuphold` timing checks with negative setup or hold limits, or `$recrem` timing checks with negative recovery or removal limits.

This following sections describe their purpose, how they work, and how to use them:

- [The Need for Negative Value Timing Checks](#)
- [Enabling Negative Timing Checks](#)
- [Other Timing Checks Using the Delayed Signals](#)
- [IOPATH Delay Annotation Using Delayed Signals](#)
- [Checking Conditions](#)
- [Toggling the Notifier Register](#)

- [SDF Back-Annotation to Negative Timing Checks](#)
- [How VCS Calculates Delays](#)

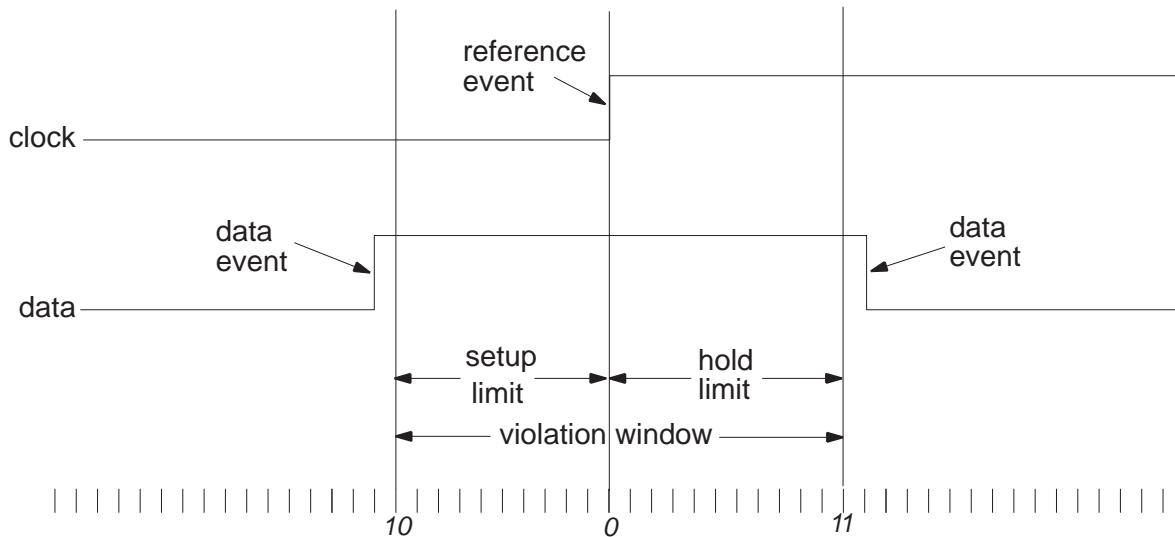
## The Need for Negative Value Timing Checks

The `$setuphold` timing check defines a timing violation window of a specified amount of simulation time before and after a reference event. For example, a transition on a clock signal, in which a data signal must remain constant. A transition on the data signal, called a data event, during the specified window is a timing violation. For example:

```
$setuphold (posedge clock, data, 10, 11, notifyreg);
```

In this example, VCS reports the timing violation if there is a transition on signal `data` less than `10` time units before, or less than `11` time units after, a rising edge on signal `clock`. When there is a timing violation, VCS toggles a notify register, in this example, `notifyreg`. You could use this toggling of a notify register to output an `x` value from a device, such as a sequential flop, when there is a timing violation.

*Figure 151 Positive Setup and Hold Limits*



In this example, both the setup and hold limits have positive values. When this occurs, the violation window straddles the reference event.

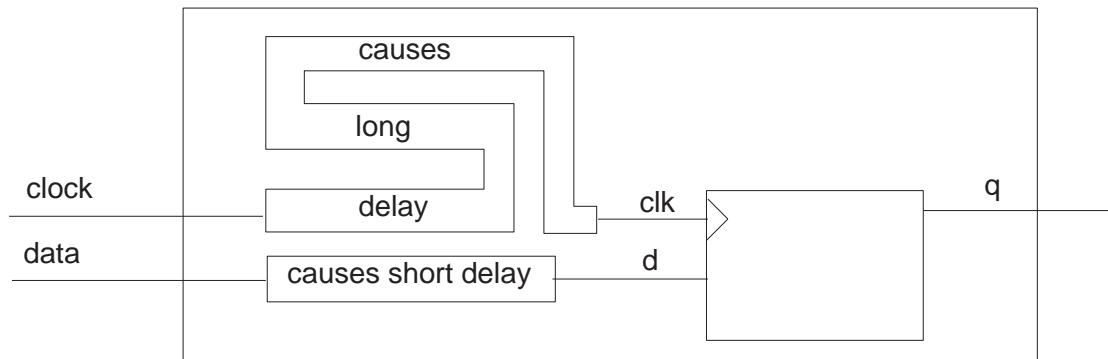
There are cases where the violation window cannot straddle the reference event at the inputs of an ASIC cell. Such a case occurs when:

- The data event takes longer than the reference event to propagate to a sequential device in the cell.
- Timing must be accurate at the sequential device.
- You need to check for timing violations at the cell boundary.

It also occurs when the opposite is true, that is, when the reference event takes longer than the data event to propagate to the sequential device.

When this happens, use the `$setuphold` timing check in the top-level module of the cell to look for timing violations when signal values propagate to that sequential device. In this case, you need to use negative setup or hold limits in the `$setuphold` timing check.

*Figure 152 ASIC Cell With Long Propagation Delays on Reference Events*

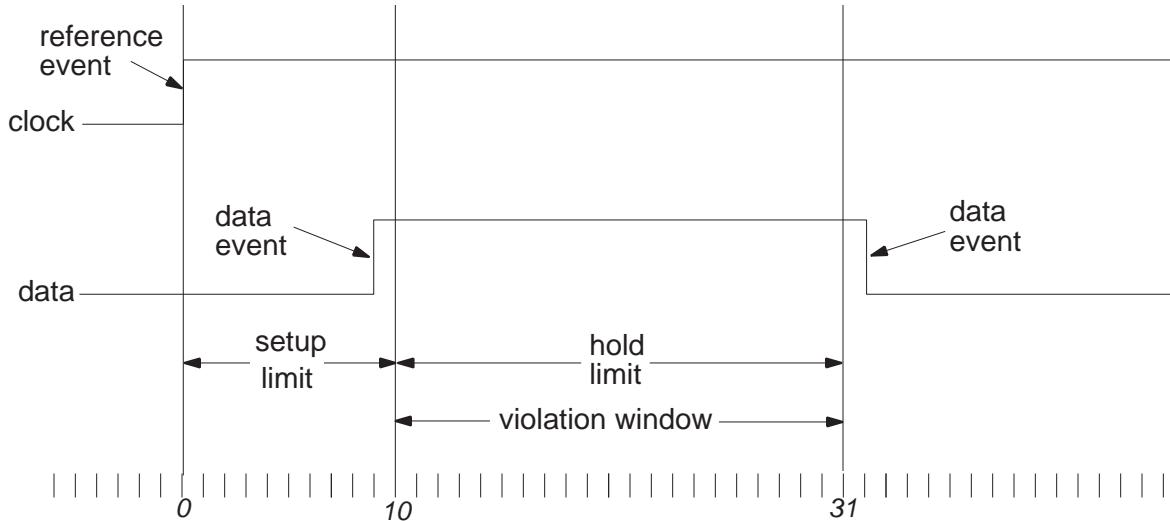


When this occurs, the violation window shifts at the cell boundary so that it no longer straddles the reference event. It shifts to the right when there are longer propagation delays on the reference event. This right shift requires a negative setup limit:

```
$setuphold (posedge c lock, data, -10, 31, notifyreg);
```

[Figure 153](#) illustrates this scenario.

Figure 153 Negative Setup Limit



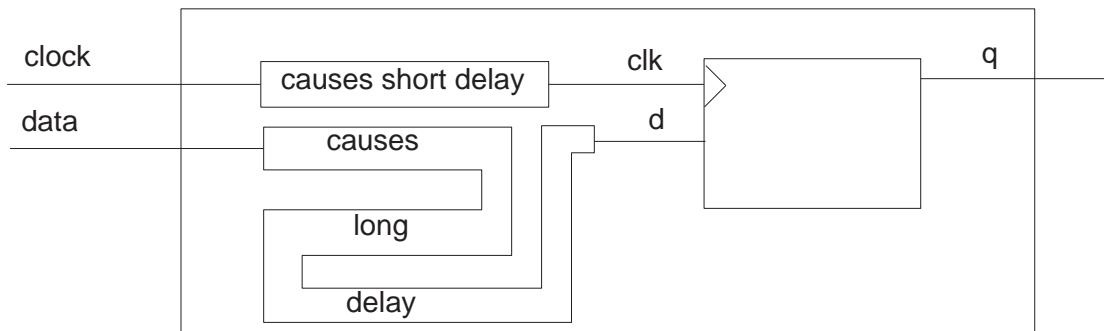
In this example, the `$setuphold` timing check is in the specify block of the top-level module of the cell. It specifies that there is a timing violation if there is a data event between 10 and 31 time units after the reference event on the cell boundary.

This is giving the reference event a “head start” at the cell boundary, anticipating that the delays on the reference event allow the data events to “catch up” at the sequential device inside the cell.

**Note:**

When you specify a negative setup limit, its value must be less than the hold limit.

Figure 154 ASIC Cell With Long Propagation Delays on Data Events

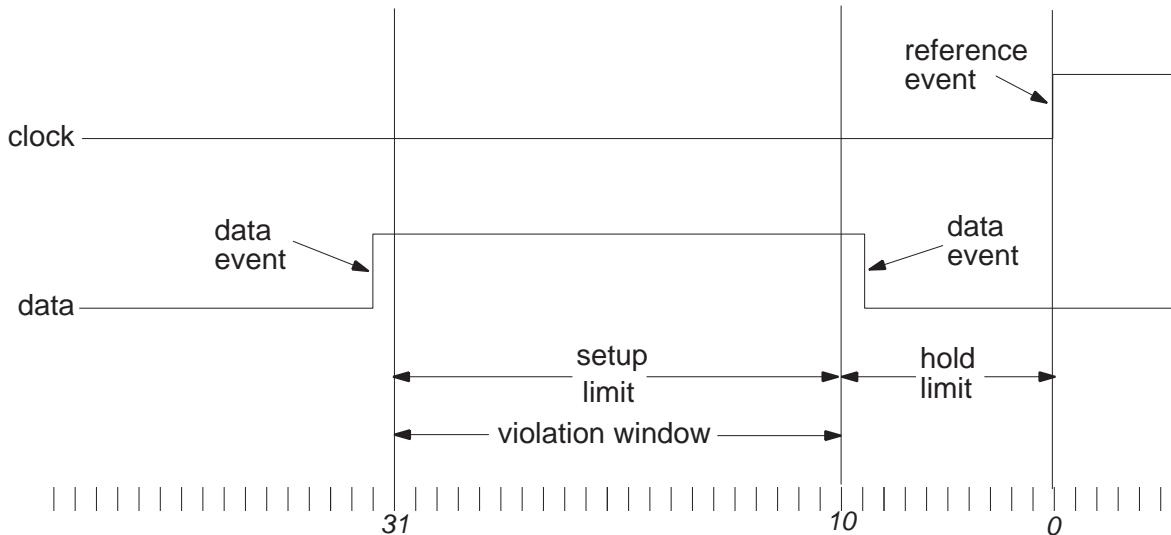


The violation window shifts to the left when there are longer propagation delays on the data event. This left shift requires a negative hold limit:

```
$setuphold (posedge clock, data, 31, -10, notifyreg);
```

[Figure 155](#) illustrates this scenario.

*Figure 155 Negative Hold Limit*



In this example, the `$setuphold` timing check is in the specify block of the top-level module of the cell. It specifies that there is a timing violation if there is a data event between 31 and 10 time units before the reference event on the cell boundary.

This is giving the data events a “head start” at the cell boundary, anticipating that the delays on the data events allow the reference event to “catch up” at the sequential device inside the cell.

**Note:**

When you specify a negative hold limit, its value must be less than the setup limit.

To implement negative timing checks, VCS creates delayed versions of the signals that carry the reference and data events and an alternative violation window where the window straddles the delayed reference event.

You can specify the names of the delayed versions by using the extended syntax of the `$setuphold` system task, or by allowing VCS to name them internally.

The extended syntax also allows you to specify expressions for additional conditions that must be true for a timing violation to occur.

## The \$setuphold Timing Check Extended Syntax

The `$setuphold` timing check has the following extended syntax:

```
$setuphold(reference_event, data_event, setup_limit,
hold_limit, notifier, [timestamp_cond, timecheck_cond,
delayed_reference_signal, delayed_data_signal]);
```

The following additional arguments are optional:

`timestamp_cond`

This argument specifies the condition which determines whether or not VCS reports a timing violation.

In the setup phase of a `$setuphold` timing check, VCS records or “stamps” the time of a data event internally. When a reference event occurs, it can compare the times of these events to see if there is a setup timing violation. If the condition specified by this argument is false, VCS does not record or “stamp” the data event so there cannot be a setup timing violation.

Similarly, in the hold phase of a `$setuphold` timing check, VCS records or “stamps” the time of a reference event internally. When a data event occurs, it can compare the times of these events to see if there is a hold timing violation. If the condition specified by this argument is false, VCS does not record or “stamp” the reference event, so there cannot be a hold timing violation.

`timecheck_cond`

This argument specifies the condition which determines whether or not VCS reports a timing violation.

In the setup phase of a `$setuphold` timing check, VCS compares or “checks” the time of the reference event with the time of the data event to see if there is a setup timing violation. If the condition specified by this argument is false, VCS does not make this comparison and so there is no setup timing violation.

Similarly, in the hold phase of a `$setuphold` timing check, VCS compares or “checks” the time of a data event with the time of a reference event to see if there is a hold timing violation. If the condition specified by this argument is false, VCS does not make this comparison and so there is no hold timing violation.

`delayed_reference_signal`

The name of the delayed version of the reference signal.

`delayed_data_signal`

The name of the delayed version of the data signal.

The following example demonstrates how to use the extended syntax:

```
$setuphold(ref, data, -4, 10, notifrl, stampreg==1, , d_ref,
 d_data);
```

In this example, the `timestamp_cond` argument specifies that `reg stampreg` must equal 1 for VCS to “stamp” or record the times of data events in the setup phase or “stamp” the times of reference events in the hold phase. If this condition is not met, and stamping does not occur, VCS does not find timing violations no matter what the time is for these events. Also in the example, the delayed versions of the reference and data signals are named `d_ref` and `d_data`.

You can use these delayed signal versions of the signals to drive sequential devices in your cell model. For example:

```
module DFF(D,RST,CLK,Q);
 input D,RST,CLK;
 output Q;
 reg notifier;
 DFF_UDP d2(Q,dCLK,dD,dRST,notifier);
 specify
 (D => Q) = 20;
 (CLK => Q) = 20;
 $setuphold(posedge CLK,D,-5,10,notifier,,,dCLK,dD);
 $setuphold(posedge CLK,RST,-8,12,notifier,,,dCLK,
 dRST);
 endspecify
endmodule

primitive DFF_UDP(q,clk,data,rst,notifier);
 output q; reg q;
 input data,clk,rst,notifier;

table
// clock data rst notifier state q
// -----
r 0 0 ? : ? : 0 ;
r 1 0 ? : ? : 1 ;
f ? 0 ? : ? : - ;
? ? r ? : ? : 0 ;
? * ? ? : ? : - ;
? ? ? * : ? : x ;
endtable
endprimitive
```

In this example, the `DFF_UDP` user-defined primitive is driven by the delayed signals `dCLK`, `dD`, `dRST`, and the `notifier` `reg`.

## Negative Timing Checks for Asynchronous Controls

The `$recrem` timing check is used for checking how close asynchronous control signal transitions are to clock signals. Similar to the setup and hold limits in `$setuphold` timing checks, the `$recrem` timing check has recovery and removal limits. The recovery limit specifies how much time must elapse after a control signal toggles from its active state before there is an active clock edge. The removal limit specifies how much time must elapse after an active clock edge before the control signal can toggle from its active state.

In the same way a reference signal, such as a clock signal and data signal can have different propagation delays from the cell boundary to a sequential device inside the cell, there can be different propagation delays between the clock signal and the control signal. For this reason, there can be negative recovery and removal limits in the `$recrem` timing check.

### The `$recrem` Timing Check Syntax

The `$recrem` timing check syntax is very similar to the extended syntax for `$setuphold`:

```
$recrem(reference_event, data_event, recovery_limit,
removal_limit, notifier, [timestamp_cond, timecheck_cond,
delayed_reference_signal, delayed_data_signal]);
```

`reference_event`

Typically, the reference event is the active edge on a control signal, such as a clear signal. Specify the active edge with the `posedge` or `negedge` keyword.

`data_event`

Typically, the data event occurs on a clock signal. Specify the active edge on this signal with the `posedge` or `negedge` keyword.

`recovery_limit`

Specifies how much time must elapse after a control signal, such as a clear signal toggles from its active state (the reference event), before there is an active clock edge (the data event).

`removal_limit`

Specifies how much time must elapse after an active clock edge (the data event), before the control signal can toggle from its active state (the reference event).

`notifier`

A register whose value VCS toggles when there is a timing violation.

`timestamp_cond`

This argument specifies the condition which determines whether or not VCS reports a timing violation.

In the recovery phase of a `$recrem` timing check, VCS records or “stamps” the time of a reference event internally. When a data event occurs it can compare the times of these events to see if there is a recovery timing violation. If the condition specified by this argument is false, VCS does not record or “stamp” the reference event so there cannot be a recovery timing violation.

Similarly, in the removal phase of a `$recrem` timing check, VCS records or “stamps” the time of a data event internally. When a reference event occurs, it can compare the times of these events to see if there is a removal timing violation. If the condition specified by this argument is false, VCS does not record or “stamp” the data event so there cannot be a removal timing violation.

`timecheck_cond`

This argument specifies the condition which determines whether or not VCS reports a timing violation.

In the recovery phase of a `$recrem` timing check, VCS compares or “checks” the time of the data event with the time of the reference event to see if there is a recovery timing violation. If the condition specified by this argument is false, VCS does not make this comparison and so there is no recovery timing violation.

Similarly, in the removal phase of a `$recrem` timing check, VCS compares or “checks” the time of a reference event with the time of a data event to see if there is a removal timing violation. If the condition specified by this argument is false, VCS does not make this comparison and so there is no removal timing violation.

`delayed_reference_signal`

The name of the delayed version of the reference signal, typically a control signal.

`delayed_data_signal`

The name of the delayed version of the data signal, typically a clock signal.

## Enabling Negative Timing Checks

To use a negative timing check you must include the `+neg_tchk` compile-time option when you compile your design. If you omit this option, VCS changes all negative limits to 0.

If you include the `+no_notifier` compile-time option with the `+neg_tchk` option, you only disable notifier toggling. VCS still creates the delayed versions of the reference and data signals and displays timing violation messages.

Conversely, if you include the `+no_tchk_msg` compile-time option with the `+neg_tchk` option, you only disable timing violation messages. VCS still creates the delayed

versions of the reference and data signals and toggles notifier regs when there are timing violations.

If you include the `+neg_tchk` compile-time option but also include the `+notimingcheck` or `+nospecify` compile-time options, VCS does not compile the `$setuphold` and `$recrem` timing checks into the `simv` executable. However, it does create the signals that you specified in the `delayed_reference_signal` and `delayed_data_signal` arguments, and you can use these to drive sequential devices in the cell. Note that there is no delay on these "delayed" arguments and they have the same transition times as the signals specified in the `reference_event` and `data_event` arguments.

Similarly, if you include the `+neg_tchk` compile-time option and then include the `+notimingcheck` runtime option instead of the compile-time option, you disable the `$setuphold` and `$recrem` timing checks that VCS compiled into the executable. At compile time, VCS creates the signals that you specified in the `delayed_reference_signal` and `delayed_data_signal` arguments, and you can use them to drive sequential devices in the cell, but the `+notimingcheck` runtime option used at compile time disables the delay on these "delayed" versions.

## Other Timing Checks Using the Delayed Signals

When you enable negative timing limits in the `$setuphold` and `$recrem` timing checks, and have VCS create delayed versions of the data and reference signals, by default the other timing checks also use the delayed versions of these signals. You can prevent the other timing checks from doing this with the `+old_ntc` compile-time option.

Having the other timing checks use the delayed versions of these signals is particularly useful when the other timing checks use a notifier register to change the output of the sequential element to `x`.

### *Example 70 Notifier Register Example for Delayed Reference and Data Signals*

```

`timescale 1ns/1ns

module top;
 reg clk, d;
 reg rst;
 wire q;

 dff dff1(q, clk, d, rst);

 initial begin
 $monitor($time,,clk,,d,,q);
 rst = 0; clk = 0; d = 0;
 #100 clk = 1;
 #100 clk = 0;
 #10 d = 1;
 #90 clk = 1;
 #1 clk = 0; // width violation
 end
endmodule

```

```

#100 $finish;
end
endmodule

module dff(q, clk, d, rst);
 output q;
 input clk, d, rst;
 reg notif;

 DFF_UDP(q, d_clk, d_d, d_rst, notif);

 specify
 $setuphold(posedge clk, d, -10, 20, notif, , , d_clk,
 d_d);
 $setuphold(posedge clk, rst, 10, 10, notif, , , d_clk,
 d_rst);
 $width(posedge clk, 5, 0, notif);
 endspecify
endmodule

primitive DFF_UDP(q,data,clk,rst,notifier);
output q; reg q;
input data,clk,rst,notifier;

table
// clock data rst notifier state q
// -----
 r 0 0 ? : ? : 0 ;
 r 1 0 ? : ? : 1 ;
 f ? 0 ? : ? : - ;
 ? ? r ? : ? : 0 ;
 ? * ? ? : ? : - ;
 ? ? ? * : ? : x ;
endtable
endprimitive

```

In this example, if you include the `+neg_tchk` compile-time option, the `$width` timing check uses the delayed version of signal `clk`, named `d_clk`, and the following sequence of events occurs:

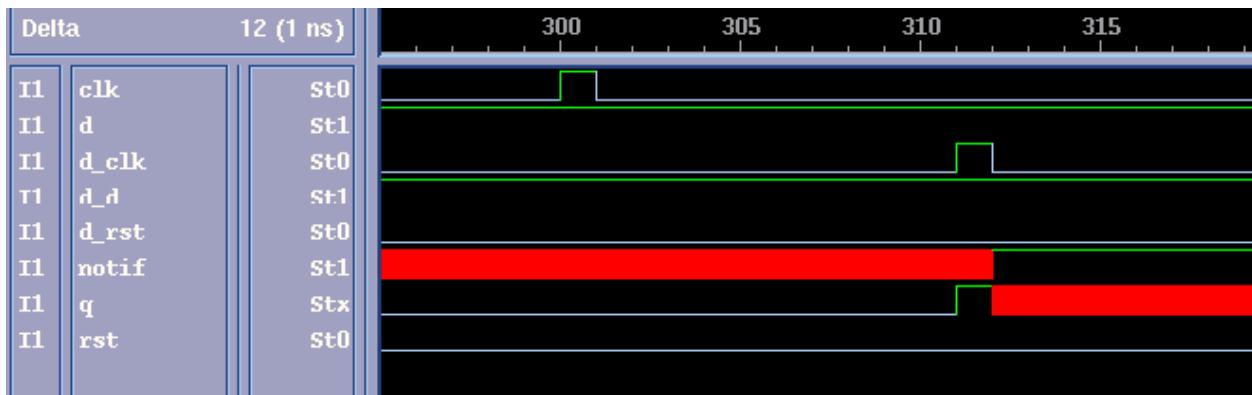
1. At time 311, the delayed version of the clock transitions to 1, causing output `q` to toggle to 1.
2. At time 312, the narrow pulse on the clock causes a width violation:

```
"test1.v", 31: Timing violation in top.dff1
$width(posedge clk:300, : 301, limit: 5);
```

The timing violation message looks like it occurs at time 301, but you do not see it until time 312.

3. Also at time 312, reg `notif` toggles from `x` to `1`. This changes output `q` from `1` to `x`. There are no subsequent changes on output `q`.

*Figure 156 Other Timing Checks Using the Delayed Versions*



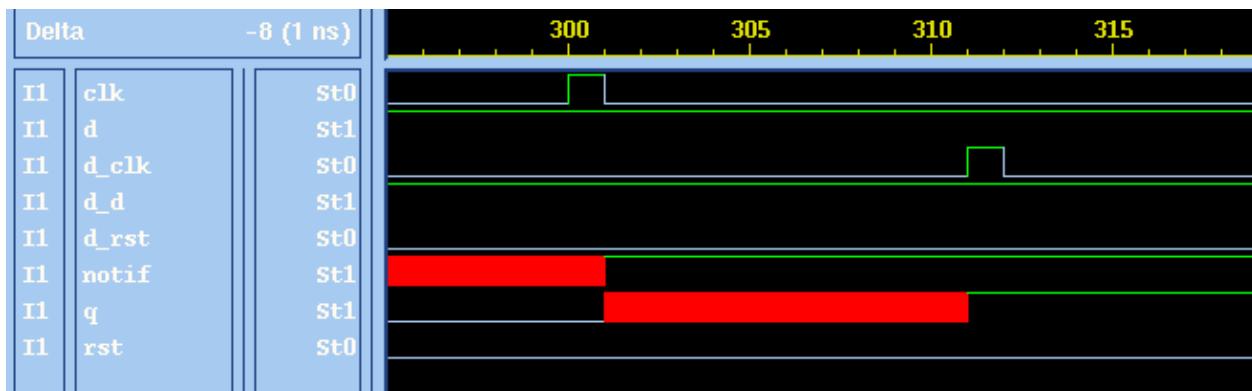
If you include both the `+neg_tchk` and `+old_ntc` compile-time options, the `$width` timing check does not use the delayed version of signal `clk`, causing the following sequence of events to occur:

1. At time 301, the narrow pulse on signal `clk` causes a width violation:

```
"test1.v", 31: Timing violation in top.dff1
$width(posedge clk:300, : 301, limit: 5);
```

2. Also at time 301, the notifier reg named `notif` toggles from `x` to `1`. In turn, this changes the output `q` of the user-defined primitive `DFF_UDP` and module instance `dff1` from `0` to `x`.
3. At time 311, the delayed version of signal `clk`, named `d_clk`, reaches the user-defined primitive `DFF_UDP`, thereby changing the output `q` to `1`, erasing the `x` value on this output.

*Figure 157 Other Timing Checks Not Using the Delayed Versions*



The timing violation, as represented by the `x` value, is lost to the design. If a module path delay that is greater than ten time units was used for the module instance, the `x` value would not appear on the output at all.

For this reason, Synopsys does not recommend using the `+old_ntc` compile-time option. It exists only for unforeseen circumstances.

## IOPATH Delay Annotation Using Delayed Signals

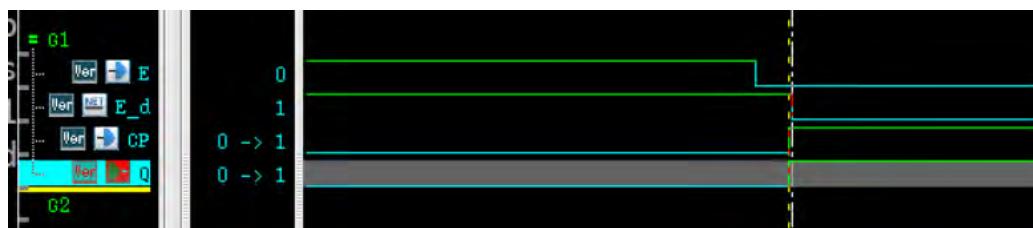
IOPATH specifies the delays on a legal path from an input port to an output port. When you enable negative timing limits in the `$setuphold` and `$recrem` timing checks, and have VCS create delayed versions of the data and reference signals, by default the other timing checks also use the delayed versions of these signals. However, VCS uses the original signals to determine the path of IOPATH delay.

### Example

```
specify
if (E == 1'b1) (CP => Q) = (10);
if (E == 1'b0) (CP => Q) = (0);
...
$setuphold (posedge CP, posedge E , -2, 4, , , CP_d, E_d);
...
endspecify
```

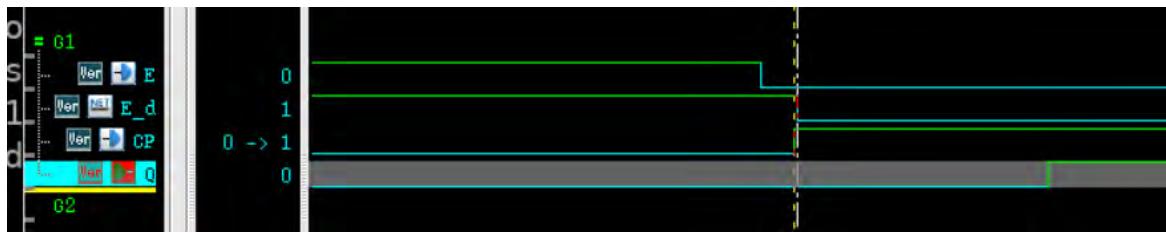
In the waveform, `E_d` is the delayed version of `E` due to negative timing check. While evaluating the delay from `CP` to `Q`, the value of `E` is used. As a result, `CP` to `Q` has no delay in this case (`E==1'b0` determines the path).

*Figure 158 Use of Original Signals to Determine IOPATH Delay*



However, if you specify the `-iopath_anno_del_sig` compile-time option, VCS uses the delayed versions of the signals to determine the path of IOPATH delay (`E_d==1'b1` determines the path).

Figure 159 Use of Delayed Signals to Determine IOPATH Delay



## Checking Conditions

VCS evaluates the expressions in the `timestamp_cond` and `timecheck_cond` arguments in either of the following cases:

- When there is a value change on the original reference and data signals at the cell boundary
- When the value changes propagate from the delayed versions of these signals at the sequential device inside the cell.

It decides when to evaluate the expressions depending on which signals are the operands in these expressions. Note the following:

- It does not matter when VCS evaluates these expressions:
  - If the operands in these expressions are neither the original nor the delayed versions of the reference or data signals
  - If these operands are signals that do not change value between value changes on the original reference and data signals and their delayed versions
- If the operands in these expressions are delayed versions of the original reference and data signals, then you want VCS to evaluate these expressions when there are value changes on the delayed versions of the reference and data signals. VCS does this by default.
- If the operands in these expressions are the original reference and data signals and not the delayed versions, then you want VCS to evaluate these expressions when there are value changes on the original reference and data signals. To specify evaluating these expressions when the original reference and data signals change value, include the `+NTC2` compile-time option.

## Toggling the Notifier Register

VCS waits for a timing violation to occur on the delayed versions of the reference and data signals before toggling the notifier register. Toggling means the following value changes:

- x to 0
- 0 to 1
- 1 to 0

VCS does not change the value of the notifier register if you have assigned a `z` value to it.

## SDF Back-Annotation to Negative Timing Checks

You can back-annotate negative setup and hold limits from SDF files to `$setuphold` timing checks and negative recovery and removal limits from SDF files to `$recrem` timing checks, if the following conditions are met:

- You included the arguments for the names of the delayed reference and data signals in the timing checks.
- You compiled your design with the `+neg_tchk` compile-time option.
- For all `$setuphold` timing checks, the positive setup or hold limit is greater than the negative setup or hold limit.
- For all `$recrem` timing checks, the positive recovery or removal limit is greater than the negative recovery or removal limit.

As documented in the OVI SDF3.0 specification:

- `TIMINGCHECK` statements in the SDF file back-annotate timing checks in the model which match the edge and condition arguments in the SDF statement.
- If the SDF statement specifies `SCOND` or `CCOND` expressions, they must match the corresponding `timestamp_cond` or `timecheck_cond` in the timing check declaration for back-annotation to occur.
- If there is no `SCOND` or `CCOND` expressions in the SDF statement, all timing checks that otherwise match are back-annotated.

## How VCS Calculates Delays

This section describes how VCS calculates the delays of the delayed versions of reference and data signals. It does not describe how you use negative timing checks; it is supplemental material intended for users who would like to read more about how negative timing checks work in VCS.

VCS uses the limits you specify in the `$setuphold` or `$recrem` timing check to calculate the delays on the delayed versions of the reference and data signals. For example:

```
$setuphold(posedge clock,data,-10,20, , , del_clock,
 del_data);
```

This specifies that the propagation delays on the reference event (a rising edge on signal `clock`), are more than `10` but less than `20` time units more than the propagation delays on the data event (any transition on signal `data`).

So when VCS creates the delayed signals, `del_clock` and `del_data`, and the alternative violation window that straddles a rising edge on `del_clock`, VCS uses the following relationship:

```
20 > (delay on del_clock - delay on del_data) > 10
```

There is no reason to make the delays on either of these delayed signals any longer than they have to be so the delay on `del_data` is `0` and the delay on `del_clock` is `11`. Any delay on `del_clock` between `11` and `19` time units would report a timing violation for the `$setuphold` timing check.

Multiple timing checks, that share reference or data events, and specified delayed signal names, can define a set of delay relationships. For example:

```
$setuphold(posedge CP,D,-10,20, notifier, ,
 del_CP, del_D);
$setuphold(posedge CP,TI,20,-10, notifier, ,
 del_CP, del_TI);
$setuphold(posedge CP,TE,-4,8, notifier, ,
 del_CP, del_TE);
```

In this example:

- The first `$setuphold` timing check specifies the delay on `del_CP` is more than `10` but less than `20` time units more than the delay on `del_D`.
- The second `$setuphold` timing check specifies the delay on `del_TI` is more than `10` but less than `20` time units more than the delay on `del_CP`.
- The third `$setuphold` timing check specifies the delay on `del_TE` is more than `4` but less than `8` time units more than the delay on `del_TE`.

Therefore:

- The delay on `del_D` is `0` because its delay does not have to be more than any other delayed signal.
- The delay on `del_CP` is `11` because it must be more than `10` time units more than the `0` delay on `del_D`.

- The delay on `del_TE` is 4 because the delay on `del_CP` is 11. The 11 makes the possible delay on `del_TE` larger than 3, but less than 7. The delay cannot be 3 or less, because the delay on `del_CP` is less than 8 time units more than the delay on `del_TE`. VCS makes the delay 4 because it always uses the shortest possible delay.
- The delay on `del_TI` is 22 because it must be more than 10 time units more than the 11 delay on `del_CP`.

In unusual and rare circumstances, multiple `$setuphold` and `$recrem` timing checks, including those that have no negative limits, can make the delays on the delayed versions of these signals mutually exclusive. When this happens, VCS repeats the following procedure until the signals are no longer mutually exclusive:

1. Sets one negative limit to 0.
2. Recalculates the delays of the delayed signals.

## Using VITAL Models and Netlists

You use VCS to validate and optimize a VHDL initiative toward ASIC libraries (VITAL) model and to simulate a VITAL-based netlist. Typically, library developers optimize the VITAL model, and designers simulate the VITAL-based netlist.

The library developer uses a single ASIC cell from the system, verifies its correctness, and optimizes that single cell. The designer simulates large numbers of cells, organized in a netlist, by applying test vectors and timing information.

This section describes how to validate and optimize a VITAL model and how to simulate a VITAL netlist. It contains the following sections:

- [Validating and Optimizing a VITAL Model](#)
- [Simulating a VITAL Netlist](#)
- [Understanding VITAL Timing Delays and Error Messages](#)

## Validating and Optimizing a VITAL Model

The library developer performs the following tasks:

- Validates the model for VITAL conformance
- Verifies the model for functionality
- Optimizes the model for performance and capacity
- Re-verifies the model for functionality

The following sections describe each of these tasks in detail.

- [Validating the Model for VITAL Conformance](#)
- [Verifying the Model for Functionality](#)
- [Optimizing the Model for Performance and Capacity](#)
- [Re-Verifying the Model for Functionality](#)
- [Understanding Error and Warning Messages](#)
- [Distributing a VITAL Model](#)

## **Validating the Model for VITAL Conformance**

Library developers can use the `vhdlan` utility to validate the conformance of the VHDL design units to VITAL 95 IEEE specifications, according to level 0 or level 1, as specified in the model.

The `vhdlan` utility checks the VITAL design units for conformance when you set the VITAL attribute on the entity (`VITAL_Level0`) and architecture (`VITAL_Level1`) to TRUE. The `vhdlan` utility does not check the design unit for VITAL conformance if the attribute is set to FALSE.

## **Verifying the Model for Functionality**

After validating the model for VITAL conformance, library developers use the binary executable to verify the model's functions. The functional verification includes checking the following:

- Timing values for the cell, including hazard detection
- Correct operation of the timing constraints and violation detection
- Other behavioral aspects of the cell according to specifications

## **Optimizing the Model for Performance and Capacity**

Library developers use `vhdlan` to analyze the VHDL design units to optimize the model for simulation. The `vhdlan` utility checks the design unit for VITAL conformance before performing any optimization.

To optimize the design units, perform the following steps:

1. Set the VITAL attribute on the entity (`VITAL_Level0`) and on the architecture (`VITAL_Level1`) to TRUE.

When you optimize architectures that have the `VITAL_Level1` attribute set to TRUE, visibility into the cell is lost and the cell is marked as PRIVATE. Ports and generics remain visible.

2. Use either the OPTIMIZE variable in the setup file or the `-optimize` option on the `vhdlan` command line as follows:

- Set the OPTIMIZE variable in the setup file.

[Table 33](#) lists the legal values of the variable, the design unit type, and the results of each setting.

**Table 33** Optimize Variable Values

| Variable | Values | Design Unit Type | Result                                                                                                                             |
|----------|--------|------------------|------------------------------------------------------------------------------------------------------------------------------------|
| OPTIMIZE | TRUE   | Non-VITAL        | The <code>vhdlan</code> utility does not perform any optimization.                                                                 |
| OPTIMIZE | TRUE   | VITAL            | The <code>vhdlan</code> utility performs the optimization on design units that are VITAL conformant.                               |
| OPTIMIZE | FALSE  | Non-VITAL        | The <code>vhdlan</code> utility does not perform any optimization.                                                                 |
| OPTIMIZE | FALSE  | VITAL            | The <code>vhdlan</code> utility does not perform optimization on design unit regardless of its VITAL conformance status (default). |

- Use the `-optimize` option on the `vhdlan` command line. The command-line option overrides the setting in the `synopsys_sim.setup` file.

## Re-Verifying the Model for Functionality

After validating and then optimizing the cell, library developers reverify the results against expected results. The optimizations performed by VCS typically result in correct code.

## Understanding Error and Warning Messages

If the VITAL conformance checks for a design unit fail, VCS issues an error message and stops the optimization of the design unit. Simulation files (`.sim` and `.o` files) are not created, and simulation is not possible for this design unit until the model is changed to conform to VITAL specifications.

If VCS reports a warning message, the optimization stops only if the message is related to the VITAL architecture, otherwise the optimization continues. Simulation files are generated, and you can simulate the design units.

[Table 34](#) lists the status of optimization and simulation file generation based on the type of messages that VCS issues.

**Table 34 Analyzer Status Messages**

| VITAL Attribute        | Message Types | Optimization | Simulation Files |
|------------------------|---------------|--------------|------------------|
| Level 0 (entity)       | error         | stops        | not created      |
| Level 1 (architecture) | error         | stops        | not created      |
| Level 0 (entity)       | warning       | continues    | created          |
| Level 1 (architecture) | warning       | stops        | created          |

For a complete list of conformance checking error messages, see [Gate-Level Simulation on page 639](#) and [Gate-Level Simulation on page 639](#).

When analyzing VITAL models, you can relax VITAL conformance violation errors to a warnings, by setting `RELAX_CONFORMANCE` variable in `synopsys_sim.setup` file to `TRUE`. This value of this variable by default is `FALSE`.

## Distributing a VITAL Model

VITAL library developers (usually, ASIC vendors) can distribute models (ASIC library) to designers in any of the following formats:

- A VHDL source file

After conformance checking and verification, you can distribute the cell library in source format. The library is unprotected, but it is portable.

- An encrypted VHDL source file

You can distribute the encrypted file similar to the VHDL source file. Because the encryption algorithms are generally not public and the code is protected, models are not portable to other simulators.

- Simulation files (the `.sim` and `.o` files)

The cell is analyzed and optimized by the ASIC vendor. The library is protected and is not portable to other simulators or simulator versions.

For the VHDL file and the encrypted VHDL source file formats, the designer can perform the final compilation to optimize the library object codes by using the `-optimize` option. ASIC vendors can provide designers with a script specifying the correct compilation procedure.

## Simulating a VITAL Netlist

A VITAL-based netlist consists of instances of VITAL cells. There are no VITAL specific or other restrictions on the location of such cells in the netlist, nor are there restrictions regarding the quantity or ratio of such cells in relation to other VHDL descriptions.

To simulate a VITAL netlist, simply invoke the binary executable.

## Applying Stimulus

You apply the input stimulus for the VITAL netlist using the same method and format that you use to apply it for any other netlist. For example, you can use WIF, text input/output, or a testbench.

## Overriding Generic Parameter Values

You can override the VITAL generic values in the following ways:

- Using `synopsys_sim.setup` file variables
- Using the elaboration option `-gv generic_name=value`

The following table describes the `SYNOPSYS_SIM.SETUP` variables and the corresponding generic and values allowed:

*Table 35 Timing Constraint and Hazard Flags*

| <b><code>synopsys_sim.setup</code><br/>Variables</b> | <b>Generics</b> | <b>Legal<br/>Values</b> | <b>Result</b>                                                             |
|------------------------------------------------------|-----------------|-------------------------|---------------------------------------------------------------------------|
| Force_TimingChecksOn_TO                              | TimingChecksOn  | TRUE                    | Timing checks are performed.                                              |
|                                                      |                 | FALSE                   | Timing checks are disabled for that cell.                                 |
|                                                      |                 | AsIs                    | User-specified value of the generic is not modified. This is the default. |
| Force_XOn_TO                                         | XOn             | TRUE                    | X's are generated with violations.                                        |
|                                                      |                 | FALSE                   | X generation is disabled for that cell.                                   |
|                                                      |                 | AsIs                    | User-specified value of the generic is not modified. This is the default. |
| Force_MsgOn_TO                                       | MsgOn           | TRUE                    | Messages are reported on violations.                                      |
|                                                      |                 | FALSE                   | Timing messages are disabled for that cell.                               |

**Table 35      Timing Constraint and Hazard Flags (Continued)**

| <b>synopsys_sim.setup Variables</b> | <b>Generics</b> | <b>Legal Values</b> | <b>Result</b>                                                             |
|-------------------------------------|-----------------|---------------------|---------------------------------------------------------------------------|
|                                     |                 | AsIs                | User-specified value of the generic is not modified. This is the default. |

For example:

The following setting in your `synopsys_sim.setup` file performs timing checks:

```
Force_TimingChecksOn_To = TRUE
```

Use the corresponding command line to set the generic:

```
% vcs top -gv TimingChecksOn=TRUE
```

These flags override the value of VITAL generic parameters. The flags have no effect if the model does not use the generic parameter. The generics XOn and MsgOn are parameters to VITAL timing and path delay subprograms.

## Understanding VCS Error Messages

VCS reports two types of errors: system errors and model/netlist errors.

### System Errors

VCS reports a system error if any of the following conditions occur:

- If there are any negative timing values after all timing values are imported and negative constraint calculations (NCC) are performed.

All the adjusted timing values must be positive or zero ( $>=0$ ) after all timing values are imported and NCC is performed. If an adjusted value is negative, NCC issues a warning message and uses zero instead.

Use the `man vss-297` and `man vss-298` command to get more information about NCC error messages.

- If you try to “look-into” the parts of the model that are invisible.

This is because the visibility is limited in VITAL cells that have been optimized and the cells are marked as PRIVATE.

### Model and Netlist Errors

A VITAL model in a VITAL netlist can generate several kinds of errors. The most important are hazard and constraint violations, both of which are associated with a violation of the timing model. The format of such errors is defined by the VITAL standard (in VHDL packages).

## Viewing VITAL Subprograms

You cannot view or access VITAL subprograms. The VITAL packages are built-in. Any reference to a VITAL subprogram (functions or procedures) or any other item in the VITAL packages is converted by VCS to a built-in representation.

## Timing Back-annotation

A VITAL netlist can import timing information from a VHDL configuration or an SDF file.

- A VHDL configuration

VHDL allows the use of a configuration block to override the values of generics specified in the entity declaration. This is done during analysis of the design.

- SDF file

VITAL netlist can import an SDF 3.0 version file. The VITAL standard defines the mapping for SDF 3.0 and the subset supported.

## VCS Naming Styles

VCS automatically determines what naming style is used according to the cell:

- For conformance checked VITAL cells (that is, VITAL entities with the VITAL\_Level0 attribute set to TRUE), VCS uses VITAL naming styles.
- For non-VITAL conformance checked cells, VCS uses the Synopsys naming style (or the style described in SDF naming file).

**Note:**

VCS ignores the SDFNAMINGSTYLE variable in the setup file when determining the naming style.

## Negative Constraints Calculation (NCC)

Adjusting the cell timing values and converting the negative values follows the elaboration and back-annotation phases. VCS follows these steps to prepare the design units for simulation:

1. Design Elaboration

Elaboration is a VHDL step, the design is created and is ready for the simulation run.

2. Back-annotation of timing delay values

Timing values are imported, and the value of generic parameters are updated. VITAL models that support NCC accept back-annotation information as in any other cell.

3. Conversion of the negative constraint values

The value of generic parameters is modified to conform to the NCC algorithm, and negative constraint values are converted to zero or positive.

VCS automatically performs NCC only when the VITAL\_Level0 attribute is set to TRUE for the VITAL entity and the internal clock delay generic (ticd) or internal signal delay generic (tisd) is set.

VCS does not run NCC on design units that have a non-VITAL design type, but you can simulate them.

#### 4. Running the simulation.

## Simulating in Functional Mode

By default, VCS generates code that provides the flexibility of choosing functional or regular VITAL simulation when simulation is run. You can use the *-novitaltiming* runtime option to get functional VITAL simulation; otherwise, you get regular, full-timing VITAL simulation. You can also use *-functional\_vital* with *vhdlan* to get full functional VITAL simulation.

Choosing the VITAL simulation mode at analysis time provides a better performance than choosing the mode at runtime, because it eliminates the runtime check for the functional VITAL simulation mode. The trade-off is that you must reanalyze your VITAL sources if you want to switch between functional and timing simulation. Therefore, you should add the appropriate option to the *vhdlan* command line after you determine which simulation mode gives the best performance while preserving correct simulation results.

Using the *-novitaltiming* runtime option eliminates all timing-related aspects from the simulation of VITAL components. With this option, VCS eliminates the following timing-related aspects: wire delays, path delays, and timing checks, and assigns 0-delay to all outputs. The elimination of timing from the simulation of the VITAL components significantly improves the performance of event simulations.

By specifying *-no\_functional\_vital* for *vhdlan*, you get full timing VITAL simulation without the ability to use functional VITAL at runtime.

However, if your design depends on one or more of the timing-related aspects, you can try reanalyzing the VITAL source files with one or more of the following options, depending on the timing-related or functional capabilities that you need to preserve:

`-keep_vital_ifs`

This option turns off some of the aggressive *novitaltiming* optimizations related to *if* statements in Level 0 VITAL cells.

`-keep_vital_path_delay`

This option preserves the calls to *VitalPathDelay*. Use this switch to preserve correct functionality of non-zero assignments to the outputs.

`-keep_vital_wire_delay`

This option preserves the calls to *VitalWireDelay*. Use this switch to preserve correct functionality of delays on the inputs.

`-keep_vital_signal_delay`

This option preserves the calls to *VitalSignalDelay*. Use this switch to preserve correct functionality of delays on signals.

`-keep_vital_timing_checks`

This option preserves the timing checks within the VITAL cell.

`-keep_vital_primitives`

This option preserves calls to VITAL primitive subprograms.

## Understanding VITAL Timing Delays and Error Messages

This section describes how VCS calculates negative timing constraints during elaboration. This section also lists the error messages that the `vhdlan` utility generates while checking design units for VITAL conformance.

- [Negative Constraint Calculation \(NCC\)](#)
- [Conformance Checks](#)
- [Error Messages](#)

### Negative Constraint Calculation (NCC)

VITAL defines the special generics `ticd`, `tisd`, `tbpd`, `SignalDelay Block`, and equations to adjust the negative setup and hold time and related IOPATH delays.

For VITAL models, NCC adjusts the timing generics for the `ticd` or `tisd` generic. The `ticd` delay is calculated based on SETUP and RECOVERY time. Therefore, NCC resets the original `ticd` delay in VITAL cells.

### Conformance Checks

For VITAL conformance, VCS checks the design units that have the `VITAL_Level0` or `VITAL_Level1` attribute set to TRUE (if the attributes are set to FALSE, VCS issues a warning). The only result of the conformance checking from VCS is the error messages.

VCS performs the following checks:

- Type checking
- Syntactic and semantic checks

## Type Checks

VCS checks and verifies the type for generics, restricted variables, timing constraints, delays, and ports.

VITAL\_Level0 timing generics are checked for type and name. The decoded name can only belong to a finite predefined set { tpd, tsetup, thold, trecovery, ...}.

**Table 36** shows the VITAL delay type names for the generics and the corresponding class for VITAL\_Level0 design units.

**Table 36** Delay Type Name and Corresponding Design Unit Class

| Generic Type Name       | Class                       |
|-------------------------|-----------------------------|
| Time                    | VITAL simple delay type     |
| VitalDelayType          | VITAL simple delay type     |
| VitalDelayArrayType     | VITAL simple delay type     |
| VitalDelayType01        | VITAL transition delay type |
| VitalDelayType01Z       | VITAL transition delay type |
| VitalDelayType01ZX      | VITAL transition delay type |
| VitalDelayArrayType01   | VITAL transition delay type |
| VitalDelayArrayType01Z  | VITAL transition delay type |
| VitalDelayArrayType01ZX | VITAL transition delay type |

VCS checks for the existence of the ports to which the generic refers. For vector subtypes, it checks the index dimensionally.

**Table 37** contains a list of the predefined timing generics. When VCS finds any port names while checking the generic names, it verifies the type of the generic name.

**Table 37** Predefined Timing Generics

| Prefix Name | Ports               | VITAL type        |
|-------------|---------------------|-------------------|
| tpd         | <InPort><OutPort>   | VITAL delay type  |
| tsetup      | <TestPort><RefPort> | simple delay type |
| thold       | <TestPort><RefPort> | simple delay type |
| trecovery   | <TestPort><RefPort> | simple delay type |

*Table 37 Predefined Timing Generics (Continued)*

| Prefix Name | Ports                        | VITAL type        |
|-------------|------------------------------|-------------------|
| tremoval    | <TestPort><RefPort>          | simple delay type |
| tperiod     | <InPort>                     | simple delay type |
| tpw         | <InPort>                     | simple delay type |
| tskew       | <Port1><Port2>               | simple delay type |
| tncsetup    | <TestPort><RefPort>          | simple delay type |
| tnchold     | <TestPort><RefPort>          | simple delay type |
| tipd        | <InPort>                     | VITAL delay type  |
| tdevice     | <InstanceName>[OutPort]      | VITAL delay type  |
| ticd        | <ClockPort>                  | simple delay type |
| tisd        | <InPort><ClockPort>          | simple delay type |
| tbpd        | <InPort><OutPort><ClockPort> | VITAL delay type  |

VITAL\_level0 control generics are only checked for type as shown in [Table 38](#).

*Table 38 Type Checks for Control Generics*

| Name           | Type    |
|----------------|---------|
| InstancePath   | String  |
| TimingChecksOn | Boolean |
| Xon            | Boolean |
| MsgOn          | Boolean |

### Syntactic and Semantic Checks

Before conformance checking, VHDL grammar checks are performed. VITAL is a subset of VHDL, so any further checks are actually semantic checks.

## Error Messages

The error messages are grouped into different classes according to the type of error or the hierarchy of error as shown in [Table 39](#).

*Table 39 Error Message Classes*

| Error Class                                   | Error Prefix |
|-----------------------------------------------|--------------|
| Syntax                                        | VITAL error  |
| Type                                          | VITAL error  |
| Context                                       | VITAL error  |
| Parameter                                     | VITAL error  |
| Illegal Value                                 | VITAL error  |
| Entity Error                                  |              |
| Package                                       |              |
| Usage                                         |              |
| Architecture Level 0                          |              |
| Architecture Level 11.<br>Constraints2. Delay |              |

Error messages have the following features:

- Display the description and location information separately.
- Display an error prefix with entity and architecture, type of error, severity level, file name, line number and the offending line from the source.
- Display only user-helpful information.
- Denote the name of the preceding reference as %s. For example, port %s means that the name of the port should appear at the output.
- Are one-liners for grep/awk retrieval from the log file
- Are numbered as follows: E-VTL001, W-VTL002, ...

[Table 40](#) and [Table 41](#) list all the VITAL error messages. Every message is prefixed with an error class specific message and sufficient context for you to find the problem object. For example, if a port is the offending object, the name of the port and entity are provided. For type violation, the offending type is shown. When there is no indication of what was

found, it means that the negation of the statement was found. For example, the error message “The actual part of ... MUST be static” indicates that the type found is not static.

Table 40 VITAL Error Messages for Level 0 Conformance Issues

| #  | Error Class     | VITAL Reference Manual section number | Error Message                                                                                                      |
|----|-----------------|---------------------------------------|--------------------------------------------------------------------------------------------------------------------|
| 1  | type            | 4.1                                   | The attribute %s { VITAL_Level0, VITAL_Level1 } MUST be declared in package VITAL_Timing and it is declared in %s. |
| 2  | type            | 4.1                                   | The type of the attribute %s { VITAL_Level0, VITAL_Level1 } MUST be Boolean and it is %s.                          |
| 3  | warning         | 4.1                                   | The value of the attribute %s { VITAL_Level0, VITAL_Level1 } MUST be True and it is %s.                            |
| 4  | scope           | 4.2                                   | %s declared in VITAL package %s cannot have an overloaded outside the package.                                     |
| 5  | scope           | 4.2.1                                 | Use of foreign architecture body %s for entity %s is prohibited.                                                   |
| 6  | Not implemented | 4.2.1                                 | The syntactic rule %s, removed in IEEE Std 1076-1993 is illegal in VITAL.                                          |
| 7  | syntax          | 4.3                                   | The only declaration allowed inside an entity's %s declarative part is VITAL_Level0 attribute declaration.         |
| 8  | syntax          | 4.3                                   | No statements allowed inside a VITAL entity's %s statement part.                                                   |
| 9  | semantic        | 4.3.1                                 | Entity %s port %s name CAN NOT contain underscore character(s).                                                    |
| 10 | semantic        | 4.3.1                                 | Entity %s port %s CAN NOT be of mode LINKAGE.                                                                      |
| 11 | semantic        | 4.3.1                                 | Entity %s: The type of the scalar port %s MUST be a subtype of Std_Logic. Type is %s.                              |
| 12 | semantic        | 4.3.1                                 | Entity %s: The type of vector port %s MUST be Std_Logic_Vector. Type is %s.                                        |
| 13 | syntax          | 4.3.1                                 | Entity %s port %s CAN NOT be a guarded signal.                                                                     |
| 14 | semantic        | 4.3.1                                 | Entity %s: a range constraint is not allowed on port %s.                                                           |

**Table 40** VITAL Error Messages for Level 0 Conformance Issues (Continued)

| #  | Error Class | VITAL Reference Manual section number | Error Message                                                                                                                                                                      |
|----|-------------|---------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 15 | semantic    | 4.3.1                                 | Entity %s port %s CAN NOT specify a user defined resolution function.                                                                                                              |
| 16 | warning     | 4.3.2.1.1                             | Entity %s: No port associated with the timing generic %s. Generic %s unused by VITAL and no check will be performed on it.                                                         |
| 17 | type        | 4.3.2.1.2                             | Entity %s: The type of the scalar generic timing parameter %s does not match the type of associated with a vector port %s.                                                         |
| 18 | type        | 4.3.2.1.2                             | Entity %s: the dimension(s) of the vector timing generic %s does not match that of the associated port %s.                                                                         |
| 19 | type        | 4.3.all                               | The type of the timing generic %s MUST be one of { %s, ...} and it is %s.                                                                                                          |
| 20 | semantic    | 4.3.2.1.3.14                          | Biased propagation delay timing generic %s needs a propagation delay timing generic associated with the same port, condition and edge.                                             |
| 21 | semantic    | 4.3.2.1.3.14                          | The type %s of biased propagation delay timing generic %s does not match the type %s of the propagation delay timing generic %s associated with the same port, condition and edge. |
| 22 | semantic    | 4.3.3                                 | The type %s of the control generic %s is illegal. Type MUST be %s.                                                                                                                 |
| 23 | semantic    | 4.4.1                                 | Entity %s: Timing generic %s value used before simulation.                                                                                                                         |
| 24 | semantic    | 4.4                                   | Architecture %s { VITAL_Level0, VITAL_Level1 } %s must be associated with a VITAL_Level0entity.                                                                                    |

Table 41 VITAL Error Messages for Level 1 Conformance Issues

| #  | Error Class | VITAL Reference Manual section number | Error Message                                                                                                                                                   |
|----|-------------|---------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 1  | semantic    | 6.2                                   | VITAL_GLOBSIG, VERR_USER, MARKSignal '%s' MUST be an entity port or an internal signal.                                                                         |
| 2  | semantic    | 6.2                                   | VITAL_GLOBSIG, VERR_USER, MARKSignal-valued attribute '%s' is not allowed in a VITAL Level 1 architecture.                                                      |
| 3  | semantic    | 6.2                                   | It is illegal for a signal %s in architecture %s to have multiple drivers. The drivers are { %s, ... }                                                          |
| 4  | semantic    | 6.2                                   | Internal signal %s of type %s in architecture %s is illegal. Type can be only of type { Std_ULogic, StdLogic_Vector }. Type is %s.                              |
| 5  | semantic    | 6.2                                   | Operators used in a VITAL_Level1 architecture MUST be defined in Std_Logic_1164 . Operator %s is defined in %s.                                                 |
| 6  | semantic    | 6.2                                   | Subprogram invoked in a VITAL_Level1 architecture MUST be defined in Std_Logic_1164 or VITAL package. Subprogram %s is defined in %s.                           |
| 7  | semantic    | 6.2                                   | Formal sub-element association %s in a subprogram call %s is not allowed.                                                                                       |
| 8  | semantic    | 6.2                                   | Type conversion %s in a subprogram call %s is not allowed.                                                                                                      |
| 9  | semantic    | 6.4                                   | Multiple wire delay blocks in architecture %s are not allowed. Offending blocks are labeled { %s, ... }. At most one block with a label "WireDelay" is allowed. |
| 10 | syntax      | 6.4                                   | Architecture %s body is allowed at most one negative constraint block to compute the internal signal delays declared in entity %s.                              |
| 11 | syntax      | 6.4                                   | Architecture %s needs at least one process statement or a concurrent procedure call.                                                                            |
| 12 | semantic    | 6.4.1                                 | Illegal block label %s. It MUST be "WireDelay."                                                                                                                 |
| 13 | context     | 6.4.1                                 | Procedure VitalWireDelay MUST be declared in package VITAL_Timing and it is declared in %s.                                                                     |
| 14 | semantic    | 6.4.1                                 | A call to a VitalWireDelay procedure outside a wire delay block is not allowed.                                                                                 |

**Table 41** VITAL Error Messages for Level 1 Conformance Issues (Continued)

| #  | Error Class | VITAL Reference Manual section number | Error Message                                                                                                                                                                       |
|----|-------------|---------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 15 | semantic    | 6.4.1                                 | At most one wire delay per port of mode IN or INOUT and associated with a wire delay concurrent procedure is allowed inside a wire delay block. Offending signals are {%, ...}.     |
| 16 | semantic    | -                                     | A VITAL predefined name % CAN NOT be overloaded outside the VITAL package %s.                                                                                                       |
| 17 | semantic    | 6.4.1                                 | Internal wire delayed signal %s representing the wire delay of port %s MUST be the same type as the port.                                                                           |
| 18 | semantic    | 6.4.1                                 | The value of port %s can be read only as an actual part to a wire delay concurrent procedure call.                                                                                  |
| 19 | semantic    | 6.4.1                                 | No range attribute specified for generate statement of a wire delay port %s.                                                                                                        |
| 20 | semantic    | 6.4.1                                 | Only a concurrent procedure call allowed inside an array port %s generate statement.                                                                                                |
| 21 | usage       | 6.4.1                                 | The index for the generate statement %s for the array port %s MUST be the name of the generate parameter %s.                                                                        |
| 22 | semantic    | 6.4.1                                 | The actual part associated with the input parameter InSig for a wire delay concurrent procedure call MUST be a name of a port of mode IN or INOUT. Offending port %s is of mode %s. |
| 23 | semantic    | 6.4.1                                 | The actual part associated with the output parameter OutSig for a wire delay concurrent procedure call MUST be a name of an internal signal. The actual part is %s of type %s.      |
| 24 | semantic    | 6.4.1                                 | TWire delay value parameter does not take negative values. Value is %s.                                                                                                             |
| 25 | semantic    | 6.4.1                                 | The actual part associated with wire delay parameter TWire MUST be locally static or a name of an interconnect delay parameter. Actual part is %s.                                  |
| 26 | semantic    | 6.4.2                                 | VITAL negative constraint block MUST have a label named "SignalDelay." Label is %s.                                                                                                 |
| 27 | semantic    | 6.4.2                                 | Negative constraint %s has no procedure call associated with it and therefore is unused by VITAL.                                                                                   |

Table 41 VITAL Error Messages for Level 1 Conformance Issues (Continued)

| #  | Error Class | VITAL Reference Manual section number | Error Message                                                                                                                                                                                                                                                                         |
|----|-------------|---------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 28 | semantic    | 6.4.2                                 | Negative constraint %s has more than one procedure call { %s, ... } associated with it. Only one procedure call per generic timing parameter is allowed.                                                                                                                              |
| 29 | context     | 6.4.2                                 | Procedure VitalSignalDelay MUST be declared in package VITAL_Timing and it is declared in %s.                                                                                                                                                                                         |
| 30 | semantic    | 6.4.2                                 | A call to VitalSignalDelay is not allowed outside a negative constraint block.                                                                                                                                                                                                        |
| 31 | semantic    | 6.4.2                                 | The actual part associated with the delay value parameter Dly in VitalSignalDelay MUST be a timing generic representing internal signal or internal clock delay. The actual part is %s.                                                                                               |
| 32 | semantic    | 6.4.2                                 | The actual part associated with the input signal parameter S in VitalSignalDelay MUST be a static name denoting an input port or the corresponding wire delay signal (if it exists).                                                                                                  |
| 33 | semantic    | 6.4.2                                 | The actual part associated with the output signal parameter DelayedS MUST be an internal signal.                                                                                                                                                                                      |
| 34 | syntax      | 6.4.3                                 | A VITAL process statement %s MUST have sensitivity list.                                                                                                                                                                                                                              |
| 35 | context     | 6.4.3                                 | Signal %s CAN NOT appear in the sensitivity list of process %s.                                                                                                                                                                                                                       |
| 36 | semantic    | 6.4.3.1.1                             | Vital <i>unrestricted</i> variable %s MUST be of type { Std_ulogic, Std_logic_vector, Boolean } only. Type is %s.                                                                                                                                                                     |
| 37 | semantic    | 6.4.3.1.1.1                           | The actual part %s of a <i>restricted</i> formal parameter %s MUST be a simple name.                                                                                                                                                                                                  |
| 38 | semantic    | 6.4.3.1.1.1                           | The initial value of the <i>restricted</i> variable %s associated with the <i>restricted</i> formal parameter GlitchData in procedure VitalPathDelay MUST be a VITAL constant or VITAL function with a locally static parameter, but it is %s.                                        |
| 39 | semantic    | 6.4.3.1.1.1                           | The initial value of the <i>restricted</i> variable %s associated with the <i>restricted</i> formal parameter TimingData in procedure %s { VitalSetupHoldCheck, VitalRecoveryRemovalCheck } MUST be a VITAL constant or VITAL function with a locally static parameter, but it is %s. |

Table 41 VITAL Error Messages for Level 1 Conformance Issues (Continued)

| #  | Error Class | VITAL Reference Manual section number | Error Message                                                                                                                                                                                                                                              |
|----|-------------|---------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 40 | semantic    | 6.4.3.1.1.1                           | The initial value of the <i>restricted</i> variable %s associated with the <i>restricted</i> formal parameter PeriodPulseData in procedure VitalPeriodPulseCheck MUST be a VITAL constant or VITAL function with a locally static parameter, but it is %s. |
| 41 | semantic    | 6.4.3.1.1.1                           | The initial value of the <i>restricted</i> variable %s associated with the <i>restricted</i> formal parameter PreviousDataIn in procedure VitalStateTable can be only a VITAL constant or a VITAL function with a locally static parameter, but it is %s.  |
| 42 | syntax      | 6.4.3.2                               | A VITAL process statement cannot be empty.                                                                                                                                                                                                                 |
| 43 | syntax      | 6.4.3.2.1                             | The condition in timing check IF statement MUST be the simple name TimingCheckOn defined in entity %s as a control generic.                                                                                                                                |
| 44 | semantic    | 6.4.3.2.1                             | A VITAL timing check statement can be only a call to one of { VITAL_Timing, VITALSetupHoldCheck, VITALRecoveryRemovalCheck, VITALPeriodPulseCheck }.                                                                                                       |
| 45 | semantic    | 6.4.3.2.1                             | The procedure %s { VITAL_Timing, VITALSetupHoldCheck, VITALRecoveryRemovalCheck, VITALPeriodPulseCheck } MUST be declared in package VITAL_Timing, but it is declared in %s.                                                                               |
| 46 | semantic    | 6.4.3.2.1                             | A call to %s ( One of { VITAL_Timing(), VITALSetupHoldCheck(), VITALRecoveryRemovalCheck(), VITALPeriodPulseCheck() } ) occurred outside a timing check section.                                                                                           |
| 47 | semantic    | 6.4.3.2.1                             | The actual part %s associated with the formal parameter %s (representing a signal name %s) MUST be locally static.                                                                                                                                         |
| 48 | semantic    | 6.4.3.2.1                             | The actual %s associated with the formal parameter HeaderMsg MUST be a globally static expression.                                                                                                                                                         |
| 49 | semantic    | 6.4.3.2.1                             | The actual %s of the timing check procedure %s associated with a formal parameter %s of type Time MUST be a locally static expression or simple name denoting the control generic of the same name.                                                        |

Table 41 VITAL Error Messages for Level 1 Conformance Issues (Continued)

| #  | Error Class | VITAL Reference Manual section number | Error Message                                                                                                                                                                                                                                                                                                                                                                                                                 |
|----|-------------|---------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 50 | semantic    | 6.4.3.2.1                             | The actual %s associated with a formal parameter %s { XOn, MsgOn } MUST be a globally static expression.                                                                                                                                                                                                                                                                                                                      |
| 51 | semantic    | 6.4.3.2.1                             | A function %s call or an operator %s invocation in the actual part to a formal parameter %s MUST be a function/operator defined in one of packages { Standard, Std_logic_1164, VITAL_Timing }.                                                                                                                                                                                                                                |
| 52 | semantic    | 6.4.3.2.1                             | The actual %s associated with the formal parameter %s { TestSignalName } MUST be locally static expression.                                                                                                                                                                                                                                                                                                                   |
| 53 | context     | 6.4.3.2.1                             | variable %s associated with a timing check violation parameter %s could not be used in another timing check statement. It appears in timing check %s.                                                                                                                                                                                                                                                                         |
| 54 | context     | 6.4.3.2.2                             | procedure VitalStateTable() MUST be declared in the package VITAL_Primitives, but it is declared in %s.                                                                                                                                                                                                                                                                                                                       |
| 55 | semantic    | 6.4.3.2.2                             | Only a call to the predefined procedure VitalStateTable() is allowed inside a VITAL functionality section.                                                                                                                                                                                                                                                                                                                    |
| 56 | semantic    | 6.4.3.2.2                             | The actual %s associated with the StateTable parameter to procedure VitalStateTable MUST be globally static expression.                                                                                                                                                                                                                                                                                                       |
| 57 | semantic    | 6.4.3.2.2                             | The index constraint on the variable %s associated with the PreviousDataIn parameter MUST match the constraint on the actual associated with the DataIn parameter.                                                                                                                                                                                                                                                            |
| 58 | semantic    | 6.4.3.2.2                             | The target of a VITAL variable assignment MUST be <i>unrestricted</i> variable denoted by a locally static name, but it is %s.                                                                                                                                                                                                                                                                                                |
| 59 | type        | 6.4.3.2.2                             | The target of an assignment statement of a standard logic type inside a functionality section requires a primary on the right side to be one of the following:1. A globally static expression2. A name of a port or an internal signal3. A function call to a standard logic function, a VITAL primitive or VITALTruthTable()4. An aggregate or a qualified expression with an aggregate operand5. A parenthesized expression |
| 60 | semantic    | 6.4.3.2.2                             | A call to function VITALTruthTable CAN NOT occur outside VITAL functionality section.                                                                                                                                                                                                                                                                                                                                         |

**Table 41** VITAL Error Messages for Level 1 Conformance Issues (Continued)

| #  | Error Class | VITAL Reference Manual section number | Error Message                                                                                                                                                                                                                   |
|----|-------------|---------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 61 | semantic    | 6.4.3.2.3                             | The procedure %s { VITALPathDelay, VITALPathDelay01, VITALPathDelay01Z } MUST be defined in package VITAL_Timing, but it is defined in %s.                                                                                      |
| 62 | semantic    | 6.4.3.2.3                             | A call to procedure %s { VITALPathDelay, VITALPathDelay01, VITALPathDelay01Z } CAN NOT occur outside a path delay section.                                                                                                      |
| 63 | semantic    | 6.4.3.2.3                             | The actual part associated with the formal parameter OutSignal of a path delay procedure %s { VITALPathDelay, VITALPathDelay01, VITALPathDelay01Z } MUST be a locally static signal name, but it is %s.                         |
| 64 | semantic    | 6.4.3.2.3                             | The actual part associated with the formal parameter Paths of a path delay procedure %s { VITALPathDelay, VITALPathDelay01, VITALPathDelay01Z } MUST be an aggregate, but it is %s.                                             |
| 65 | semantic    | 6.4.3.2.3                             | The sub-element PathDelay of the actual part associated with the formal parameter Paths to a path delay procedure %s { VITALPathDelay, VITALPathDelay01, VITALPathDelay01Z } MUST be globally static, but it is %s.             |
| 66 | semantic    | 6.4.3.2.3                             | The sub-element InputChangeTime of the actual associated with the formal parameter Paths %s { VITALPathDelay, VITALPathDelay01, VITALPathDelay01Z } MUST be a LastEvent attribute or a locally static expression, but it is %s. |
| 67 | semantic    | 6.4.3.2.3                             | The actual associated with the formal parameter GlitchMode to a path delay procedure %s MUST be a literal, but it is %s.                                                                                                        |
| 68 | semantic    | 6.4.3.2.3                             | The actual part associated with the formal parameter GlitchData MUST be a locally static name, but it is %s.                                                                                                                    |
| 69 | semantic    | 6.4.3.2.3                             | The actual part associated with the formal parameter %s { Xon, MsgOn } MUST be a locally static expression or a simple name denoting control generic of the same name, but it is %s.                                            |
| 70 | semantic    | 6.4.3.2.3                             | The actual part associated with the formal parameter %s { OutSignalName, DefaultDelay, OutputMap } MUST be a locally static expression.                                                                                         |

**Table 41** VITAL Error Messages for Level 1 Conformance Issues (Continued)

| #  | Error Class | VITAL Reference Manual section number | Error Message                                                                                                                                                                                                                      |
|----|-------------|---------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 71 | No Check    | 6.4.3.2.3                             | Port of type %s { OUT, INOUT, BUFFER } has to be driven by a VITAL primitive procedure call or a path delay procedure, but the driver is %s.                                                                                       |
| 72 | semantic    | 6.4.4                                 | The actual associated with the formal parameter %s of class VARIABLE or SIGNAL on VITAL primitive %s MUST be a static name, but it is %s.                                                                                          |
| 73 | semantic    | 6.4.4                                 | The actual part associated with the formal parameter %s of class CONSTANT to a procedure call %s MUST be a locally static expression, but it is %s.                                                                                |
| 74 | semantic    | 6.4.4                                 | The actual part associated with the formal parameter ResultMap to a procedure call %s MUST be a locally static expression, but it is %s.                                                                                           |
| 75 | semantic    | 6.4.4                                 | The actual part associated with the formal parameter %s { TruthTable, StateTable } on table primitive procedure call %s MUST be a constant whose value expression is an aggregate with fields that are locally static expressions. |
| 76 | No Check    | 7.1.1                                 | VITAL logic primitive %s MUST be defined in package %s.                                                                                                                                                                            |
| 77 | No Check    | 7.3.1                                 | Symbol %s CAN NOT appear in Table %s.                                                                                                                                                                                              |
| 78 | No Check    | 7.3.3.1                               | Wrong number of inputs to an object %s of type VitalTruthTable. The number MUST equal to the value of the DataIn parameter VitalTruthTable.                                                                                        |
| 79 | No Check    | 7.3.3.1                               | Wrong dimensions for table %s of type %s { VitalTruthTable, VitalStateTable }.                                                                                                                                                     |
| 80 | Package     | 7.4.3.2.2                             | procedure VitalStateTable() MUST be declared in VitalPrimitives, but it is declared in %s.                                                                                                                                         |
| 81 | Package     | 7.4.3.2.3                             | procedure %s { VITALPathDelay, VITALPathDelay01, VITALPathDelay01Z } MUST be defined in package VITAL_Timing, but it is defined in %s.                                                                                             |

## Additional Diagnostics

VCS provides the following additional diagnostics under the `-diag=sdf` sub-option.

- [Support for Identifying Non-Annotated Timing Arc and Timing Check Statements](#)

### Support for Identifying Non-Annotated Timing Arc and Timing Check Statements

VCS provides additional diagnostics under the `-diag=sdf` sub-option that lists down all the non-annotated timing arcs and paths, mentioned in the specify block of cell library by SDF file.

The syntax is as follows:

```
%vcs -diag=sdf:verbose <.....>
```

### Usage Example

The following example illustrates the usage of timing check:

```
`timescale 1ns/1ns
module test();
wire q;
reg d, clk;
mydff u(q,clk,d);
mydff u1(q,clk,d);

initial begin
 $sdf_annotate("example.sdf",,,);
 $monitor("%d",$time," q=%b d=%b clk=%b",q,d,clk);
 d=0;
 clk=0;
#20 d = 1;
#40 d = 1'bx;
#200 $finish;
end
always
#20 clk = ~clk;

endmodule

(DELAYFILE
 (SDFVERSION "OVI 2.1")
 (DESIGN "test")
 (DIVIDER /)
 (TIMESCALE 1ns)
 (CELL
 (CELLTYPE "test"))
```

```
(INSTANCE)
(DELAY
(ABSOLUTE
(IOPATH test/u/din test/u/qout (5) (3))
(IOPATH test/u/clk test/u/qout (5) (3))
)
)

(TIMINGCHECK
(SETUPHOLD test/u1/din (posedge test/u1/clk) (5:10:15) (5:10:15))
)
)
```

To run the example, use the following commands:

```
% vcs -diag=sdf:verbose example.v
% simv
% cat sdfAnnotateInfo
```

The following is the output:

```
Static entries in elaborated design under "test":
Annotated by SDF "example.sdf":
```

```
No. of Pathdelays = 4 Annotated = 50.00%
No. of Tchecks = 4 Annotated = 50.00%
```

| Percentage | Total | Annotated |
|------------|-------|-----------|
| IOPATH     | 4     | 2         |
| 50.00%     |       |           |

Path Delays Summary of above

|           |   |   |
|-----------|---|---|
| SETUPHOLD | 4 | 2 |
| 50.00%    |   |   |

Timing checks Summary of above

OverAll Static entries in elaborated design:
No. of Pathdelays = 4 Annotated = 50.00%
No. of Tchecks = 4 Annotated = 50.00%

| Percentage | Total | Annotated |
|------------|-------|-----------|
| IOPATH     | 4     | 2         |
| 50.00%     |       |           |

Path Delays Summary of above

|           |   |   |
|-----------|---|---|
| SETUPHOLD | 4 | 2 |
| 50.00%    |   |   |

Timing checks Summary of above

OverAll Static entries in elaborated design:  
 No. of Pathdelays = 4 Annotated = 50.00%  
 No. of Tchecks = 4 Annotated = 50.00%

| Percentage | Total | Annotated |
|------------|-------|-----------|
| IOPATH     | 4     | 2         |
| 50.00%     |       |           |

Path Delays Summary of above

|           |   |   |
|-----------|---|---|
| SETUPHOLD | 4 | 2 |
| 50.00%    |   |   |

Timing checks Summary of above

# \*\*\*\*\* Design Not Annotated \*\*\*\*\* #

```
Module Name: mydff, Instance Name: test.u
$setuphold(posedge clk, din, 3, 3);
```

```
Module Name: mydff, Instance Name: test.u1
(din => qout) = (4);
(clk => qout) = (4);
```

In this example there are two instances of mydff, which are u, u1 respectively.

In SDF file, IOPATH is specified only for instance u and not for u1.

```
(IOPATH test/u/din test/u/qout (5)(3))
(IOPATH test/u/clk test/u/qout (5)(3))
```

Hence in sdfAnnotateInfo file, you can see that IOPATH is 50% annotated.

No. of Pathdelays = 4 Annotated = 50.00%

No. of Tchecks = 4 Annotated = 50.00%

Under the section, "\*\*\*\* Design Not Annotated \*\*\*", you can find more details.

```
Module Name: mydff, Instance Name: test.u1
(din => qout) = (4);
(clk => qout) = (4);
```

**Note:**

If the minimum value is not mentioned in SDF and when VCS takes a default minimum value, then there are entries `sdfAnnotateInfo` file in the case of typical delay.

# 14

## Coverage

---

VCS monitors the execution of the HDL code during simulation. Verification engineers can determine which part of the code has not been tested yet so that they can focus their efforts on those areas to achieve 100% coverage. VCS offers two coverage techniques to test your HDL code: Code coverage and Functional coverage.

This chapter consists of the following sections:

- [Code Coverage](#)
  - [Functional Coverage](#)
  - [Options For Coverage Metrics](#)
- 

### Code Coverage

The following coverage metrics are classified as code coverage:

- Line Coverage — This metric measures statements in your HDL code that have been executed in the simulation.
- Toggle Coverage — This metric measures the bits of logic that have toggled during simulation. A toggle simply means that a bit changes from 0 to 1 or from 1 to 0. It is one of the oldest metrics of coverage in hardware designs and can be used at both the register transfer level (RTL) and gate level.
- Condition Coverage — This metric measures how the variables or sub-expressions in the conditional statements are evaluated during simulation. It can find errors in the conditional statements that cannot be found by other coverage analysis.
- Branch Coverage — This metric measures the coverage of expressions and case statements that affect the control flow (such as if-statement and while-statement) of the HDL. It focuses on the decision points that affect the control flow of the HDL execution.
- FSM Coverage — This metric verifies that every legal state of the state machine has been visited and that every transition between states has been covered.

For more information about coverage technology and how you can generate the coverage information for your design, see the *Coverage Technology User Guide* in the VCS Online Documentation.

---

## Functional Coverage

Functional coverage checks the overall functionality of the implementation. To perform functional coverage, you must define coverage points for the functions to be covered in the DUT. VCS supports both Native Testbench (NTB) and SystemVerilog covergroup models. Covergroups are specified by users. They allow the system to monitor values and transitions for variables and signals. They also enable cross coverage between variables and signals.

For more information about NTB or SystemVerilog functional coverage models, see the Testbench category in the VCS Online Documentation and see Chapter 19, “Functional Coverage” in the *SystemVerilog LRM IEEE Std. 1800 - 2012* respectively.

---

## Options For Coverage Metrics

`-cm line|cond|fsm|tgl|branch|assert`

Specifies elaborating for the specified type or types of coverage. The argument specifies the types of coverage:

`line`

Elaborate for line or statement coverage.

`cond`

Elaborate for condition coverage.

`fsm`

Elaborate for FSM coverage.

`tgl`

Elaborate for toggle coverage.

`branch`

Elaborate for branch coverage

`assert`

Elaborate for SystemVerilog assertion coverage.

For more information on Coverage options, see the *Coverage Technology User Guide* in the *VCS Online Documentation*.

# 15

## Using OpenVera Native Testbench

---

OpenVera Native Testbench is a high-performance, single-kernel technology in VCS that enables:

- Native compilation of testbenches written in OpenVera and in SystemVerilog.
- Simulation of these testbenches along with the designs.

This technology provides a unified design and verification environment in VCS for significantly improving overall design and verification productivity. Native Testbench is uniquely geared towards efficiently catching hard-to-find bugs early in the design cycle, enabling not only completing functional validation of designs with the desired degree of confidence, but also achieving this goal in the shortest time possible.

Native Testbench is built around the preferred methodology of keeping the testbench and its development separate from the design. This approach facilitates development, debug, maintenance and reusability of the testbench, as well as ensuring a smooth synthesis flow for your design by keeping it clean of all testbench code. Further, you have the choice of either compiling your testbench along with your design or separate from it. The latter choice not only saves you from unnecessary recompilations of your design, it also enables you to develop and maintain multiple testbenches for your design.

This chapter describes the high-level, object-oriented verification language of OpenVera, which enables you to write your testbench in a straightforward, elegant and clear manner and at a high level essential for a better understanding of and control over the design validation process. Further, OpenVera assimilates and extends the best features found in C++ and Java along with syntax that is a natural extension of the hardware description languages (Verilog and VHDL). Adopting and using OpenVera, therefore, means a disciplined and systematic testbench structure that is easy to develop, debug, understand, maintain and reuse.

Thus, the high-performance of Native Testbench technology, together with the unique combination of the features and strengths of OpenVera, can yield a dramatic improvement in your productivity, especially when your designs become very large and complex.

This chapter includes the following topics:

- [Usage Model](#)
- [Key Features](#)

## Usage Model

As any other VCS applications, the usage model to simulate OpenVera testbench includes the following steps:

For two-step flow:

### Compilation

```
% vcs [ntb_options] [compile_options] file1.vr file2.vr
file3.v file4.v
```

### Simulation

```
% simv [run_options]
```

For three-step flow:

### Analysis

Always analyze Verilog before VHDL.

```
% vlogan -ntb [vlogan_options] file1.vr file2.vr file3.v
% vhdlan [vhdlan_options] file3.vhd file4.vhd
```

### Note:

Specify the VHDL bottommost entity first, and then move up in order.

### Elaboration

```
% vcs [other_ntb_options] [compile_options] design_unit
```

### Simulation

```
% simv [run_options]
```

## Example

In this example, you have an interface file, a Verilog design `arb.v`, OpenVera testbench `arb.vr`, all instantiated in a Verilog top file, `arb.test_top.v`.

```
//Interface
#ifndef INC_ARB_IF_VRH
#define INC_ARB_IF_VRH

interface arb {
 input clk CLOCK;
 output [1:0] request OUTPUT_EDGE OUTPUT_SKew;
 output reset OUTPUT_EDGE OUTPUT_SKew;
 input [1:0] grant INPUT_EDGE INPUT_SKew;
```

```

 } // end of interface arb

#endif

//Verilog module: arb.v
module arb (clk, reset, request, grant) ;
 input [1:0] request ;
 output [1:0] grant ;
 input reset ;
 input clk ;

parameter IDLE = 2, GRANT0 = 0, GRANT1 = 1;

reg last_winner ;
reg winner ;
reg [1:0] grant ;
reg [1:0] next_grant ;

reg [1:0] state, nxState;

...
endmodule

//OpenVera Testbench: arb.vr

#define OUTPUT_EDGE PHOLD
#define OUTPUT_SKEW #1
#define INPUT_SKEW #-1
#define INPUT_EDGE PSAMPLE
#include <veraDefines.vrh>

#include "arb.if.vrh"

program arb_test
{ // start of top block

...
} // end of program arb_test

```

**Note:**

You can find the complete example in the following path:

\$VCS\_HOME/doc/examples/testbench/ov/Tutorial/arb

In this example, we have an interface file, a Verilog design, `design.v` instantiated in a VHDL top.vhd. Testbench is in OpenVera.

```
//Interface: verilog_mod.if.vrh
interface verilog_mod {
```

```

 input clk CLOCK ;
 output din_single PHOLD #1 ;
 output [7:0] din_vector PHOLD #1 ;
 input dout_wire_single PSAMPLE #-1;
 input idout_wire_single PSAMPLE #-1 hdl_node
"/top/dout_wire_single" ;
 input [7:0] dout_wire_vector PSAMPLE #-1;
 input [7:0] idout_wire_vector PSAMPLE #-1 hdl_node
"/top/dout_wire_vector" ;
 input dout_reg_single PSAMPLE #-1;
 input idout_reg_single PSAMPLE #-1 hdl_node "/top/dout_reg_single" ;
 input [7:0] dout_reg_vector PSAMPLE #-1;
 input [7:0] idout_reg_vector PSAMPLE #-1 hdl_node
"/top/dout_reg_vector" ;
} // end of interface verilog_mod

//Verilog module: design.v

module verilog_mod1 (
clk,din_single,din_vector,dout_wire_single,dout_reg_single,dout_wire_vector,dout_reg_vector
) ;
input clk;
input din_single;
input [7:0] din_vector;
output dout_wire_single ;
output dout_reg_single ;
output dout_wire_vector ;
output [7:0] dout_reg_vector ;
...
endmodule

-- VHDL Top: top.vhd
...
entity top is
 generic (
 EMU : boolean := false);
end top;

architecture vhdl_top of top is

component verilog_mod1
 port (
 clk : IN std_logic ;
 din_single : IN std_logic ;
 din_vector : IN std_logic_vector(7 downto 0) ;
 dout_wire_single : OUT std_logic ;
 dout_wire_vector : OUT std_logic_vector(7 downto 0) ;
 dout_reg_single : OUT std_logic ;
 dout_reg_vector : OUT std_logic_vector(7 downto 0)
);
end component;
...

```

```

begin -- ntbmx_test
 ...
 vshell: test
 port map (SystemClock => SystemClock,
 \verilog_mod.clk\ => clk,
 \verilog_mod.din_single\ => din_single,
 ...
);
 ...
end vhdl_top;

//OpenVera Testbench: test.vr

#include <veraDefines.vrh>
#define MAX_COUNT 10
#include "interface.if"
...
program test {
 integer i ;
 bit b ;
 integer n ;
 force_it_p fp ;
 ...
}

```

**Note:**

You can find the complete example in \$VCS\_HOME/doc/examples/nativetestbench/mixedhdl/testcase\_2

## Usage Model

For two-step flow:

### Compilation

```
% vcs -ntb arb.v arb.vr arb.test_top.v
```

### Simulation

```
% simv
```

For three-step flow

## Analysis

```
% vlogan -ntb test.vr design.v
% vhdlan top.vhd
```

### Note:

Specify the VHDL bottom-most entity first, and then move up in order.

### Elaboration

```
% vcs top
```

### Simulation

```
% simv
```

## Importing VHDL Procedures

VHDL procedures can be imported into the NTB domain using the `hdl_task` statement:

```
hdl_task OpenVera_name ([parameters])
 "vhdl_task [lib].[package].[VHDL_name]"
```

The only difference to the OpenVera `hdl_task` syntax is that NTB requires the `vhdl_task` keyword. This keyword is required because NTB must be able to distinguish between Verilog and VHDL procedures at analysis time (`vlogan`). The `[lib]`, `[package]` and `[VHDL_name]` entries must point to the VHDL library and package where the `[VHDL_name]` procedure are described. The VHDL procedures are best described in packages so that they can be accessed globally.

The parameters of the VHDL procedure can be of `in`, `out` or `inout` type and are mapped between the OpenVera and VHDL type by use of the global `-ntb_opts sigtype=[type]` command-line option to `vlogan`:

*Table 42 Mapping OpenVera and VHDL Datatypes*

| OpenVera data type | VHDL data type    | sigtype    |
|--------------------|-------------------|------------|
| bit                | STD_LOGIC         | STD_LOGIC  |
| bit[N-1:0]         | STD_LOGIC_VECTOR  |            |
| bit                | STD_ULOGIC        | STD_ULOGIC |
| bit[N-1:0]         | STD_ULOGIC_VECTOR | (default)  |
| bit                | BIT               | BIT        |
| bit[N-1:0]         | BIT_VECTOR        |            |
| bit[N-1:0]         | SIGNED            | SIGNED     |

**Table 42** *Mapping OpenVera and VHDL Datatypes (Continued)*

| OpenVera data type | VHDL data type | sigtype  |
|--------------------|----------------|----------|
| bit[N-1:0]         | UNSIGNED       | UNSIGNED |
| bit[N-1:0]         | INTEGER        | INTEGER  |
| bit                | BOOLEAN        | BOOLEAN  |
| integer            | INTEGER        | any      |

Note that this flow is limited to one global signal type, so all parameters of all imported and exported type must be the same base `ntb_sigtype`, for example, `STD_LOGIC` and `STD_LOGIC_VECTOR`.

If two or more concurrent calls to an imported procedure can occur, the later one is queued and executed when the procedure is free again. Although this matches OpenVera behavior, the timing shift is probably not what you intended. The solution to this problem is the `-ntb_opts task_import_poolsize=[size]` option to `vlogan`. Here you can define the maximum number of imported tasks or procedures that can be called in parallel without blocking.

## Exporting OpenVera Tasks

OpenVera tasks can be exported into the VHDL and Verilog domains using the `export` keyword in the task definition.

For using the function in VHDL, `vlogan` creates a VHDL wrapper package named `[OpenVera program name]_pkg`. This package is automatically compiled into the `WORK` library. The VHDL part of the design can thus call the OpenVera task in any process that has no sensitivity list. As a prerequisite, the calling entity only needs to include the corresponding “use” statement:

```
use work.[OpenVera program name]_pkg.all;
```

The mapping of the OpenVera and VHDL data types is defined by the `-ntb_opts sigtype=[type]` command-line option as described earlier. The `-ntb_opts task_export_poolsize` command-line option can be used to increase the maximum number of concurrent calls to exported tasks. Note, however that in contrast to the imported tasks, exceeding this limit can cause a runtime error of the simulation.

Example:

```
---- start OpenVera code fragment ----
export task vera_decrement (var bit[31:0] count)
{
 count = count - 1;
}
```

```
program my_testbench
{ ...
 ---- end OpenVera code fragment ----

task automatic vera_decrement (inout reg [31:0] count) ...
```

The corresponding VHDL procedure named `vera_decrement` is created in `my_testbench_pkg` package and analyzed into the `WORK` library.

## Using Template Generator

To ease the process of writing a testbench in OpenVera, VCS provides you with a testbench template generator. The template generator supports both a Verilog and a VHDL top design.

Use the following command to invoke the template generator on a Verilog or VHDL design unit:

```
% ntb_template -t design_module_name [-c clock] design_file\
[-vcs vcs_compile-time_options]
```

Where:

**-t *design\_module\_name***

Specifies the top-level design module name.

***design\_file***

Name of the design file.

**-c**

Specifies the clock input of the design. Use this option only if the specified *design\_file* is a Verilog file.

**-template**

Can be omitted.

**-program**

Optional. Use it to specify program name.

**-simcycle**

Optional. Use this to override the default cycle value of 100.

**-vcs *vcs\_compile-time\_options***

Optional. Use it to supply a VCS compile-time option. Multiple `-vcs vcs_compile-time_options` options can be used to specify multiple options. Use this option only for Verilog on top designs.

## Example

An example SRAM model is used in this demonstration of using the template generator to develop a testbench environment.

For details on the OpenVera verification language, refer to the *OpenVera Language Reference Manual: Native Testbench*.

### Design Description

The design is an SRAM whose RTL Verilog model is in the file `sram.v`. It has four ports:

- `ce_N` (chip enable)
- `rdWr_N` (read/write enable)
- `ramAddr` (address)
- `ramData` (data)

#### *Example 71*

During a read operation, when `ce_N` is driven low and `rdWr_N` is driven high, `ramData` is continuously driven from inside the SRAM with the value stored in the SRAM memory element specified by `ramAddr`. During a write operation, when both `ce_N` and `rdWr_N` are driven low, the value driven on `ramData` from outside the SRAM is stored in the SRAM memory element specified by `ramAddr`. At all other times, `ce_N` is driven high, and as a result, `ramData` gets continuously driven from inside the SRAM with the high-impedance value `Z`.

## Generating the Testbench Template, the Interface, and the Top-level Verilog Module from the Design

As previously mentioned, Native Testbench provides a template generator to start the process of constructing a testbench. The template generator is invoked on `sram.v` as shown below:

```
% ntb_template -t sram sram.v
```

Where:

- The `-t` option is followed with the top-level design module name, which is `sram`, in this case.
- `sram` is the name of the module.
- `sram.v` is the name of the file containing the top-level design module.
- If the design uses a clock input, then the `-c` option is to be used and followed with the name of the clock input. Doing so provides a clock input derived from the system-clock for the interface and the design. In this example, there is no clock input required by the design.

Template generator generates the following files:

- `sram.vr.tmp`
- `sram.if.vrh`
- `sram.test_top.v`

### **sram.vr.tmp**

This is the template for testbench development. The following is an example, based on the `sram.v` file of the output of the previous command line:

```
//sram.vr.tmp
#define OUTPUT_EDGE PHOLD
#define OUTPUT_SKEW #1
#define INPUT_SKEW #-1
#define INPUT_EDGE PSAMPLE
#include <veraDefines.vrh>

// define interfaces, and verilog_node here if necessary
#include "sram.if.vrh"

// define ports, binds here if necessary

// declare external tasks/classes/functions here if
//necessary
```

```
// declare verilog_tasks here if necessary
// declare class typedefs here if necessary

program sram_test
{ // start of top block

 // define global variables here if necessary

 // Start of sram_test

 // Type your test program here:

 //
 // Example of drive:
 // @1 sram.ce_N = 0 ;
 //
 //
 // Example of expect:
 // @1,100 sram.example_output == 0 ;
 //

} // end of program sram_test

// define tasks/classes/functions here if necessary
```

### **sram.if.vrh**

This is the interface file which provides the basic connectivity between your testbench signals and your design's ports and/or internal nodes. All signals going back and forth between the testbench and the design go through this interface. The following is the sram.if.vrh file which results from the previous command line:

```
//sram.if.vrh
#ifndef INC_SRAM_IF_VRH
#define INC_SRAM_IF_VRH
interface sram {
 output ce_N OUTPUT_EDGE OUTPUT_SKew;
 output rdWr_N OUTPUT_EDGE OUTPUT_SKew;
 output [5:0] ramAddr OUTPUT_EDGE OUTPUT_SKew;
 inout [7:0] ramData INPUT_EDGE INPUT_SKew OUTPUT_EDGE
OUTPUT_SKew;
} // end of interface sram

#endif
```

Notice that, for example, the direction of ce\_N is now "output" instead of "input". The signal direction specified in the interface is from the point of view of the testbench and not the DUT.

This file must be modified to include the clock input.

### **sram.test\_top.v**

This is the top-level Verilog module that contains the testbench instance, the design instance, and the system-clock. The system clock can also provide the clock input for both the interface and the design. The following is the `sram.test_top.v` file that results from the previous command line:

```
//sram.test_top.v
module sram_test_top;
 parameter simulation_cycle = 100;

 reg SystemClock;

 wire ce_N;
 wire rdWr_N;
 wire [5:0] ramAddr;
 wire [7:0] ramData;

`ifdef SYNOPSYS_NTB
 sram_test vshell(
 .SystemClock (SystemClock),
 .\sram.ce_N (ce_N),
 .\sram.rdwR_N (rdWr_N),
 .\sram.ramAddr (ramAddr),
 .\sram.ramData (ramData)
);
`else

 vera_shell vshell(
 .SystemClock (SystemClock),
 .sram_ce_N (ce_N),
 .sram_rdwR_N (rdWr_N),
 .sram_ramAddr (ramAddr),
 .sram_ramData (ramData)
);
`endif

`ifdef emu
/* DUT is in emulator, so not instantiated here */
`else
 sram dut(
 .ce_N (ce_N),
 .rdWr_N (rdWr_N),
 .ramAddr (ramAddr),
 .ramData (ramData)
);
`endif

 initial begin
 SystemClock = 0;
 forever begin
 #(simulation_cycle/2)
 SystemClock = ~SystemClock;
 end
 end
endmodule
```

```

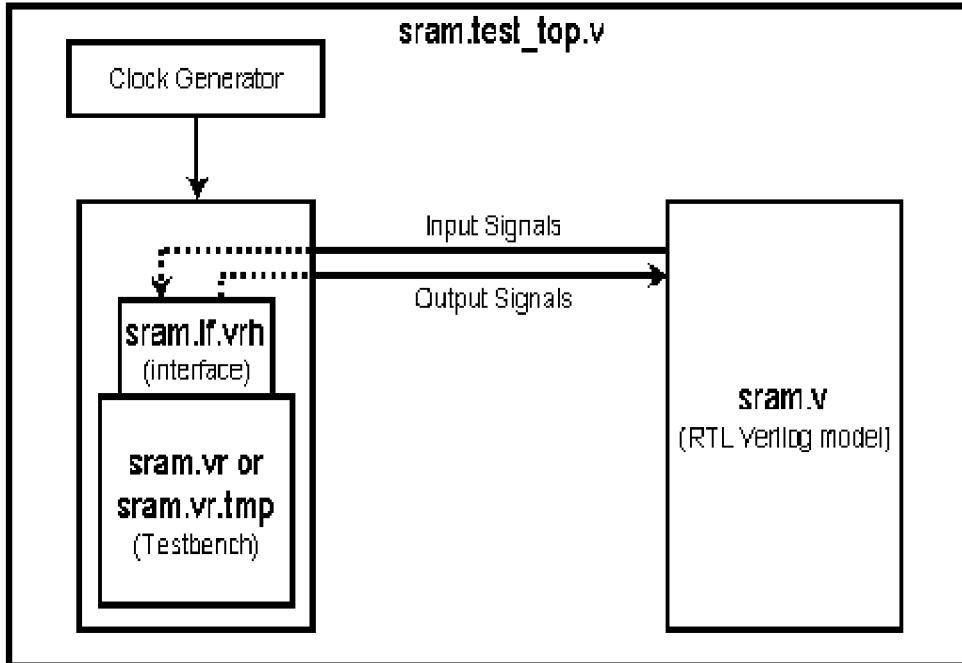
 end
 end

endmodule

```

[Figure 160](#) shows how the three generated files and the design connect and fit in with each other in the final configuration.

*Figure 160 Testbench and Design Configuration*



### Testbench Development and Description

Your generated testbench template, `sram.vr.tmp`, contains a list of macro definitions for the interface, include statements for the interface file and the library containing predefined tasks and functions, comments indicating where to define or declare the various parts of the testbench, and the skeleton program shell that will contain the main testbench constructs. Starting with this template, you can develop a testbench for the SRAM and rename it `sram.vr`. An example testbench is shown in [Example 72](#).

#### *Example 72 Example testbench for SRAM, sram.vr*

```

// macro definitions for Interface signal types and skews
#define OUTPUT_EDGE PHOLD // for specifying posedge-drive type
#define OUTPUT_SKEW #1 // for specifying drive skew
#define INPUT_SKEW #-1 // for specifying sample skew
#define INPUT_EDGE PSAMPLE // for specifying posedge-sample type

#include <vera_defines.vrh> // include the library of predefined
 // functions and tasks

```

```
#include "sram.if.vrh" // include the Interface file

program sram_test { // start of program sram_test

 reg [5:0] address = 6'b00_0001; // declare, initialize address (for
 // driving ramAddr during Write and
 // Read)
 reg [7:0] rand_bits; // declare rand_bits (for driving
 // ramData during Write)
 reg [7:0] data_result; // declare data_result (for receiving
 // ramData during Read)

 @ (posedge sram.clk);
 rand_bits = random(); // move to the first posedge of clock
 // initialize rand_bits with a random
 // value using the random() function

 @1 sram.ramAddr = address; // move to the next posedge of clock,
 // drive ramAddr with the value of
 // address
 sram.ce_N = 1'b1; // disable SRAM by driving ce_N high
 sram.ramData = rand_bits; // drive ramData with rand_bits and
 // keep it ready for a Write
 sram.rdWr_N = 1'b0; // drive rdWr_N low and keep it ready
 // for a Write

 @1 sram.ce_N = 1'b0; // move to the next posedge of clock,
 // and enable a SRAM Write by driving
 // ce_N low
 printf("Cycle: %d Time: %d \n", get_cycle(), get_time(0));
 printf("The SRAM is being written at ramAddr: %b Data written: %b \n",
 address, sram.ramData);

 @1 sram.ce_N = 1'b1; // move to the next posedge of clock,
 // disable SRAM by driving ce_N high
 sram.rdWr_N = 1'b1; // drive rdWr_N high and keep it ready
 // for a Read
 sram.ramData = 8'bzzzz_zzzz; // drive a high-impedance value on
 // ramData

 @1 sram.ce_N = 1'b0; // move to the next posedge of clock,
 // enable a SRAM Read by driving ce_N
 // low

 @1 sram.ce_N = 1'b1; // move to the next posedge of clock,
 // disable SRAM by driving ce_N high
 data_result = sram.ramData; // sample ramData and receive the data
 // from SRAM in data_result
 printf("Cycle: %d Time: %d\n", get_cycle(), get_time(0));
 printf("The SRAM is being read at ramAddr: %b Data read : %b \n",
 address, data_result);

} // end of program sram_test
```

The main body of the testbench is the program, which is named `sram_test`. The program contains three data declarations of type `reg` in the beginning. It then moves execution through a Write operation first and then a Read operation. The memory element of the SRAM written to and read from is `6'b 00_0001`. The correct functioning of the SRAM implies data that is stored in a memory element during a Write operation must be the same as that which is received from the memory element during a Read operation later.

The example testbench only demonstrates how any memory element can be functionally validated. For complete functional validation of the SRAM, the testbench would need further development to cover all memory elements from `6'b00_0000` to `6b'11_1111`.

### Interface Description

The generated `.if.vrh` file has to be modified to include the clock input. The modified interface is shown in [Example 73](#).

*Example 73 Example for Interface for SRAM, sram.if.vrh*

```
#ifndef INC_SRAM_IF_VRH
#define INC_SRAM_IF_VRH

interface sram {
 input clk CLOCK; // add clock
 output ce_N OUTPUT_EDGE OUTPUT_SKew;
 output rdWr_N OUTPUT_EDGE OUTPUT_SKew;
 output [5:0] ramAddr OUTPUT_EDGE OUTPUT_SKew;
 inout [7:0] ramData INPUT_EDGE OUTPUT_EDGE OUTPUT_SKew;
} // end of interface sram
#endif
```

The interface consists of signals that are either driven as outputs into the design or sampled as inputs from the design. The clock input, `clk`, is derived from the system clock in the top-level Verilog module.

### Top-level Verilog Module Description

The generated top-level module has been modified to include the clock input for the interface and eliminate code that was not relevant. The clock input is derived from the system clock. [Example 74](#) shows the modified top-level Verilog module for the SRAM.

*Example 74 Top-level Verilog Module, sram.test\_top.v*

```
module sram_test_top;
 parameter simulation_cycle = 100;
 reg SystemClock;
 wire ce_N;
 wire rdWr_N;
 wire [5:0] ramAddr;
 wire [7:0] ramData;
 wire clk = SystemClock; /* Add this line. Interface
 clock input derived from the system clock*/
`ifdef SYNOPSYS_NTB
sram_test vshell(
 .SystemClock (SystemClock),
 .\sram.clk(clk),
 .\sram.ce_N (ce_N),
 .\sram.rdWr_N (rdWr_N),
 .\sram.ramAddr (ramAddr),
 .\sram.ramData (ramData)
);
`endif
```

```

`else

 vera_shell vshell(
 .SystemClock (SystemClock),
 .sram_ce_N (ce_N),
 .sram_rdWr_N (rdWr_N),
 .sram_ramAddr (ramAddr),
 .sram_ramData (ramData)
);
`endif

// design instance
sram dut(
 .ce_N (ce_N),
 .rdWr_N (rdWr_N),
 .ramAddr (ramAddr),
 .ramData (ramData)
);

// system-clock generator
initial begin
 SystemClock = 0;
 forever begin
 #(simulation_cycle/2)
 SystemClock = ~SystemClock;
 end
end
end

endmodule

```

The top-level Verilog module contains the following:

- The system clock, `SystemClock`. The system clock is contained in the port list of the testbench instance.
- The declaration of the interface clock input, `clk`, and its derivation from the system clock.
- The testbench instance, `vshell`. The module name for the instance must be the name of the testbench program, `sram_test`. The instance name can be something you choose. The ports of the testbench instance, other than the system clock, refer to the interface signals. The period in the port names separates the interface name from the signal name. A backslash is appended to the period in each port name because periods are not normally allowed in port names.
- The design instance, `dut`.

## Compiling Testbench With the Design And Running

The VCS command line for compiling both your example testbench and design is the following:

For two-step flow:

### Compilation

```
% vcs -ntb sram.v sram.test_top.v sram.vr
```

### Simulation

```
% simv
```

For three-step flow

### Analysis

```
% vlogan -ntb sram.v sram.test_top.v sram.vr
```

### ElaborationCompilation

```
% vcs top
```

### Simulation

```
% simv
```

You will find the simulation output to be the following:

```
Cycle: 3 Time: 250
The SRAM is being written at ramAddr: 000001 with ramData: 10101100
Cycle: 6 Time: 550
The SRAM is being read at ramAddr: 000001 its ramData is: 10101100
$finish at simulation time 550
V C S S i m u l a t i o n R e p o r t
```

## Key Features

VCS supports the following features for OpenVera testbench:

- [Using Reference Verification Methodology](#)
- [Class Dependency Source File Reordering](#)
- [Using Reference Verification Methodology](#)
- [Functional Coverage](#)
- [Using Reference Verification Methodology](#)

---

## Multiple Program Support

Multiple program support enables multiple testbenches to run in parallel. This is useful when testbenches model stand-alone components (for example, Verification IP (VIP) or work from a previous project). Because components are independent, direct communication between them except through signals is undesirable. For example, UART and CPU models would communicate only through their respective interfaces, and not via the testbench. Thus, multiple program support allows the use of stand-alone components without requiring knowledge of the code for each component, or requiring modifications to your own testbench.

## Configuration File Model

The configuration file that you create, specifies file dependencies for OpenVera programs.

Specify the configuration file as an argument to `-ntb_opts` as shown in the following usage model:

For two-step flow:

```
% vcs -ntb -ntb_opts config=configfileVerilog_and_OV_files
```

For three-step flow:

```
% vlogan -ntb -ntb_opts config=configfile
```

## Configuration File

The configuration file contains the program construct.

The program keyword is followed by the OpenVera program file (`.vr` file) containing the testbench program and all the OpenVera program files needed for this program. For example:

```
//configuration file
program
 main1.vr
 main1_dep1.vr
 main1_dep2.vr
 ...
 main1_depN.vr
 [NTB_options]

program
 main2.vr
 main2_dep1.vr
 main2_dep2.vr
 ...
 main2_depN.vr
 [NTB_options]
```

```
program
 mainN.vr
 mainN_dep1.vr
 mainN_dep2.vr
 ...
 mainN_depN.vr
 [NTB_options]
```

In this example, `main1.vr`, `main2.vr` and `mainN` files each contain a program. The other files contain items such as definitions of functions, classes, tasks and so on needed by the program files. For example, the `main1_dep1.vr`, `main1_dep2.vr` .... `main1_depN.vr` files contain definitions relevant to `main1.vr`. Files `main2_dep1.v`, `main2_dep2.vr` ... `main2_depN.vr` contain definitions relevant to `main2.vr`, and so forth.

## Usage Model for Multiple Programs

You can specify programs and related support files with multiple programs in two different ways:

1. Specifying all OpenVera programs in the configuration file
2. Specifying one OpenVera program on the command line, and the rest in the configuration file

**Note:**

- Specifying multiple OpenVera files containing the program construct at the VCS command prompt is an error.
- If you specify one program at the VCS command line and if any support files are missing from the command line, VCS issues an error.

### Specifying all OpenVera programs in the configuration file

When there are two or more program files listed in the configuration file, the VCS command line is:

```
% vcs -ntb -ntb_opts config=configfile
```

Or

```
% vlogan -ntb -ntb_opts config=configfile
```

The configuration file, could be:

```
program main1.vr -ntb_define ONE
program main2.vr -ntb_incdir /usr/vera/include
```

### Specifying one OpenVera program on the command line, and the rest in the configuration file

You can specify one program in the configuration file and the other program file at the command prompt.

```
% vcs -ntb -ntb_opts config=configfile main2.vr
```

Or

```
% vlogan -ntb -ntb_opts config=configfile main2.vr
```

The configuration file used in this example is:

```
program main1.vr
```

In the previous example, `main1.vr` is specified in the configuration file and `main2.vr` is specified on the command line along with the files needed by `main2.vr`.

## NTB Options and the Configuration File

The configuration file supports different OpenVera programs with different NTB options such as '`include`', '`define`', or '`timescale`'. For example, if there are three OpenVera programs `p1.vr`, `p2.vr` and `p3.vr`, and `p1.vr` requires the `-ntb_define VERA1` runtime option, and `p2.vr` should run with `-ntb_incdir /usr/vera/include` option, specify these options in the configuration file:

```
program p1.vr -ntb_define VERA1
program p2.vr -ntb_incdir /usr/vera/include
```

and specify the command line as follows.

```
% vcs -ntb -ntb_opts config=configfile p3.vr
```

Or

```
% vlogan -ntb -ntb_opts config=configfile p3.vr
```

Any NTB options mentioned at the command prompt, in addition to the configuration file, are applicable to all OpenVera programs.

In the configuration file, you may specify the NTB options in one line separated by spaces, or on multiple lines.

```
program file1.vr -ntb_opts no_file_by_file_pp
```

The following options are allowed for multiple program use.

- `-ntb_define` macro
- `-ntb_incdir` directory

- `-ntb_opts no_file_by_file_pp`
- `-ntb_opts tb_timescale=value`
- `-ntb_opts dep_check`
- `-ntb_opts print_deps`
- `-ntb_opts use_sigprop`
- `-ntb_opts vera_portname`

See the appendix on “Compile-time Options” or “Elaboration Options” for descriptions of these options.

## Class Dependency Source File Reordering

In order to ease transitioning of legacy code from Vera’s make-based single-file compilation scheme to VCS NTB, where all source files have to be specified on the command line, VCS provides a way of instructing the compiler to reorder Vera files in such a way that class declarations are in topological order (that is, base classes precede derived classes).

In Vera, where files are compiled one-by-one, and extensive use of header files is a must, the structure of file inclusions makes it very likely that the combined source text has class declarations in topological order.

If specifying a command line like the following leads to problems (error messages related to classes), adding the analysis option `-ntb_opts dep_check` to the command line directs the compiler to activate analysis of Vera files and process them in topological order with regard to class derivation relationships.

```
% vcs -ntb *.vr
```

Or

```
% vlogan -ntb *.vr
```

By default, files are processed in the order specified (or wildcard-expanded by the shell). This is a global option, and affects all Vera input files, including those preceding it, and those named in `-f file.list`.

When using the option `-ntb_opts print_deps` in addition to `-ntb_opts dep_check` with `vlogan/vcs`, the reordered list of source files is printed on standard output. This could be used, for example, to establish a baseline for further testbench development.

For example, assume the following files and declarations:

```
b.vr: class Base {integer i;}
d.vr: class Derived extends Base {integer j;}
p.vr: program test {Derived d = new;}
```

File `d.vr` depends on file `b.vr`, since it contains a class derived from a class in `b.vr`, whereas `p.vr` depends on neither, despite containing a reference to a class declared in the former. The `p.vr` file does not participate in inheritance relationships. The effect of dependency ordering is to properly order the files `b.vr` and `d.vr`, while leaving files without class inheritance relationships alone.

The following command lines result in reordered sequences.

```
% vcs -ntb -ntb_opts dep_check d.vr b.vr p.vr
```

Or

```
% vlogan -ntb -ntb_opts dep_check d.vr b.vr p.vr
```

```
% vcs -ntb -ntb_opts dep_check p.vr d.vr b.vr
```

Or

```
% vlogan -ntb -ntb_opts dep_check p.vr d.vr b.vr
```

The first command line yields the order `b.vr d.vr p.vr`, while the second line yields, `p.vr b.vr d.vr`.

## Circular Dependencies

With some programming styles, source files can appear to have circular inheritance dependencies in spite of correct inheritance trees being cycle-free. This can happen, for example, in the following scenario:

```
a.vr: class Base_A {...}
 class Derived_B extends Base_B {...}
b.vr: class Base_B {...}
 class Derived_A extends Base_A {...}
```

In this example, classes are derived from base classes that are in the other file, respectively, or more generally, when the inheritance relationships project onto a loop among the files. This is, however, an abnormality that should not occur in good programming styles. VCS detects and reports the loop, and will use a heuristic to break it. This may not lead to successful compilation, in which case you can use the `-ntb_opts print_deps` option to generate a starting point for manual resolution; however, if possible, the code should be rewritten.

## Dependency-based Ordering in Encrypted Files

As encrypted files are intended to be mostly self-contained library modules that the testbench builds upon, they are excluded from reordering regardless of dependencies (these files should not exist in unencrypted code). VCS splits Vera input files into those that are encrypted or declared as such by having the `.vvp` or `.vrhp` file extension or as specified using the `-ntb_vipext` option, and others. Only the latter unencrypted files are subject to dependency-based reordering, and encrypted files are prefixed to them.

**Note:**

The `-ntb_opts dep_check` compilation/analysis option specifically resolves dependencies involving classes and enums. That is, you only consider definitions and declarations of classes and enums. Other constructs such as ports, interfaces, tasks and functions are not currently supported for dependency check.

---

## Using Encrypted Files

VCS NTB allows distributors of Verification IP (Intellectual Property) to make testbench modules available in encrypted form. This enables the IP vendors to protect their source code from reverse-engineering. Encrypted testbench IP is regular OpenVera code, and is not subject to special processing other than to protect the source code from inspection in the debugger, through the PLI, or otherwise.

Encrypted code files provided on the command line are detected by VCS, and are combined into one preprocessing unit that is preprocessed separately from unencrypted files, and is for itself, always preprocessed in `-ntb_opts no_file_by_file_pp` mode. The preprocessed result of encrypted code is prefixed to preprocessed unencrypted code.

VCS only detects encrypted files on the command line (including `-f` option files), and does not descend into include hierarchies. While the generally recommended usage methodology is to separate encrypted from unencrypted code, and not include encrypted files in unencrypted files, encrypted files can be included in unencrypted files if the latter are marked as encrypted-mode by naming them with extensions `.vvp`, `.vrhp`, or additional extensions specified using the `-ntb_vipext` option. This implies that the extensions are considered OpenVera extensions similar to using `-ntb_fileext` for unencrypted files. This causes those files and everything they include to be preprocessed in encrypted mode.

---

## Functional Coverage

The VCS implementation of OpenVera supports the `covergroup` construct. For more information about the covergroup and other functional coverage model, see the section "Functional Coverage Groups" in the VCS OpenVera Language Reference Manual.

---

## Using Reference Verification Methodology

VCS supports the use of Reference Verification Methodology (RVM) for implementing testbenches as part of a scalable verification architecture.

The usage model for using RVM with VCS is:

For two-step flow:

### Compilation

```
% vcs -ntb -ntb_opts rvm [ntb_options] [compile_options] file1.vr
file2.vr file3.v file4.v
```

### Simulation

```
% simv [run_options]
```

For three-step flow

### Analysis

Always analyze Verilog before VHDL.

```
% vlogan -ntb -ntb_opts rvm [vlogan_options] file1.vr file2.vr file3.v
% vhdlan [vhdlan_options] file3.vhd file4.vhd
```

#### Note:

Specify the VHDL bottom-most entity first, and then move up in order.

#### Elaboration

```
% vcs [other_ntb_options] [compile_options] design_unit
```

### Simulation

```
% simv [run_options]
```

For details on the use of RVM, see the *Reference Verification Methodology User Guide*. Though the manual descriptions refer to Vera, NTB uses a subset of the OpenVera language and all language specific descriptions apply to NTB.

Differences between the usage of NTB and Vera are:

- NTB does not require header files (.vrh) as described in the *Reference Verification Methodology User Guide* chapter “Coding and Compilation.”
- NTB parses all testbench files in a single compilation.
- The VCS command-line option *-ntb\_opts rvm* must be used with NTB.

## Limitations

- The handshake configuration of notifier is not supported (since there is no handshake for triggers-syncs in NTB).
- RVM enhancements for assertion support in Vera 6.2.10 and later are not supported for NTB.
- If there are multiple consumers and producers, there is no guarantee of fairness in reads from channels, and so on.

# 16

## Using SystemVerilog

---

VCS supports the SystemVerilog (SV) language (with some exceptions) as defined in the *Standard for SystemVerilog -- Unified Hardware Design, Specification, and Verification Language* (IEEE SystemVerilog LRM Std 1800™-2012).

This chapter describes the following:

- [Use Model](#)
- [Using UVM With VCS](#)
- [Using VMM with VCS](#)
- [Using OVM with VCS](#)
- [Debugging SystemVerilog Designs](#)
- [Functional Coverage](#)
- [SystemVerilog Constructs](#)
- [Extensions to SystemVerilog](#)

For SystemVerilog assertions, see Chapter - [Using SystemVerilog Assertions](#).

---

## Use Model

The use model to analyze, elaborate/compile and simulate your design with SystemVerilog files is as follows:

For two-step flow:

### **Compilation**

```
% vcs -sverilog [compile_options] Verilog_files
```

### **Simulation**

```
% simv [simv_options]
```

For three-step flow:

### **Analysis**

```
% vlogan -sverilog [vlogan_options] file4.sv file5.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

### **Note:**

Specify the VHDL bottommost entity first and then move up in order.

### **Elaboration**

```
% vcs [elab_options] top_cfg/entity/module
```

### **Simulation**

```
% simv [simv_options]
```

To analyze SV files, use the `-sverilog` option with `vlogan/vcs`, as shown in the above use model.

## **Using UVM With VCS**

This version of VCS provides native support for UVM-1.1d. These libraries are located in:

- `$VCS_HOME/etc/uvm-1.1`

UVM 1.1 is now replaced with UVM 1.1d, which is the default. You can load UVM 1.1d by:

- Using the `-ntb_opts uvm` option
- Explicitly specifying the `-ntb_opts uvm-1.1` option

### **Note:**

If you need to override the UVM macro values, ensure that you override that macro in all the vlogans with `-ntb_opts <uvm-version>`. If there is an inconsistency in UVM macro overrides between the vlogans, it may lead to simulation functionality errors.

The following sections explain the options for using UVM with VCS:

- [Update on UVM-1.2](#)
- [Update on UVM-ieee](#)
- [Update on UVM-ieee-2020](#)
- [Natively Compiling and Elaborating UVM-1.1d](#)
- [Natively Compiling and Elaborating UVM-1.2](#)

- [Natively Compiling and Elaborating UVM-ieee](#)
  - [Natively Compiling and Elaborating UVM-ieee-2020](#)
  - [Compiling the External UVM Library](#)
  - [Accessing HDL Registers Through UVM Backdoor](#)
  - [Generating UVM Register Abstraction Layer Code](#)
  - [Recording UVM Transactions](#)
  - [Recording UVM Phases](#)
  - [UVM Template Generator](#)
  - [Using Mixed VMM/UVM Libraries](#)
  - [Migrating from OVM to UVM](#)
  - [Where to Find UVM Examples](#)
  - [Where to Find UVM Documentation](#)
- 

## Update on UVM-1.2

You can load UVM-1.2 using the `-ntb_opts uvm-1.2` option.

**Note:**

You may see some backward compatibility issues while migrating to UVM-1.2.  
For the changes required, refer to the UVM-1.2 release notes

---

## Update on UVM-ieee

You can load UVM-ieee library using the `-ntb_opts uvm-ieee` compile-time option.

---

## Update on UVM-ieee-2020

You can load UVM-ieee-2020 library using the `-ntb_opts uvm-ieee-2020` compile-time option.

---

## Update on UVM-ieee-2020-2.0

You can load UVM-ieee-2020-2.0 library using the `-ntb_opts uvm-ieee-2020-2.0` compile-time option.

---

## Natively Compiling and Elaborating UVM-1.1d

You can compile and elaborate SystemVerilog code, which extends from UVM-1.1d base classes by using the following command:

```
% vcs -sverilog -ntb_opts uvm = [compile_options] \
user_source_files_using_UVM
```

For a mixed-HDL or UUM environment, compile UVM-1.1d with `vlogan` using the following command:

```
% vlogan -ntb_opts uvm [compile_options]
// no source files here!

% vlogan -ntb_opts uvm [compile_options] \
<user source files using UVM>
```

### Note:

- Complete the first step that compiles the UVM library before using the subsequent commands. The first `vlogan` call compiles the UVM library without any need of specifying user source files.
- In specific cases, the subsequent `vlogan` command might error out with Error-[UM] Undefined Macro. In this scenario you must explicitly add ``include uvm_macros.svh` to the file getting this error.

Elaborate the design as follows:

```
%vcs -ntb_opts uvm [elab_options] <top module>
```

Using the `-ntb_opts uvm` option is same as specifying the version explicitly using the `-ntb_opts uvm-1.1` option. However, it is best to specify the version explicitly, because later versions of UVM might carry the default UVM library.

## Natively Compiling and Elaborating UVM-1.2

You can compile and elaborate SystemVerilog code, which extends from UVM-1.2 base classes using the following command:

```
% vcs -sverilog -ntb_opts uvm-1.2 [compile_options] \
user_source_files_using_UVM
```

For a mixed-HDL or UUM environment, compile UVM-1.2 with `vlogan` using the following command:

```
% vlogan -ntb_opts uvm-1.2 [compile_options]
// no source files here!
```

```
% vlogan -ntb_opts uvm-1.2 [compile_options] \
<user source files using UVM>
```

## Natively Compiling and Elaborating UVM-ieee

You can compile and elaborate SystemVerilog code, which extends from UVM-ieee base classes using the following command:

```
% vcs -sverilog -ntb_opts uvm-ieee [compile_options] \
<user_source_files_using_UVM>
```

For a mixed-HDL or UUM environment, compile UVM-ieee with `vlogan` using the following command:

```
% vlogan -ntb_opts uvm-ieee [compile_options]
// no source files here!

% vlogan -ntb_opts uvm-ieee [compile_options] \
<user source files using UVM>
```

## Natively Compiling and Elaborating UVM-ieee-2020

You can compile and elaborate SystemVerilog code, which extends from UVM-ieee-2020 base classes using the following command:

```
% vcs -sverilog -ntb_opts uvm-ieee-2020 [compile_options] \
<user_source_files_using_UVM>
```

For a mixed-HDL or UUM environment, compile UVM-ieee-2020 with `vlogan` using the following command:

```
% vlogan -ntb_opts uvm-ieee-2020 [compile_options]
// no source files here!

% vlogan -ntb_opts uvm-ieee-2020 [compile_options] \
<user source files using UVM>
```

## Compiling the External UVM Library

If you want to use a UVM version from Accellera in place of the UVM-1.1d version shipped with VCS, follow either of these procedures:

- [Using the -ntb\\_opts uvm Option](#)
- [Explicitly Specifying UVM Files and Arguments](#)

## Using the `-ntb_opts uvm` Option

When you set the `VCS_UVM_HOME` environment variable to specify a UVM library directory, VCS uses this location even if the `-ntb_opts uvm` option is used. For example,

```
% setenv VCS_UVM_HOME <path_to_uvm_library>
```

Here, `<path_to_uvm_library>` is the absolute path to the directory that contains the `uvm_pkg.sv` file. Typically, the `uvm_pkg.sv` file is present in the `src` directory inside the Accellera distribution for UVM.

```
% vcs -sverilog -ntb_opts uvm [compile_options] \
user_source_files_using_UVM
```

This is also supported for the UUM flow and for using vlogan.

### Specifying External `uvm_dpi.cc` Source

While using `-ntb_opts uvm`, the `uvm_dpi.cc` is picked up from the UVM installation directory inside the VCS installation directory. However, you might want to use the custom UVM DPI files instead of the ones shipped with the UVM library.

## Explicitly Specifying UVM Files and Arguments

The following example shows how to compile and elaborate the UVM extended code by explicitly specifying the UVM files and arguments:

```
% vcs -sverilog +incdir+${UVM_HOME}/src \
${UVM_HOME}/src/uvm_pkg.sv \
${UVM_HOME}/src/dpi/uvm_dpi.cc \
-CFLAGS -DVCS \
[compile_options] \
user_source_files_using_UVM
```

For a mixed-HDL or UUM environment, compile with `vlogan` using the following command:

```
% vlogan -sverilog +incdir+${UVM_HOME}/src \
${UVM_HOME}/src/uvm_pkg.sv

% vlogan -sverilog +incdir+${UVM_HOME}/src \
<user source files using UVM>
```

Elaborate the design as follows:

```
% vcs[elab_options] \
${UVM_HOME}/src/dpi/uvm_dpi.cc <top module> \
-CFLAGS -DVCS
```

**Note:**

`${UVM_HOME}` should point to your UVM release path. It can also point to  `${VCS_HOME}/etc/uvm-1.1`.

## Accessing HDL Registers Through UVM Backdoor

If you are using tests that need to access HDL registers through the default UVM register backdoor mechanism, add the following option to your command line:

- `-debug_access+r` (for read access)

or

- `-debug_access+f` (for force/write access)

```
% vcs -svverilog {-debug_access+r|-debug_access+f} -ntb_opts uvm
[compile_options] \ user_source_files_using_UVM
```

**Note:**

The `-debug_access+r` or `-debug_access+f` option may affect simulation performance. Therefore, you should use the `+vcs+learn+pli` option to improve the HDL access.

To simulate, use the following command:

```
% simv +UVM_TESTNAME=your_uvm_test [simv_options]
```

If you use the `-b` option with `ralgen`, the `-debug_access+r` or `-debug_access+f` option is not required and the HDL backdoor is enabled through cross-module references instead of the VPI. This provides better performance.

**Note:**

If you want to access VHDL nodes when using UVM-1.1 library, you must specify the `-CFLAGS -DVCS` option during elaboration. With UVM-1.2 library, VHDL nodes access is enabled by default.

## Generating UVM Register Abstraction Layer Code

VCS ships a utility called `ralgen`. Given a description of the available registers and memories in a design, `ralgen` automatically generates the UVM RAL abstraction model for these registers and memories. The description of these registers and memories can be in RALF format or in the IPXACT schema.

To generate a register model from a RALF file, use the following command:

```
% ralgen [options] -t <topname> -uvm <filename.ralf>
```

Here, *filename.ralf* is the name of the RALF input file and *topname* is the top block or system name in the RALF file.

To generate a register model from an IPXACT file, you use a two-step flow. The first step is to generate RALF from IPXACT as follows:

```
% ralgen -ipxact2ralf <input_file>
```

The second step is same as the one described above. For more information, see the *UVM Register Abstraction Layer Generator User Guide*.

## Recording UVM Transactions

UVM has additional features that allow you to take advantage of VCS transaction recording and Verdi's transaction debugging capabilities. These features are available with both the UVM-1.1d and UVM-1.2 libraries.

No compile-time option is needed for UVM-1.1d and UVM-1.2. You can enable recording by using a runtime option. The transaction and the report recordings are stored in the simulation VPD file.

Compiling and Simulating UVM-1.1d and UVM-1.2

To compile and simulate your UVM-1.1d or UVM-1.2 code, see [Enabling FSDB Transaction Recording](#).

## Recording UVM Phases

In addition to UVM transaction recording capabilities, VCS allows you to record the UVM phases and enables the phase debugging capabilities. With this phase recording, you can see the start time and the end time for each component in each phase and the connectivity information for ports in `end_of_elab`. This feature is available with UVM-1.1 libraries in VCS.

To turn on UVM phase recording, use `+UVM_PHASE_RECORD` at runtime and pass the `-debug_access` option during compilation. The phase recordings are stored in the simulation VPD file.

You can enable UVM phase recording in your UVM-1.1 by running the VCS 3-step flow:

- ▶ Define `+UVM_VCS_RECORD` at first step while compiling UVM library:

```
% vlogan -ntb_opts uvm +define+UVM_VCS_RECORD % vlogan -ntb_opts uvm
[compile_options] \ <user source files here>
```

- ▶ Pass the `-debug_access` option at the compilation command line, as shown:

```
% vcs -sverilog -ntb_opts -debug_access [compile_options]
```

- ▶ Enable phase recording during simulation by adding `+UVM_PHASE_RECORD`.

```
% simv +UVM_TESTNAME=<your_uvm_test> \+UVM_PHASE_RECORD [simv_options]
```

You can then use Verdi to debug the UVM phases in *UVM Phase View*. This is supported for interactive debug.

## UVM Template Generator

UVM template generator (`uvmgen`) is a template generator for creating robust and extensible UVM-compliant environments. The primary purpose of `uvmgen` is to minimize the VIP and environment development cycle by providing detailed templates for developing UVM-compliant verification environments. You can also use `uvmgen` to quickly understand how different UVM base classes can be used in different contexts. This is possible because the templates use a rich set of the latest UVM features to ensure the appropriate base classes and their features are picked up optimally.

In addition, `uvmgen` can be used to generate both individual templates and complete UVM environments.

`uvmgen` is a part of the VCS installation. It can be invoked by using the following command:

```
% uvmgen [-L libdir] [-X] [-o fname] [-O]
```

where,

- `L` Takes user-defined library for template generation
- `X` Excludes the standard template library
- `o` Generates templates in the specified file
- `O` Overwrites if the file already exists
- `q` Quick mode to generate the complete environment

For more information, see the *UVM Template Generator (uvmgen) User Guide*.

## Using Mixed VMM/UVM Libraries

For interoperability reasons (using UVM components in a VMM environment and vice versa), VCS allows you to load the VMM and UVM libraries simultaneously along with the VMM/UVM interop kit.

The VMM/UVM interop kit is located in the following directory:

- `$VCS_HOME/etc/uvm-1.1/uvm_vmm_pkg.sv` (for UVM-1.1 and later UVM releases)

You can load mixed VMM-1.2 and UVM by using a combination of the following VCS options:

- -ntb\_opts uvm[1.1]+rvm

-or-

- -ntb\_opts rvm+uvm[1.1]

`-ntb_opts uvm[1.1]+rvm` is supported for both the mixed-HDL and UUM flows.

For example, in the two-step flow:

```
%vcs -ntb_opts uvm+rvm compile-time_options source_files
```

In the UUM three-step flow:

```
%vlogan -ntb_opts uvm <no_other_files or options>
%vlogan -ntb_opts uvm+rvm <compile_options> <compile_files>
%vcs -ntb_opts uvm+rvm <compile_options> <top_module_name>
```

You can turn off the automatic inclusion of `uvm_vmm_pkg.sv` using the `+define+NO_VMM_UVM_INTEROP` option.

For example, in the two-step flow:

```
%vcs -ntb_opts uvm+rvm +define+NO_VMM_UVM_INTEROP compile-time_options
source_files
```

In the UUM flow:

```
vlogan -ntb_opts uvm
```

```
vlogan -ntb_opts uvm+rvm +define+NO_UVM_VMM_INTEROP other_options
source_files
```

```
vcs -ntb_opts uvm+rvm compile-time_options source_files
```

By default, the mixed environment is driven by a VMM top timeline. However, you can define a UVM top using the `+define+UVM_ON_TOP` option, as shown:

```
%vcs -ntb_opts uvm+rvm +define+UVM_ON_TOP compile-time_options
source_files
```

In UUM flow:

```
vlogan -ntb_opts uvm
```

```
vlogan -ntb_opts uvm+rvm +define+UVM_ON_TOP other_options source_files
```

```
vcs -ntb_opts uvm+rvm compile-time_options top_module_name
```

The UVM/VMM interop kit examples are located in the `$VCS_HOME/doc/examples/uvm_vmm_interop_kit` directory.

For details on how to use the native VMM/UVM interop kit, refer to the Class Reference section available in the *VCS Online documentation > UVM\_VMM documentation*.

**Note:**

In this version of VCS, the UVM-EA and VMM-1.2 interop kit is no longer included. If you need either one of these kits, contact [vcs\\_support@synopsys.com](mailto:vcs_support@synopsys.com).

---

## Migrating from OVM to UVM

To convert your OVM code to UVM, you can use a script stored in  `${VCS_HOME}/bin/OVM_UVM_Rename.pl`. This script makes the migration process easy.

**Note:**

This process is simple for SystemVerilog code that extends from OVM 2.1.1 onward.

Use the following command to convert your OVM code to UVM code:

```
% OVM_UVM_Rename.pl
```

This script hierarchically changes all occurrences of `ovm` to `uvm_` for files with `.v`, `.vh`, `.sv`, and `.svh` extensions.

Change the simulation command line by replacing `OVM_TESTNAME` with `UVM_TESTNAME`.

**Note:**

Some additional work is required for the base classes that differ between OVM and UVM. For example, you may need to modify callbacks, some global function names, arguments, and so on.

---

## Where to Find UVM Examples

The UVM-1.1d interop examples are located in the following directory:

```
$VCS_HOME/doc/examples/uvm
```

The UVM-VMM interop examples are located in the following directory:

```
$VCS_HOME/doc/examples/uvm_vmm_interop_kit
```

---

## Where to Find UVM Documentation

The UVM-1.1d and UVM-VMM interop documentation is available in the following locations.

### UVM-1.1d Documentation

The PDF version of the *UVM-1.1d User Guide* (`uvm_users_guide_1.1.pdf`) is located under VCS documentation in SolvNetPlus.

The PDF version of the *UVM-1.1d Reference Guide* (`UVM_Class_Reference_1.1.pdf`) is located under VCS documentation in SolvNetPlus.

### UVM-VMM Interop Documentation

The unified HTML version of the *UVM-VMM Interop Reference Guide* is accessible in VCS *Online documentation > UVM\_VMM documentation*.

---

## Using VMM with VCS

The use model to use VMM with VCS is as follows:

For two-step flow:

### Compilation

```
% vcs -sverilog -ntb_opts rvm [compile-time_options] Verilog_files
```

### Simulation

```
% simv [simv_options]
```

For three-step flow

### Analysis

```
% vlogan -sverilog -ntb_opts rvm [vlogan_options] file4.sv file5.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

### Note:

Specify the VHDL bottommost entity first, then move up in order.

### Elaboration

```
% vcs [elab_options] top_cfg/entity/module
```

### Simulation

```
% simv [simv_options]
```

To analyze SystemVerilog files using VMM, use the options `-sverilog` and `-ntb_opts` `rvm` with `vlogan/vcs`, as shown in the above use model.

For more information on VMM, refer to the *Verification Methodology Manual for SystemVerilog*.

## Using OVM with VCS

VCS provides native support for OVM 2.1.2. The libraries are located in:

```
$VCS_HOME/etc/ovm
```

### Native Compilation and Elaboration of OVM 2.1.2

You can compile and elaborate SystemVerilog code that extends from OVM 2.1.2 base classes by using the following command:

```
% vcs -sverilog -ntb_opts ovm [compile_options] \
<user source files using OVM>
```

When you natively compile and elaborate the OVM code, you do not have to explicitly include OVM source files in the user code as they get parsed by default.

For a mixed-HDL or UUM environment, compile OVM 2.1.2 with `vlogan` using the following command:

```
% vlogan -ntb_opts ovm [compile_options]
// no source files here!
% vlogan -ntb_opts ovm -sverilog [compile_options] \
<user source files using OVM>
```

#### Note:

- Complete the first step that compiles the OVM library before using the subsequent command. The first `vlogan` call compiles the OVM library, which does not contain any user source files.
- In specific cases, the subsequent `vlogan` command might error out with Error-[UM] Undefined macro. In this scenario, explicitly add `include "ovm\_macros.svh" to the file encountering this error.

Elaborate the design as follows:

```
% vcs -ntb_opts ovm [elab_options] <top module>
% simv +OVM_TESTNAME=<ovm testname> <simv options>
```

Using the `-ntb_opts ovm` option is same as specifying the version by explicitly using the `-ntb_opts ovm-2.1.2` option.

In some cases, if you have explicitly included `ovm.svh`, then the OVM source code is recompiled in subsequent `vlogan` command. To avoid re-compilation, you need to add the `+define+OVM_SVH` option in the subsequent `vlogan` commands.

```
% vlogan -ntb_opts ovm [compile_options]
// no source files here!

% vlogan -ntb_opts ovm -sverilog +define+OVM_SVH [compile_options] \
<user
source files using OVM>
```

In cases where `include "ovm\_pkg.sv" is present in the user code, recompilation of the OVM source code is required. To avoid this, you need to pass the `+define+OVM_PKG_SV` option in the subsequent `vlogan` commands.

```
% vlogan -ntb_opts ovm [compile-time_options]
// no source files here!

% vlogan -ntb_opts ovm -sverilog +define+OVM_PKG_SV
[compile-time_options] user_source_files_using_OVM
```

## Compiling the External OVM Library

If you want to use an OVM version from Accellera in place of the OVM 2.1.2 version shipped with VCS, use one of the following procedures:

- [Using the -ntb\\_opts ovm Option](#)
- [Explicitly Specifying OVM Files and Arguments](#)

### Using the -ntb\_opts ovm Option

When you set the `VCS_OVM_HOME` environment variable to specify a OVM library directory, VCS uses this location even if the `-ntb_opts ovm` option is used. For example,

```
% setenv VCS_OVM_HOME /<path_to_ovm_library>/myOVM-2.1.2
% vcs -sverilog -ntb_opts ovm [compile_options] \
<user source files using OVM>
```

This is also supported for the UUM flow and for using `vlogan`.

### Explicitly Specifying OVM Files and Arguments

The following example shows how to compile and elaborate the OVM extended code by explicitly specifying the OVM files and arguments:

```
% vcs -sverilog +incdir+${OVM_HOME} \
${OVM_HOME}/ovm_pkg.sv \
[compile_options] \
<user source files using OVM>
```

For a mixed-HDL or UUM environment, compile with vlogan using the following command:

```
% vlogan -sverilog +incdir+${OVM_HOME} \
${OVM_HOME}/ovm_pkg.sv

% vlogan -sverilog +incdir+${OVM_HOME} \
user_source_files_using_OVM
```

**Note:**

`${OVM_HOME}` should point to your OVM release path. It can also point to  `${VCS_HOME}/etc/ovm-2.1.2`

## Recording OVM Transactions

The OVM version shipped with VCS has additional features that allows you to take advantage of VCS and Verdi's transaction recording and debugging capabilities.

To turn on OVM transaction recording, you need to use a specific compile-time option for OVM or use the `-debug_access` option with VCS in the two-step flow and then enable recording using a different runtime option. The transaction and report recordings are stored in the simulation VPD file.

To compile your OVM code, add the `-debug_access` option to your `vcs` command line.

For the three-step flow, you need to provide the `+define+OVM_VCS_RECORD` option to the first `vlogan` command line, as shown along with any of the `-debug` options.

For example, in the two step flow:

```
% vcs -sverilog -ntb_opts ovm -debug_access\
[compile-time_options]
```

In the UUM flow:

```
% vlogan -ntb_opts ovm +define+OVM_VCS_RECORD[compile_options]
// no source files here!
```

```
% vlogan -ntb_opts ovm [compile_options] \
<user source files using OVM>
```

**Note:**

- Complete the first step that compiles the OVM library before using the subsequent commands. The first `vlogan` call compiles the OVM library. Define `OVM_VCS_RECORD` at this step to enable transaction recording that is without any specified user source files.
- In specific cases, the subsequent `vlogan` commands might error out with `Error-[UM] Undefined macro`. In this scenario, you must explicitly add ``include "ovm_macros.svh"` to the file getting this error.

Elaborate the design as follows:

```
% vcs -ntb_opts ovm [elab_options] top_module -debug_access
```

To simulate, use +OVM\_TR\_RECORD to turn on the transaction recording and use +OVM\_LOG\_RECORD to turn on the recording of OVM report log messages:

```
% simv +OVM_TESTNAME=<my_ovm_testname> +OVM_TR_RECORD \
+OVM_LOG_RECORD [simv_options]
```

You can then use Verdi to debug the transactions and log messages. This is supported for interactive debug. The recorded streams with transactions and report logs are available in the VMM/OVM folder of the transaction browser.

## Debugging SystemVerilog Designs

VCS provides UCLI commands to perform the following tasks to debug a design:

| Task                          | Related UCLI commands are... |
|-------------------------------|------------------------------|
| Line stepping                 | step next run                |
| Thread debugging              | step thread                  |
| Setting breakpoints           | stop run                     |
| Mailbox related information   | show                         |
| Semaphore related information | show                         |

For detailed information on the UCLI commands, see the *UCLI User Guide*.

## Functional Coverage

The VCS implementation of SystemVerilog supports the `covergroup` construct, which you specify as the user. These constructs allow the system to monitor values and transitions for variables and signals. They also enable cross-coverage between variables and signals.

If you have covergroups in your design, VCS collects the coverage data during simulation and generates a database, `simv.vdb`. Once you have `simv.vdb`, you can use the Unified Report Generator to generate text or HTML reports. For more information about covergroups, see the *VCS SystemVerilog LRM*. For more information about functional coverage generated in VCS, see the *Coverage Technology User Guide*.

---

## SystemVerilog Constructs

VCS has implemented the following SystemVerilog constructs in recent releases:

- [Extern Task and Function Calls through Virtual Interfaces](#)
- [Modport Expressions in an Interface](#)
- [Interface Classes](#)
- [Package Exports](#)
- [Severity System Tasks as Procedural Statements](#)
- [Width Casting Using Parameters](#)
- [The std::randomize\(\) Function](#)
- [SystemVerilog Bounded Queues](#)
- [wait\(\) Statement with a Static Class Member Variable](#)
- [Support for Consistent Behavior of Class Static Properties](#)
- [Parameters and Local Parameters in Classes](#)
- [SystemVerilog Math Functions](#)
- [Streaming Operators](#)
- [Constant Functions in Generate Blocks](#)
- [Support for Aggregate Methods in Constraints Using the “with” Construct](#)
- [Debugging During Initialization SystemVerilog Static Functions and Tasks in Module Definitions](#)
- [Explicit External Constraint Blocks](#)
- [Generate Constructs in Program Blocks](#)
- [Error Condition for Using a Genvar Variable Outside of its Generate Block on page 796](#)
- [Randomizing Unpacked Structs](#)
- [Using a Package in a SystemVerilog Module, Program, and Interface Header](#)
- [Disabling DPI Tasks and Functions](#)
- [Support for Overriding Parameter Values through Configuration](#)
- [Support for Inclusion of Dynamic Types in Sensitivity List](#)

- Support for Assignment Pattern Expression in Non-Assignment Like Context
- User-Defined Nettypes
- Generic Interconnect Nets
- Support for Associative Array With Unpacked Structure as Key
- Specifying a SystemVerilog Keyword Set by LRM Version at Command Line
- Support for .triggered Property with Clocking Block Name
- Support for Intra Assignment Delay With Non-Blocking Assignments in Program Block
- Support for Nested Randsequence
- Support for Typed Constructor Call as an Argument to Task or Function
- Support for Select of Unpacked Structure Array as an Argument to Task or Function
- Support for Function Returning Unpacked Structure in Conditional Operator in a Continuous Assignment Statement
- Support for Built-in Method Which Returns Work Library Name
- Support for Function call in Clocking Block Hierarchical Expression

## Extern Task and Function Calls through Virtual Interfaces

You can define tasks and functions in an interface with one or more of the modules connected by the interface. You declare them as export in a modport or as extern in the interface. When they are called through virtual interfaces, the actual task or function that VCS executes depends on the interface instance of the virtual interface.

### Example of exporting tasks in modports

```
interface simple_bus ; // Define the interface
modport slave (export task Read);
endinterface: simple_bus

module memMod (simple_bus sb_intf);
task sb_intf.Read; // Read method
...
endtask
endmodule

module top;
simple_bus sb_intf(); // Instantiate the interface
memMod mem(sb_intf.slave); // exports the Read tasks
endmodule
```

### Example of extern tasks in interfaces

```
interface intf;
 extern task T1();
 extern task T2();
endinterface

module top;
 intf i1();
 intf i2();

 virtual intf vi;
 M1 m1(i1);
 M2 m2(i1);
 M3 m3(i2);
 M4 m4(i2);

 initial begin
 vi = i1;
 vi.T1(); // Task i1.T1 in M1
 vi.T2(); // Task i1.T2 in M2
 vi = i2;
 vi.T1(); // Task i2.T1 in M3
 vi.T2(); // Task i2.T2 in M4
 end
endmodule

module M1(intf i1);
task i1.T1;
...
endtask
endmodule

module M2(intf i1);
task i1.T2;
...
endtask
endmodule

module M3(intf i2);
task i2.T1;
...
endtask
endmodule

module M4(intf i2);
task i2.T2;
...
endtask
endmodule
```

The definition of extern subroutines within an interface shall observe the following rules:

- Each interface instance may have different implementations of its *extern* subroutines.
  - The same *extern* subroutine of different interface instances can be defined in different modules.
  - Different *extern* subroutines of the same interface instance can be defined in different modules.
- Every interface instance must have one and only definition of its *extern* subroutines.
  - If an interface instance containing an *extern* subroutine, then one of the modules connected must define that subroutine.
  - Any *extern* subroutine of an interface instance cannot be defined in more than one module.
  - The module implementing any *extern* subroutine can be instantiated only once.
- These rules apply for exported subroutines in modports as well.

### Limitations

The feature has the following limitations:

- External task and function calls through virtual interface are not supported in constraints.
- The interface containing an external task or function can only be passed as port to module and program scopes. It cannot be instantiated inside the module defining the external task/function of interface and doing so, it gives an error.

## Modport Expressions in an Interface

As described in the IEEE SystemVerilog LRM Std 1800™-2012 Section 25.5.4 “Modport expressions,” a modport expression allows you to include the following in the modport list in an interface:

- elements of an array or structure
- concatenation of elements
- assignment pattern expressions

VCS has implemented modport expressions. For assignment pattern expressions, constant expressions are allowed only with input ports.

The following interface example includes modport expressions in a modport connection list.

*Figure 161 Modport Expressions*

```
interface interoceter_intf;
 logic [63:0] intrctr_bus;
 const longint flag=1879048192;
 wire [3:0] w1,w2;
 logic [7:0] log1,log2;
 modport mp1 (output .out(intrctr_bus[31:0]),
 input .in(flag),.pt({log1,log2}));
 modport mp2 (output .out(intrctr_bus[63:32]),
 input .in(7));
endinterface
```

part-select of a vector



concatenating elements



The modport expressions in [Figure 161](#) are as follows:

```
.out(intrctr_bus[31:0])
.in(flag)
.pt({log1,log2})
.out(intrctr_bus[63:32])
.in(7)
```

Modport expressions consist of the following:

1. a port name preceded by a period, for example:.out or.pt.
2. The expression enclosed in parentheses, for example:  
(intrctr\_bus[64:32])

## Limitations

In addition to the assignment pattern expressions of elements declared in the interface being limited to the input ports, the feature has the following limitations:

- ref ports with modport expressions are not supported.
- Cross-module references in modport expressions are not supported.
- Modport expressions containing a mix of nets and variables are not supported.
- Modport expressions connected to OpenVera ports are not supported.
- Multiple driver checks are not yet supported.

## Interface Classes

An interface class can be seen more as a virtual class whose methods have to be pure virtual (see IEEE SystemVerilog LRM Std 1800™-2012 Section 8.22 “Polymorphism: dynamic method lookup”). In addition to the pure virtual methods, interface classes can have type declarations (see the IEEE SystemVerilog LRM 1800™-2012 Section 8.24 “Out-of-block declarations”) and parameter declarations (see IEEE SystemVerilog LRM Std 1800™-2012 Section 6.20 “Constants” and Section 8.27 “Typedef class”). Constraint blocks, nested classes and covergroups are not allowed inside an interface class. An interface class cannot be nested inside any other class.

Just like a virtual class, an interface class cannot be instantiated. However, its handle can be created and all the pure virtual methods declared inside the interface class or its base classes can be accessed through its class handle.

At runtime, a virtual class handle needs to point to a concrete class, which is derived from the virtual class directly or indirectly and provides a concrete implementation of all its pure virtual methods. This approach applies a restriction that all the common methods should go into their virtual class, so that they can be accessed using a virtual class handle and all the objects that can be used to be pointed to by such virtual class handles must be descendants of the virtual class. Interface classes do away with the restriction of being descendant of it in inheritance hierarchy to be able to be pointed to by its handle, thus mitigating the effect of not having multiple inheritance.

Interface classes can have multiple inheritance, that is they can inherit zero or more interface classes using the `extends` keyword. In case of multiple inheritance, name conflicts should be resolved.

An interface class handle can point to class objects that implement the interface class. A class implements an interface class using the keyword `implements`. A class can implement zero or more interface classes using the `implements` keyword. The `implements` keyword is not inheritance. It is a condition. So, classes implementing interface classes do not inherit any type or parameters from them. However, implementing classes can refer to types and parameters inside an interface using the scope resolution `::` operator. A class implements an interface class. If it itself implements the interface class or if any of its ancestor implements the interface class.

For a non-interface class to implement an interface class, it must provide implementations for the set of methods declared in interface class as pure virtual that satisfy the requirements of a virtual class method override (see the IEEE SystemVerilog LRM Std 1800™-2012 Section 8.21 “Abstract classes and pure virtual methods”).

```
interface class I;
 pure virtual function string getName();
endclass

class Foo implements I;
```

```

 virtual function string getName();
 return "Foo";
 endfunction
 endclass

 class Bar extends Foo;
 virtual function string getName();
 return "Bar";
 endfunction
 endclass

 module test;
 Bar bar = new;
 Foo foo = new;
 I i;
 //I i1 = new;
 // Like virtual class, interface class can't be instantiated.
 initial begin
 i = bar;
 $display(i.getName());
 i = foo;
 $display(i.getName());
 end
 endmodule

```

In the above example, interface class `I` has a pure virtual method `getName`, and a non-interface class `Foo` implements interface class `I`. Because `Bar` class extends `Foo`, so, `Bar` also implicitly implements interface class `I`. class `Foo` provides implementation for pure virtual function declared inside interface class `I`.

A class that implements an interface class must have a virtual method for every pure virtual method in its interface class. A class that is implementing an interface class can provide implementation for an interface class pure virtual method either by inheriting/ overriding a virtual method or by defining its own virtual method.

A virtual class can also implement an interface class. When a virtual class implements an interface class it must either provide a method implementation for the pure virtual method of interface class or re-declare the method prototype with the pure qualifier.

An interface class handle can point to objects of only those classes that either directly or indirectly implement an interface class using the `implements` keyword.

Methods declared inside the interface classes are allowed to have default arguments. The default argument should be a constant expression and it should be same for all the implementing classes. The default argument should be evaluated in the scope containing the method declaration.

## Difference Between Extends and Implements

When a class extends another class, it inherits all the members and methods of its superclass that are accessible to it based upon the access type. However, when a class

implements an interface class, nothing is inherited. If the class needs to access a type or parameter of the interface class, then the class should use the scope resolution `::` operator to access a member of the interface class, just like the way the static members of a class are accessed.

- An interface class can extend zero or more interface classes using the `extends` keyword.
- An interface class cannot extend a non-interface class.
- An interface class cannot implement another interface or non-interface class.
- An interface class cannot extend a type parameter.
- An interface class cannot extend a forward declared interface class.
- A non-interface class can extend zero or one non-interface class using the `extends` keyword.
- A non-interface class cannot extend an interface class.
- A non-interface class can implement zero or more interface classes using the `implements` keyword.
- A non-interface class cannot implement non-interface class.
- A non-interface class can extend a class and implement interface classes simultaneously.
- A non-interface class cannot implement a type parameter.
- A non-interface class cannot implement a forward declared interface class.

```
interface class Shape;
 typedef string NAME;
 pure virtual function NAME getShape();
endclass

interface class Area;
 pure virtual function int getArea();
endclass

class Rectangle implements Shape, Area;
 int x;
 int y;
 //virtual function NAME getShape();
 //illegal NAME is not accessible

 virtual function Shape::NAME getShape(); // legal
 return "Rectangle";
 endfunction

```

```

virtual function int getArea();
 return (x*y);
endfunction
endclass

class Square extends Rectangle;
 virtual function string getShape();
 return "Square";
 endfunction
endclass

module test;
 Shape s;
 Rectangle r = new;
 Square sq = new;
 initial begin
 s = r;
 $display(s.getShape());
 s = sq;
 $display(s.getShape());
 end
endmodule

```

In the above example, the `Rectangle` class implements the `Shape` and `Area` interface classes using the `implements` keyword and provides implementation for both the pure virtual methods declared inside these interface classes. The `Square` class indirectly/implicitly implements the `Shape` and `Area` interface classes.

The above example also shows that types and parameters inside the interface class cannot be accessed directly by the class implementing the interface class. Because, when a class implements an interface class, it does not inherit anything from interface class. Types and parameters inside an interface class are static and can be accessed using the class scope resolution `::` operator (see the IEEE SystemVerilog LRM Std 1800™-2012 Section 8.24 “Out-of-block declarations”)

## Cast and Interface Class

If a class implements an interface class, then the class' object can be assigned to be the handle of that interface class.

```

interface class I1;
endclass
interface class I2;
endclass

interface class I3 extends I1, I2;
endclass

interface class I4;
endclass

```

```

class A implements I3, I4;
endclass

class B extends A;
endclass

A a = new;
B b = new;
I1 i1 = a; // legal, as class A implements interface
 // class I3 which extends I1;
I2 i2;
I3 i3 = a; // legal, as class A implements interface class I3;
$cast(i2, b); // casting is not required, as class B
 // extends A which implements interface class
 // I3 which in turn extends I2;
$cast(a, i2); // valid, casting is must here.

```

Interface class handles can be cast dynamically if the actual object assigned to destination is valid.

```

$cast(i4, i3); // valid, as i3 is pointing to object of
 // class "A", which implements interface
 // class I4;

```

## Name Conflicts and Resolution

A class can implement multiple interface classes and interface classes can extend multiple interface classes. In such cases, identifiers from multiple name spaces may become visible in the single name space leading to name conflicts. Such name conflicts must be resolved, even when there is no usage of the identifier in the current scope.

### Name Conflicts During Implementation

A class can implement multiple interface classes and when the same method name appears in more than one interface classes, a method name conflict happens that must be resolved by providing an implementation of the method that simultaneously provides implementation for all the implemented interface classes' method with the same name.

```

interface class I1;
 pure virtual task t();
endclass

interface class I2;
 pure virtual task t();
endclass

class A implements I1, I2;
 virtual task t();
 $display("A::t");
 endtask
endclass

```

In this example class `A` is implementing two interface classes `I1` and `I2` both of which has method with name `t`. The class `A` resolves the conflict by providing an implementation of the virtual task `A::t` that simultaneously provides implementation for both `I1::t` and `I2::t`. However, it may not be always possible to resolve such conflict and thus it results in an error.

```
interface class I1;
 pure virtual task t(int i);
endclass

interface class I2;
 pure virtual task t();
endclass

class A implements I1, I2;
 virtual task t();
 $display("A::t");
 endtask
endclass
```

In this example, although `A::t` provides a valid implementation for `I2::t`, but it does not provide a valid implementation for `I1::t`, so, it is an error.

### Name Conflicts During Inheritance

An interface class can inherit type, parameters, and pure virtual methods from multiple base classes. If the same name is inherited from multiple base interface classes, then a name conflict occurs and it must be resolved. Types and parameters name conflict should be resolved by providing a declaration of type/parameter that overrides all such name collisions. For methods it should provide a single prototype that overrides all the name collisions. The method prototype must also be a valid virtual method override (see the IEEE SystemVerilog LRM Std 1800™-2012 Section 8.21 “Abstract classes and pure virtual methods”) for any inherited method of the same name.

```
interface class I1;
 pure virtual task t(int i);
endclass

interface class I2;
 pure virtual task t(int i);
endclass

interface class I3 extends I1, I2;
 pure virtual task t(int i);
endclass
```

In this example, `I3` inherits method `t` from both `I1` and `I2`. So, it provides a prototype for method `t` which resolves the conflict and the prototype is also a valid virtual method override.

```

interface class I1;
 pure virtual task t(int i);
endclass

interface class I2 extends I1;
endclass

interface class I3 extends I1;
endclass

interface class I4 extends I2, I3;
endclass

```

In the case of diamond relationship, name conflict does not occur if the originating class for the method is same. So, in the above example, there is no conflict for method name `t` in class `I4` because `I2::t` and `I3::t`, which `I4` inherits, are actually coming from the same interface class `I1`. Therefore, there is only a single copy of `t` in `I4` leading to no name conflict.

```

interface class I1;
 typedef int INT;
 pure virtual task t(INT i);
endclass

interface class I2;
 typedef int INT;
 pure virtual task t1(INT j);
endclass

interface class I3 extends I1, I2;
 typedef int INT;
endclass

```

In the above example, even though `INT` is of the same type in both the interface classes `I1` and `I2`, still `I3` needs to resolve the conflict by redefining the type.

## Interface Class and Randomization

It is legal to call the `randomize()` method on an interface class handle. An inline constraint is also legal on an interface class handle. However, it is of little use because interface class cannot have any members. The `rand_mode()` and `constraint_mode()` methods are illegal interface class handle.

Unlike non-interface classes, interface classes have virtual and empty `pre_randomize()` and `post_randomize()` built-in methods. So, any direct call to these methods using interface class handle leads to call to these empty methods. However, when `randomize` is called on interface class handle, it leads to the call of `randomize` method on the class object that are pointed to by the interface class handle. This in turn internally calls `pre_randomize()` and `post_randomize()` methods of the actual object that is pointed to by the interface class handle.

```

interface class I;
endclass

class A implements I;
 rand int i;
 function void pre_randomize();
 $display("A::pre_randomize", i);
 endfunction

 function void post_randomize();
 $display("A::post_randomize", i);
 endfunction
endclass

A a = new;
I i = a;

i.randomize(); // it would call A::pre_randomize()
 // and A::post_randomize() internally.
i.pre_randomize(); // built-in empty body I::pre_randomize
 // would be called.

```

## Package Exports

Declarations imported into a package are not visible by way of subsequent imports of that package by default. Package export declarations allow a package to specify those imported declarations to be made visible in subsequent imports.

There are three forms of export directives:

```
export pkg::name;
```

This directive both imports and exports `name` from the specified package named `pkg`.

```
export pkg::*;


```

This directive exports all names imported from the `pkg` package into the current package. The imports can be by name reference or by named export directive.

```
export *::*;


```

Exports all names imported from any packages into the current package. The imports can be by name reference or by named export directive. An export directive `*::*` must match at least one of the import directive.

Unlike package import directives, package export directives *can only* occur at *package scope* and cannot occur in `$unit`.

[Figure 162](#) illustrates the package export functionality:

*Figure 162 The Package Import Functionality Example 1*

```

package p1;
 int x, y;
endpackage
package p2;
 import p1::x;
 export p1::*;
endpackage

package p3;
 import p1::*;
 import p2::*;
 export p2::*;
 int q = x;
endpackage

```

exports `p1::x` as the variable named `x`

`p1::x` and `p2::x` are the same declaration

`p1::x` and `q` are made available from `p3`

Although `p1::y` is a candidate for import, it is not actually imported since it is not referenced.

Since `p1::y` is not imported, it is not made available by the export

## Severity System Tasks as Procedural Statements

The severity system tasks can be included as procedural statements in user-defined tasks and functions, in `initial`, `final` and any `always` blocks.

[Example 75](#) shows the usage of these system tasks in an initial block

*Example 75 Severity Statements in Procedural Blocks*

```

initial
 if (Verilog_simulator == "VCS")
 $display("\n\t Smart User! \n");
 else
 begin
 #10 $warning(2, "\n\t Expect a performance cost \n\n");
 if (Verilog_simulator == "Questa_questionable")
 #10 $info (3, "\n\t you paid too much \n\n");
 if (Verilog_simulator == "Indecisive")
 #10 $fatal(1, "\n\t give up now\n");

```

In [Example 75](#), because the conditional expression (`Verilog_simulator == "VCS"`) is true, VCS displays the following when it compiles and simulates [Example 75](#):

Smart User!

For conditional expression (`Verilog_simulator == "Indecisive"`), VCS displays the following when it compiles and simulates [Example 75](#):

```
Warning: "expl.sv", 21: mod: at time 10
 2
 Expect a performance cost
```

```
Fatal: "expl.sv", 25: mod: at time 20
 give up now
```

#### Note:

The severity system tasks can be used as elaboration system tasks. Elaboration system tasks require the `-assert svaext` compile-time option and the keyword argument.

## Width Casting Using Parameters

VCS used to support width casting using integers only, such as `4'(x)`. However, according to the IEEE SystemVerilog LRM Std 1800™-2012 Section 6.24.1 “Cast operator”:

“If the casting type is a constant expression with a positive integral value, the expression in parentheses shall be padded or truncated to the size specified. It shall be an error if the size specified is zero or negative.”

VCS now supports width casting for any constant expression also. For example:

```
parameter p = 16;
(p+1)'(x-2) // This is now supported
```

According to the syntax:

```
casting_type ::= simple_type | constant_primary | signing | string |
 const
constant_primary ::= // from A.8.4
primary_literal | ps_parameter_identifier constant_select
| specparam_identifier [[constant_range_expression]]
| genvar_identifier35 | [package_scope | class_scope] enum_identifier
| constant_concatenation | constant_multiple_concatenation
| constant_function_call | (constant_mintypmax_expression)
| constant_cast | constant_assignment_pattern_expression |
type_reference36
```

The constant expressions could also include parameter cross-module references. So, the following examples are legal and are now supported.

**Example 76 Casting for a Parameter with an Expression**

```
module test (input clk);
 parameter integer signed XYZ_NUMBER_VL = 3;

 logic [3:0] next_vls_in_use_reg;
 int next_vl_to_use_reg, next_vl_to_use_re2;
 int XYZ_VL_SIZE, vl_index;

 always @ (posedge clk) begin
 vl_index = 5;

 next_vl_to_use_re2 = 4'(3); // ok
 next_vl_to_use_reg = XYZ_NUMBER_VL'(vl_index) ;
 // the line above with an expression now supported
 end
endmodule
```

**Example 77 Casting for a Parameter with a Localparam**

```
program p1;
 localparam int aa=4;
 localparam int bb = 10;

 logic[aa-1:0] mytime;
 initial begin
 mytime = aa'(bb); // line now supported
 end
endprogram
```

Casting type can be any positive constant expression. The expression in the parenthesis can be padded or truncated based on the casting type. Cast type can also be parameter cross-module references (which are constant expressions) that can include concatenation as well as assignment patterns.

**Example 78 Casting type a positive constant expression**

```
module m #(p = 0);
endmodule

module test;
 localparam int P1=4;
 localparam int P2 = 10;

 logic[P1-1:0] mytime;

 m #(2) u1();

 initial begin
 mytime = (u1.P1+u1.P1)'(bb);
```

```
// line above now supported
end
endmodule
```

---

## The std::randomize() Function

The `randomize()` function randomizes variables that are not class members.

### Syntax

```
[std::]randomize(variable-identifier-list)
[with constraint-block]
```

### Description

SystemVerilog defines extensive randomization methods and operators for class members. Most modeling methodologies recommend the use of classes for randomization. However, there are situations where the data to be randomized is not available in a class. SystemVerilog provides the `std::randomize()` function to randomize variables that are not class members.

The `std::randomize()` function can be used in the following scopes:

- module
- function
- task
- class method

Arguments to `std::randomize()` can be of integral types including:

- integer
- bit vector
- enumerated type

Object handles and strings cannot be used as arguments to `std::randomize()`.

The variables passed to `std::randomize()` must be visible in the scope where the function is called. Cross-module references are not allowed as arguments to the `std::randomize()` function.

All constraint expressions currently available with `obj.randomize()` in VCS can be used as constraints in the *constraint-block*.

Only constraints specified in the constraint block are honored. Any rand mode specified on the class members is ignored when `std::randomize()` is called with the given class member.

The `pre_randomize()` and `post-randomize()` tasks are not called when `std::randomize()` is used within a class member function.

The `std::` prefix must be explicitly specified for the `randomize()` call.

The `std::randomize()` function is supported in VCS. Files containing `std::randomize()` calls can be compiled with `vlogan`.

The function using `std::randomize()` can be declared in a task inside a package that can be imported into modules and programs.

## Example

```
module M;
 bit[11:0] addr;
 integer data;

 function bit genAddrData();
 bit success;
 success = std::randomize(addr, data);
 return success;
 endfunction

 function bit genConstrainedAddrData();
 bit success;
 success = std::randomize(addr, data)
 with {addr > 1000; addr + data < 20000;};
 return success;
 endfunction

endmodule
```

The `genAddrData` function uses `std::randomize(addr, data)` to assign random values to `addr` and `data` variables. The `std::randomize()` function randomizes any variables that are visible in the scope.

The `getConstrainedAddrData()` function uses `std::randomize(addr, data)` to assign random values to `addr` and `data` variables.

## SystemVerilog Bounded Queues

A bounded queue is a queue limited to a fixed number of items. For example:

```
bit q[$:255];
a bit queue whose maximum size is 257 bits
```

```
int q[$:5000];
an int queue whose maximum size is 50001
```

This section explains how bounded queues work in certain operations.

```
q1 = q2;
```

This is a bounded queue assignment. VCS copies the items in q2 into q1 until q1 is full or until all the items in q2 are copied into q1. The bound number of items in the queues remain the same as declared.

```
q.push_front(new_item)
```

If you add a new item to the front of a full bounded queue, VCS deletes the last item in the back of the queue.

```
q.push_back(new_item)
```

If the bounded queue is full, a new item cannot be added to the back of the queue and the queue remains the same.

```
q1 === q2
```

The behavior of a bounded queue comparison is same as an as an unbounded queue, that is, the bound sizes should be the same when the two bounded queues are equal.

### **Limitation for SystemVerilog Bounded Queues**

Bounded queues are not supported in constraints.

## **wait() Statement with a Static Class Member Variable**

A wait statement with a static class member variable is now supported. Consider the following example:

```
class foo;
 static bit is_true = 0;
 task my_task();
 fork
 begin
 #20;
 is_true = 1;
 end
 begin
 wait(is_true == 1);
 $display("%0d: is_true is now %0d", $time, is_true);
 end
 join
 endtask: my_task
endclass: foo
```

```
program automatic main;
 foo foo_i;
 initial begin
 foo_i = new();
 foo_i.my_task();
 end
endprogram: main
```

## Support for Consistent Behavior of Class Static Properties

VCS supports access and usage of class static members in structural contexts including continuous assign and force.

Consider the following test.sv test case that uses class static member in a continuous assign statement:

```
class mymode;
 static int mode;
endclass

class testbench
 mymode p;
 function new();
 p = new();
 endfunction
endclass

module test()

 testbench tb = new();
 int local_mode;

 assign local_mode = tb.p.mode; //class static variable in
continous assign
 always@(local_mode) $display($time, " local_mode changed: module
tb.p = %d", local_mode);
 always@(tb.p.mode) $display($time, " tb.p.mode changed: module tb.p =
%d", tb.p.mode);
 initial begin
 #1;
 $display($time, " local_mode = %d, tb.p.mode = %d", local_mode,
tb.p.mode);
 #10;
 $display($time, " local_mode = %d, tb.p.mode = %d", local_mode,
tb.p.mode);
 end
endmodule

module top;

 test t();


```

```
testbench tb = new();
initial begin
 tb.p.mode = 0;
 #10;
 tb.p.mode = 10;
 #10;
end

endmodule
```

The expected results after simulating this test case is that the `local_mode` local variable must be equal to `tb.p.mode` at all times.

The results are as follows:

```
1 local_mode = 0, tb.p.mode = 0
10 tb.p.mode changed: module tb.p = 10
10 local_mode changed: module tb.p = 10
11 local_mode = 10, tb.p.mode = 10
```

## Parameters and Local Parameters in Classes

You can include parameters and local parameters (localparams) in classes. For example:

```
class cls;
 localparam int Lp = 10;
 parameter int P = 5;
endclass
```

## SystemVerilog Math Functions

Verilog defines math functions that behave the same as their corresponding math functions in C. These functions are as follows:

|                         |                   |
|-------------------------|-------------------|
| <code>\$ln(x)</code>    | Natural logarithm |
| <code>\$log10(x)</code> | Decimal logarithm |
| <code>\$exp(x)</code>   | Exponential       |
| <code>\$sqrt(x)</code>  | Square root       |
| <code>\$pow(x,y)</code> | $x^{**}y$         |
| <code>\$floor(x)</code> | Floor             |
| <code>\$ceil(x)</code>  | Ceiling           |
| <code>\$sin(x)</code>   | Sine              |

|                             |                                         |
|-----------------------------|-----------------------------------------|
| <code>\$cos (x)</code>      | Cosine                                  |
| <code>\$tan (x)</code>      | Tangent                                 |
| <code>\$asin (x)</code>     | Arc-sine                                |
| <code>\$acos (x)</code>     | Arc-cosine                              |
| <code>\$atan (x)</code>     | Arc-tangent                             |
| <code>\$atan2 (x, y)</code> | Arc-tangent of $x/y$                    |
| <code>\$hypot (x, y)</code> | $\sqrt{x^2+y^2}$                        |
| <code>\$sinh (x)</code>     | Hyperbolic sine                         |
| <code>\$cosh (x)</code>     | Hyperbolic cosine                       |
| <code>\$tanh (x)</code>     | Hyperbolic tangent                      |
| <code>\$asinh (x)</code>    | Arc-hyperbolic sine                     |
| <code>\$acosh (x)</code>    | Arc-hyperbolic cosine                   |
| <code>\$atanh (x)</code>    | Arc-hyperbolic tangent                  |
| <code>\$clog2 (n)</code>    | Ceiling of log base 2 of n (as integer) |

## Streaming Operators

Streaming operators that can be applied to any bit-stream data types consists of the following:

- Any integral, packed, or string type
- Unpacked arrays, structures, or class of the above types
- Dynamically sized arrays (dynamic, associative, or queues) of any of the above types

## Packing (Used on RHS)

### Primitive Operation

```
expr_target = {>>|<< slice{expr_1, expr_2, ..., expr_n }}
```

The `expr_target` and `expr_i` can be any primary expressions of any streamed data types.

The slice determines the size of each block measured in bits. If specified, it may be either a constant integral expression, or a simple type.

The << or >> determines the order in which blocks of data are streamed.

### Streaming Concatenation

```
expr_target = {>>slice1 {expr1, expr2, {<< slice2{expr3, expr4}}}}
```

## Unpacking (Used on LHS)

### Primitive operation

```
{>>|<< slice{expr_1, expr_2, ..., expr_n } } = expr_src;
```

If the unpacked operation includes unbounded dynamically sized types, the process is greedy. The first dynamically sized item is resized to accept all the available data (excluding subsequent fixed sized items) in the stream. Any remaining dynamically sized items are left empty.

### Streaming Concatenation

```
{>>slice1 {expr1, expr2, {<< slice2{expr3, expr4}} } = expr_src;
```

## Packing and Unpacking

```
{>>|<< slice_target{target_1, target_2, ..., target_n } } = {>>|<< slice_src{src_1, src_2, ..., src_n } };
```

## Propagation and force Statement

Any operand (either dynamic or not) in the stream can be propagated and forced/released correctly.

## Error Conditions

It has the following error conditions:

- Compile-time error for associative arrays as assignment target
- Runtime error for any null class handles in packing and unpacking operations

## Structures with Streaming Operators

Although the whole structure is not allowed in the stream, any structure members and excluded sub-structures could be used as an operand of both packing and unpacking operations.

For example:

```
s1 = {>>{expr_1, expr_2, .., expr_n}} //invalid
s1.data = {>>{expr_1, expr_2, expr_n}} //valid
```

## Support for with Expression

VCS supports the `with` expression with streaming operator.

The syntax of the `with` expression defined in the IEEE SystemVerilog LRM Std 1800™-2012 is as follows:

```
stream_expression ::= expression [with [array_range_expression]]
array_range_expression ::= expression
| expression : expression
| expression +: expression
| expression -: expression
```

### Semantics

The feature has the following semantics:

1. You can set the array expression range within the `with` construct to an integral type or to an expression that evaluates to an integral type. You cannot use other types.

For example,

```
function int eval(int a);
 eval = a;
endfunction

{ >> { target with [0 : 7]} } = 31; //case 1
{ >> { target with [0 : eval(31)]} } = 31; //case 2
```

For case 2, the function evaluates to an integral type. Hence, this case is allowed.

2. You can set the expression before the `with` construct to any single unpacked dimensional array that includes a queue.

For example,

```
bit target[][];
```

Usage:

```
{ >> { target with [0:2]} } = data;
```

This case is illegal, as the expression before the `with` expression has multiple unpacked dimension.

Usage:

```
{ >> { target[3] with [0:2]} } = data;
```

However, this case is legal as the expression before the `with` expression has a single unpacked dimension.

3. The expression within the `with` construct is evaluated immediately before its corresponding array is streamed (packed or unpacked). Thus, the expression can refer to data that are unpacked by the same operator but before the array.

For example,

```
{ >> { a, target with [a:0], b } } = data;
```

In this case, assuming target is a single dimension unpacked array and the value of a is 2 before streaming and the value of a is 4 after streaming, the value of the index should be 4.

4. When you use the `with` expression within the context of an unpack operation and the array is a variable-sized array, the `with` expression must be resized to accommodate the expression range.

For example,

```
Size of data = 32, size of a = 8, sizeof b = 8
bit target[];
{ >> { a, target, b } } = data;
```

In this case, a total of 16 bits are allocated to the target.

```
{ >> { a, target with [0: 7], b } } = data;
```

However, in this case only 8 bits are allocated to the target. Therefore, you can limit the data allocation to any dynamic type using the `with` expression.

5. If the array is a fixed-size array and the expression range evaluates to a range outside the extent of the array, only the range that lies within the array is unpacked and an error is issued.

For example,

```
bit target[0:3];
{ >> {target with [0:7]} } = data;
```

This is illegal, as it exceeds the array bound.

6. If the range expression evaluates to a range smaller than the extent of the array (fixed or variable size), only the specified items are unpacked into the designated array locations. The rest of the array is unmodified.
7. When you use the `with` expression within the context of a pack (on RHS), it behaves in the same way as an array slice.

For example,

```
data = { >> {target with [a:0]} };
```

The above example can be interpreted directly as `{>> {target[value_of_a:0]}{}}`. In array slice, variable index might not be allowed in all the cases, but with the `with` expression, you can provide a variable index.

8. You can set the array range expression within the `with` construct to be of integral type and it evaluates to values that lie within the bounds of a fixed-size array or to a positive value for dynamic arrays or queues.

---

## Constant Functions in Generate Blocks

Calls to constant user-defined functions can be included in `generate` blocks.

As stated in the IEEE SystemVerilog LRM Std 1800™-2012, you can use these constant functions to build complex calculations. The standard also establishes things that cannot be in a user-defined function for it to operate as a constant function. For example, a constant function cannot have an `output`, `inout`, or `ref` argument, they cannot contain statements that schedule events after the function has returned its value, or contain the `fork` construct. There are more than a few other requirements.

A call to a constant function can occur in a `generate` block but the `generate` block cannot contain a definition or declaration of a constant function.

[Figure 163](#) contains a module definition that includes a `generate` block and a user-defined function that qualifies as a constant function. The `generate` block contains a call to this constant function.

*Figure 163 Calling a Constant Function in a Generate Block*

```

module interoceter();
parameter adrs_width = 4;
generate
 genvar dim1;
 genvar dim2;
 for (dim1 = 1; dim1 <= adrs_width; dim1 = dim1 + 1)
 begin : outer_floop
 for (dim2 = 0; dim2 < adrs_width; dim2 = dim2 + const_func(dim1))
 begin : inner_floop
 reg [2:0] P;
 end : inner_floop
 end : outer_floop
 endgenerate

 function integer const_func;
 input [31:0] lwrdf; integer intgr0;
 begin : func_main
 if (lwrdf > 0)
 begin : ifloop
 lwrdf = {lwrdf >> 1};
 intgr0 = 1;
 while (lwrdf > 0)
 begin : wloop
 lwrdf = {lwrdf >> 1};
 intgr0 = {intgr0 << 1};
 end : wloop
 end : ifloop
 else
 intgr0 = 0; // return 0 when lwrdf <= 0
 const_func = intgr0;
 end : func_main
 endfunction
 endmodule

```

## Support for Aggregate Methods in Constraints Using the “with” Construct

Aggregate methods in constraint blocks using the `with` construct have two variants, as shown in the following code example:

```

byte arr[3] = { 10, 20, 30 };
class C;
 rand int x1;

```

```

rand int x2;
rand int x3;
rand int x4;

constraint cons {
 // Newly implemented variant
 x1 == arr.sum() with (item * item);
 x2 == arr.sum(x) with (x + x);

 // Previously implemented variant
 // Supported in older releases
 x3 == arr.sum() with (arr[item.index] * arr[item.index]);
 x4 == arr.sum(x) with (arr[x.index] + arr[x.index]);
}
endclass

```

The first variant is implemented. For a discussion and examples of aggregate methods in constraints using the `with` construct, see IEEE SystemVerilog LRM Std 1800™-2012 Section 7.12.4 “Iterator index querying”.

As specified in the standard, the entire `with` expression must be in parentheses.

## Debugging During Initialization SystemVerilog Static Functions and Tasks in Module Definitions

You can tell VCS to enable UCLI debugging when initialization begins for static SystemVerilog tasks and functions in module definitions with the `-ucli=init` runtime option and keyword argument.

This debugging capability enables you also to set breakpoints during initialization.

If you omit the `=init` keyword argument and just enter the `-ucli` runtime option, the UCLI begins after initialization and you cannot debug inside static initialization routines during initialization.

**Note:**

- Debugging static SystemVerilog tasks and functions in program blocks during initialization does not require the `=init` keyword argument.
- This feature does not apply to VHDL or SystemC code.

When you enable this debugging, VCS displays the following prompt indicating that the UCLI is in the initialization phase:

init%

When initialization ends, the UCLI returns to its usual prompt:

ucli%

During the initialization, the `run` UCLI command with the `0` argument (`run 0`), or the `-nba` or `-delta` options runs VCS until initialization ends. As usual, after initialization, the `run 0` command and argument runs the simulation until the end of the current simulation time.

During initialization the following restrictions apply:

- UCLI commands that alter the simulation state, such as a `force` command create an error condition.
- Attaching or configuring Cbug, or in other ways enabling C, C++, or SystemC debugging during initialization is an error condition.
- The following UCLI commands are not allowed during initialization:

session management commands: `save` and `restore`

signal and variable commands: `force`, `release`, and `call`

The signal value and memory dump specification commands: `memory -read/-write` and `dump`

The coverage commands: `coverage` and `assertion`

## Example

Consider the following code example:

```
module mod1;
class C;
 static int I=F();
 static function int F();
 logic log1;
 begin
 log1 = 1;
 $display("%m log1=%0b",log1);
 $display("In function F");
 F = 10;
 end
 endfunction
endclass
endmodule
```

If you simulate this example, with the `-ucli` runtime option, you see the following:

```
Command: simv =ucli
Chronologic VCS simulator copyright 1991-year
Contains Synopsys proprietary information.
Compiler version version-number; Runtime version version-number;
simulation-start-date-time
mod1.\C::F log1=1
In function F
 V C S S i m u l a t i o n R e p o r t
Time: 0
```

```
CPU Time: 0.510 seconds; Data structure size: 0.0Mb
simulation-ends-day-date-time
```

VCS executed the \$display tasks right away and the simulation immediately ran to completion.

If you simulate this example, with the -ucli=init runtime option and keyword argument, you see the following:

```
Command: simv -ucli=init
Chronologic VCS simulator copyright 1991-year
Contains Synopsys proprietary information.
Compiler version version-number; Runtime version version-number;
simulation-start-date-time
init%
```

Note that VCS has not executed the \$display system tasks yet and the prompt is init%.

You can set a breakpoint, for example:

```
init% stop -in \C::F
1
```

To run through the initialization phase:

```
init% run 0
Stop point #1 @ 0 s;
init%
```

The breakpoint halts VCS.

If you run the simulation up to the end of the initialization phase with the run 0 UCLI command again, you see the following:

```
init% run 0
mod1.\C::F log1=1
In function F
ucli%
```

Now VCS executes the \$display system tasks and changes the prompt to ucli%.

## Explicit External Constraint Blocks

External constraint blocks are constraint blocks, also called the constraint bodies, that are outside of a class and at the same hierarchical level of that class. You enable them with external constraint prototypes in the class.

There are two forms of external constraint prototypes:

- explicit — where you include the `extern` keyword in the prototype.
- implicit — where you omit the `extern` keyword in the prototype.

The explicit form is implemented in this release.

The following code example shows these two forms of external constraint prototypes.

```
class Class1;
 rand int int1,int2;
 constraint imp_ext_cnstr_proto1; // implicit form
 extern constraint exp_ext_cnstr_proto2; // explicit form
...
endclass
```

The external constraint block, or body for these prototypes must be at the same hierarchical level as the class and follow the class definition.

The following are external constraint blocks or bodies for these external constraint prototypes:

```
constraint Class1::imp_ext_cnstr_proto1 {
 int1 inside {0, [3:5], [7:31]};
constraint Class1::exp_ext_cnstr_proto2 {
 int2 dist {100 := 1, 101 := 2};}
```

Besides the `extern` keyword, the difference between the implicit and explicit forms is how VCS responds when the external constraint block or body for a prototype is missing:

- With the implicit form, VCS handles a missing external constraint block as an empty constraint block. This is not an error condition and VCS just outputs a warning message. For example:

```
Warning-[BCNACMBP] Missing constraint definition
doc_example.sv, 6
prog, "constraint imp_ext_cnstr_proto1;"
The constraint imp_ext_cnstr_proto1 declared in the class Class1 is
not defined.
Provide a definition of the constraint body imp_ext_cnstr_proto1 or
remove the constraint declaration imp_ext_cnstr_proto1 from the class
declaration Class1.
```

An empty constraint block is same as the following:

```
constraint imp_ext_cnstr_proto1 { };
```

With a missing external constraint block for the implicit form, VCS continues to compile or elaborate and generates the simv executable because it is not an error condition. If you do not notice the warning message you might expect to see the missing constraint block constraining the values of the random variables.

- With the explicit form, a missing external constraint block is an error condition. For example:

```
Error-[SV_MEECD] Missing explicit external constraint def
doc_example.sv, 7
prog, "constraint exp_ext_cnstr_proto2;"

The explicit external constraint 'exp_ext_cnstr_proto2' declared in
the class 'Class1' is not defined.

Provide a definition of the constraint body 'exp_ext_cnstr_proto2'
or remove the explicit external constraint declaration
'exp_ext_cnstr_proto2' from the
class declaration 'Class1'.
```

With a missing external constraint block for the explicit form, VCS does not compile or elaborate because it is an error condition.

## Using an Empty Constraint Block

You can use the implicit form of a constraint prototype, without the corresponding constraint block in a subclass to remove a constraint from a base class. For example:

```
module top;
class C;
rand int x;
constraint protoC_1 { x < 5; }
constraint protoC_2 { x > 3; }
endclass

class CD extends C;
rand int y;
constraint protoC_1; // removing this constraint in
// this subclass
constraint protoCD_1 { x < 6; } // applying a new constraint
// on x
endclass

C ci = new;
CD cdi = new;
int res1;
int res2;

initial begin
repeat (20) begin
 res1 = ci.randomize(); // here x can have value 4 only
 res2 = cdi.randomize(); // here x can have values 4 and 5
 if ((res1 == 1) && (res2 == 1))
 $display("niru>> ci.x=%d cdi.x=%d",ci.x, cdi.x);
end
end

endmodule
```

## Generate Constructs in Program Blocks

Generate constructs are now supported not just in modules but also in program blocks.

These constructs are described in The IEEE Verilog LRM Std 1364-2005 in the following sections:

12.4 Generate constructs

12.4.1 Loop generate constructs

12.4.2 Conditional generate constructs

The following are examples of these constructs in a program block:

```
program prog;
...
generate
 reg reg1;
endgenerate

if (1) logic log1;

genvar gv1;
for(gv1=1; gv1<10; gv1++) logic log2;

case (param1)
 0 : logic log3;
 ...
endcase

endprogram
```

The first is a generate region, specified with the `generate` and `endgenerate` keywords inside a program block:

```
generate
 reg reg1;
endgenerate
```

The second is a conditional generate construct with the `if` keyword:

```
if (1) logic log1;
```

The third is a generate loop variable declared with the `genvar` keyword, followed by a `for` loop for that variable:

```
genvar gv1;
for(gv1=1; gv1<10; gv1++) logic log2;
```

The fourth is a generate case construct:

```
case (param1)
 0 : logic log3;
 ...
endcase
```

## Error Condition for Using a Genvar Variable Outside of its Generate Block

A `genvar` variable declared in local scope of a `generate` block that is used outside that block is an error condition. The following code example shows this error condition:

```
module test;
 generate
 for (genvar i = 0; i < 1; i++)
 begin
 a1: assert final (1);
 end
 endgenerate
 generate
 for (i = 0; i < 1; i++)
 begin
 a1: assert final (1);
 end
 endgenerate
endmodule
```

Compiling/elaborating this example with the following command line:

```
% vcs generate.sv -sverilog -assert svaext
```

Results in the following error message:

```
Error-[IND] Identifier not declared
generate.sv, 9
 Identifier 'i' has not been declared yet. If this error is not
 expected,
 please check if you have set `default_netttype to none.

1 error
```

To fix this error, declare `genvar i` in module scope.

## Randomizing Unpacked Structs

You can now randomize members of an unpacked struct. You can do this in the following ways:

- use the scope randomize method `std::randomize()`
- use the class randomize method `randomize()`

You can also:

- disable and re-enable randomization in an unpacked struct with the `rand_mode()` method.
- use in-line random variable control to specify the randomized variables with an argument to the `randomize()` method.

## Using the Scope Randomize Method `std::randomize()`

The following example illustrates using this method:

*Example 79 First Example of the Scope Randomize Method `std::randomize()`*

```
module test();

typedef struct {
 bit [1:0] b1;
 integer i1;
} ST1;

ST1 st1;

initial
repeat (4)
begin
 std::randomize(st1);
 #10 $display("\n\n\t at %0t", $time);
 $display("\t st1.b1 is %0d", st1.b1);
 $display("\t st1.i1 is %0d", st1.i1);
end

endmodule
```

This example randomizes struct instance `st1`. The `$display` system tasks display the following:

```
at 10
 st1.b1 is 2
 st1.i1 is 1474208060
```

```
at 20
```

```
st1.b1 is 1
st1.i1 is 816460770
```

```
at 30
st1.b1 is 3
st1.i1 is -1179418145
```

```
at 40
st1.b1 is 0
st1.i1 is -719881993
```

The following is another code example that randomizes members of an unpacked struct and uses constraints:

*Example 80 Second Example of the Scope Randomize Method std::randomize()*

```
module test;
 typedef struct {
 rand byte aa;
 byte bb;
 } ST;

 ST st;
 bit [3:0] c;

 initial begin
 std::randomize(st.bb); // std randomization on a
 // struct member
 std::randomize(st) with { st.aa > 10; };
 // support st.aa in with block
 std::randomize(c,st) with { st.aa > c; };
 $display("\n\n\t at %0t",$time);
 $display("\t st.aa is %0d",st.aa);
 $display("\t st.bb is %0d",st.bb);
 $display("\t bit c is %0d",c);
 end
endmodule
```

The \$display system task displays the following:

```
at 0
st.aa is 121
st.bb is -9
bit c is 0
```

*Example 81 Third Example of the Scope Randomize Method std::randomize()*

```
module test;
 typedef struct {
 byte a0;
 byte b0;
 }
```

```

} ST0;
typedef struct {
 byte aa;
 ST0 st0;
} ST_NONE;

typedef struct {
 rand byte aa;
 byte bb;
} ST_PART;

typedef struct {
 rand byte aa;
 randc byte bb;
} ST_ALL;

ST_NONE st;
ST_PART st1;
ST_ALL st2;

initial begin
 repeat (5) begin
 // random variables: st.aa st.st0.a0 st.st0.b0
 std::randomize(st);

 // random variables: st1.aa st.bb
 std::randomize(st1) with {st1.aa>st1.bb;};

 // random variables: st2.aa st2.bb
 std::randomize(st2);

 $display("st %p",st);
 $display("st1 %p",st1);
 $display("st2 %p",st2);
 end
end

endmodule

```

This example randomizes unpacked struct instance st1. The \$display system task displays the following:

```

st '{aa:54, st0:'{a0:60, b0:125}}
st1 '{aa:-125, bb:-126}
st2 '{aa:-9, bb:-90}
st '{aa:27, st0:'{a0:-75, b0:-6}}
st1 '{aa:-37, bb:-47}
st2 '{aa:-106, bb:49}
st '{aa:-60, st0:'{a0:-86, b0:-60}}
st1 '{aa:-71, bb:-103}
st2 '{aa:-120, bb:-15}
st '{aa:44, st0:'{a0:-50, b0:5}}
st1 '{aa:-69, bb:-96}

```

```
st2 '{aa:96, bb:95}
st '{aa:122, st0:'{a0:-94, b0:-16}}
st1 '{aa:-2, bb:-63}
st2 '{aa:18, bb:-12}
```

## Using the Class Randomize Method randomize()

The following example illustrates using this method.

*Example 82 The Class Randomize Method randomize()*

```
module test();

typedef struct {
 rand bit [1:0] b1;
 rand integer il;
} ST1;

class CC;
 rand ST1 st1;
endclass

CC cc = new;

initial
repeat (4)
begin
 cc.randomize();
#10 $display("\n\n\t at %0t", $time);
$display("\t cc.st1.b1 is %0d", cc.st1.b1);
$display("\t cc.st1.il is %0d", cc.st1.il);
end

endmodule
```

This example randomizes instance `cc` of class `CC` that contains unpacked struct `ST`. The `$display` system task displays the following:

```
at 10
cc.st1.b1 is 3
cc.st1.il is -1241023056

at 20
cc.st1.b1 is 3
cc.st1.il is -1877783293

at 30
cc.st1.b1 is 1
cc.st1.il is 629780255
```

```
at 40
cc.st1.b1 is 3
cc.st1.i1 is 469272579
```

Here is another code example:

*Example 83 Another Example of the Class Randomize Method randomize()*

```
module test;

typedef struct {
 bit[3:0] c;
 randc bit[1:0] d;
} ST0;

typedef struct {
 rand bit[5:0] a;
 rand bit[5:0] b;
 rand ST0 st0;
 bit [5:0] e;
} ST;

class CC;
 rand ST st;
endclass

CC cc = new;

initial begin
repeat (10) begin
 // random variables: cc.st.a cc.st.b and cc.st.st0.d
 // state variables: cc.st.e and cc.st.st0.c
 cc.randomize() with { st.a<10 ; st.b>10; st.a+st.b==64; };

 $display("st %p",cc.st);
end
end

endmodule
```

This example randomizes class instance `cc` according to the constraint that follows the `with` keyword. The `$display` system task displays the following:

```
st '{a:'h7, b:'h39, st0:{c:'h0, d:'h0}, e:'h0}
st '{a:'h8, b:'h38, st0:{c:'h0, d:'h1}, e:'h0}
st '{a:'h1, b:'h3f, st0:{c:'h0, d:'h3}, e:'h0}
st '{a:'h1, b:'h3f, st0:{c:'h0, d:'h2}, e:'h0}
st '{a:'h1, b:'h3f, st0:{c:'h0, d:'h0}, e:'h0}
st '{a:'h8, b:'h38, st0:{c:'h0, d:'h1}, e:'h0}
st '{a:'h9, b:'h37, st0:{c:'h0, d:'h2}, e:'h0}
st '{a:'h9, b:'h37, st0:{c:'h0, d:'h3}, e:'h0}
st '{a:'h7, b:'h39, st0:{c:'h0, d:'h3}, e:'h0}
st '{a:'h8, b:'h38, st0:{c:'h0, d:'h1}, e:'h0}
```

## Disabling and Re-enabling Randomization

You can disable and re-enable randomization in an unpacked struct with the `rand_mode()` method.

*Example 84 Disabling and Re-enabling Randomization with the rand\_mode() Method*

```
module test();

typedef struct {
 rand integer il;
} ST1;

class CC;
 rand ST1 st1;
endclass

CC cc = new;

initial
repeat (10)
begin
 cc.randomize();
 #10 $display("\n\t at %0t", $time);
 $display("\t cc.st1.il is %0d", cc.st1.il);
end

initial
begin
 #55 cc.rand_mode(0);
 #20 cc.rand_mode(1);
end

endmodule
```

In this example, the `rand_mode()` method, with its arguments, disables and re-enables randomization in class instance `cc`. The `$display` system task displays the following:

```
at 10
cc.st1.il is -902462825

at 20
cc.st1.il is -1241023056

at 30
cc.st1.il is 69704603

at 40
cc.st1.il is -1877783293

at 50
cc.st1.il is -795611063
```

```

at 60
cc.st1.i1 is 629780255

at 70
cc.st1.i1 is 629780255

at 80
cc.st1.i1 is 629780255

at 90
cc.st1.i1 is 1347943271

at 100
cc.st1.i1 is 469272579

```

In this example randomization is disabled at simulation time 55 and re-enabled at simulation time 75, enabling new random values at simulation time 90.

The following is another code example:

*Example 85 Another Example of Disabling and Re-enabling Randomization with the rand\_mode() Method*

```

module test;

typedef struct {
 bit[3:0] c;
 randc bit[1:0] d;
} ST0;

typedef struct {
 rand bit[5:0] a;
 rand bit[5:0] b;
 rand ST0 st0;
 bit [5:0] e;
} ST;

class CC;
 rand ST st;
 rand bit[2:0] n1;
endclass

CC cc = new;

initial
begin
 cc.st.rand_mode(0);
 repeat (10)
 begin
 // random variables: cc.n1
 // state variables: all members of cc.st
 cc.randomize();
 $display("turn off st %p , cc.n1 %b",

```

```

 cc.st,cc.n1);
 end
 cc.st.rand_mode(1);
 cc.st.st0.rand_mode(0);
repeat (10)
begin
 // random variables: cc.n1 cc.st.a cc.st.b
 // state variables: cc.st.e cc.st.st0.c cc.st.st0.d
 cc.randomize();
 $display("turn off st.st0 %p , cc.n1 %b",
 cc.st,cc.n1);
 end
 cc.st.st0.rand_mode(1);
end

endmodule

```

In this example the `rand_mode()` method disables randomization in unpacked struct instance `cc.st.st0` and then re-enables it. The `$display` system task displays the following:

```

turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 111
turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 000
turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 011
turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 011
turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 001
turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 111
turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 111
turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 011
turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 001
turn off st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 100
turn off st.st0 '{a:'h39, b:'h17, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 010
turn off st.st0 '{a:'h26, b:'h1f, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 001
turn off st.st0 '{a:'h9, b:'h3, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 010
turn off st.st0 '{a:'h23, b:'he, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 101
turn off st.st0 '{a:'h21, b:'h18, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 000
turn off st.st0 '{a:'h34, b:'h1d, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 001
turn off st.st0 '{a:'h2f, b:'h27, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 011
turn off st.st0 '{a:'h2f, b:'h17, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 100
turn off st.st0 '{a:'hd, b:'h34, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 010
turn off st.st0 '{a:'h27, b:'h11, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 010

```

## Using In-Line Random Variable Control

The following example illustrates the usage of in-line random variable control to specify the randomized variables with an argument to the `randomize()` method.

### *Example 86 In-line Random Variable Control*

```

module test();

typedef struct {
 rand integer il;

```

```

} ST1;

typedef struct {
 rand integer i1;
} ST2;

class CC;
 rand ST1 st1;
 rand ST2 st2;
endclass

CC cc = new;

initial
begin
 #10 cc.randomize();
 $display("\n\t at sim time %0t", $time);
 $display("\t cc.st1.i1 is %0d", cc.st1.i1);
 $display("\t cc.st2.i1 is %0d", cc.st2.i1);
 #10 cc.randomize(st1);
 $display("\n\t at sim time %0t", $time);
 $display("\t cc.st1.i1 is %0d", cc.st1.i1);
 $display("\t cc.st2.i1 is %0d", cc.st2.i1);
 #10 cc.randomize(null);
 $display("\n\t at sim time %0t", $time);
 $display("\t cc.st1.i1 is %0d", cc.st1.i1);
 $display("\t cc.st2.i1 is %0d", cc.st2.i1);
 #10 cc.randomize(st2);
 $display("\n\t at sim time %0t", $time);
 $display("\t cc.st1.i1 is %0d", cc.st1.i1);
 $display("\t cc.st2.i1 is %0d", cc.st2.i1);
end

endmodule

```

This example supplies the `randomize()` method with arguments for unpacked struct instances `st1` and `st2` and the `null` keyword.

1. At simulation time 20, randomization is limited to `st1`.
2. At simulation time 30, randomization is turned off.
3. At simulation time 40, randomization is limited to `st2`.

The `$display` system task displays the following:

```

at sim time 10
cc.st1.i1 is -902462825
cc.st2.i1 is -1241023056

at sim time 20
cc.st1.i1 is 69704603
cc.st2.i1 is -1241023056

```

```

at sim time 30
cc.st1.il is 69704603
cc.st2.il is -1241023056

at sim time 40
cc.st1.il is 69704603
cc.st2.il is -1877783293

```

At simulation 20, a new random value is in `st1` but not `st2`.

At simulation time 30, there are no new random values.

At simulation time 40, a new random value is in `st2` but not `st1`.

Here is another code example:

*Example 87 Another Example of In-line Random Variable Control*

```

module test;

typedef struct {
 bit[3:0] c;
 randc bit[1:0] d;
} ST0;

typedef struct {
 rand bit[5:0] a;
 rand bit[5:0] b;
 rand ST0 st0;
 bit [5:0] e;
} ST;

class CC;
 ST st;
 rand bit[2:0] n1;
endclass

CC cc = new;

initial begin
 // random variables: cc.n1
 // state variables: all members of cc.st
repeat (5) begin
 cc.randomize();
 $display("default st %p , cc.n1 %b",cc.st,cc.n1);
end

 // random variables: cc.st.a cc.st.b cc.st.st0.d
 // state variables: cc.n1 cc.st.e cc.st.st0.c
repeat (5) begin
 cc.randomize(st);
 $display("inline st %p , cc.n1 %b",cc.st,cc.n1);

```

```
end
end
endmodule
```

In this example the `randomize()` method is called without an argument and then with the `st` struct instance argument. The `$display` system tasks display the following:

```
default st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 111
default st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 000
default st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 011
default st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 011
default st '{a:'h0, b:'h0, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 001
inline st '{a:'h1f, b:'h27, st0:{c:'h0, d:'h0}, e:'h0} , cc.n1 001
inline st '{a:'h11, b:'h34, st0:{c:'h0, d:'h1}, e:'h0} , cc.n1 001
inline st '{a:'h17, b:'h2a, st0:{c:'h0, d:'h2}, e:'h0} , cc.n1 001
inline st '{a:'h1f, b:'h9, st0:{c:'h0, d:'h3}, e:'h0} , cc.n1 001
inline st '{a:'h3, b:'h12, st0:{c:'h0, d:'h3}, e:'h0} , cc.n1 001
```

VCS executes the second `$display` system task after it executes the `randomize()` method with the `st` argument.

## Limitation

Random class objects as members of an unpacked struct are not yet implemented (NYI). For example:

```
module test;

class CC0;
 rand int a;
endclass

typedef struct {
 rand bit[5:0] a;
 rand bit[5:0] b;
 rand CC0 cc0; // this is not allowed in this release
} ST;

endmodule
```

## Using a Package in a SystemVerilog Module, Program, and Interface Header

Importing from a package to a module, program, or interface by including the package in the module, program, or interface header is now implemented.

This technique of importing from a package is described in the IEEE SystemVerilog LRM Std 1800™-2012 Section 26.4 “Using packages in module headers”.

The primary purpose of this syntax and usage is to enable you to import names in the parameter list or port list, without importing the package into the enclosing scope (`$unit`).

To illustrate this technique, you should import from a package into a module definition and then into a program definition, as shown in [Example 88](#) and [Example 89](#). This technique is also implemented for importing from a package to an interface.

*Example 88 Importing a Package in a Module Header*

```
package my_pkg;
 typedef reg [3:0] my_type1;
 typedef int my_type2;
endpackage

module my_module import my_pkg::*;
 (input my_type1 a, output my_type2 z);
M
endmodule
```

In [Example 88](#), the design objects declared in the `my_pkg` package are imported into the `my_module` module with the `import` keyword followed by the name of the package. Use the wildcard `*` (asterisk) to specify importing all design objects in the package.

[Example 89](#) shows importing from packages in a program header.

*Example 89 Importing Packages in a Program Header*

```
package pack1;
 typedef struct {
 real r1;
 } struct1;
 typedef enum bit {H,T} bool_sds;
endpackage:pack1

package pack3;
 integer int1=0;
endpackage: pack3

program prog1 import pack1::struct1,pack3::*;
 (output out1,out2);
M
endprogram: prog1
```

The header of the `prog1` program includes the `import` keyword followed by the `pack1` and `pack3` packages. Import the `struct1` structure from `pack1` into the `prog1` program. Then import all the design objects in `pack3` into the program using the wildcard `*` (asterisk).

**Note:**

VCS does not support using the same name for a package and a module within a compilation unit.

---

## Disabling DPI Tasks and Functions

Based on the IEEE Standard for SystemVerilog 1800™-2012 section 35.9, VCS now provides the support for disable statement to terminate the activity of a task or a named block that is currently executing a DPI call.

When a DPI import subroutine is disabled, the DPI-C code is made to follow a simple disable protocol. The protocol allows the C code to perform any necessary resource cleanup. An import DPI routine is in the disabled state when a disable statement targets either the import DPI routine or a parent of the calling chain to the import DPI routine for disabling. An imported subroutine can enter the disabled state immediately after the return of a call to an exported subroutine, and it adheres to the following disable protocol:

- A disabled import DPI routine shall acknowledge within a program that they have been disabled by calling the `svAckDisabledState()` API function.
- A subroutine can determine that it is in the disabled state by calling the API function `svIsDisabledState()`.
- When an imported task returns due to a disable, it shall return a value of 1. Otherwise, it shall return 0.
- Once an imported subroutine enters the disabled state, it is illegal for the current function call to make any further calls to exported subroutines.

If any of the protocol items are not correctly followed, a fatal simulation error is issued.

## Use Model

The following switch must be passed to VCS to enable this functionality:

`-dpi_lrm_task`

In the case of UUM flow, the switch must be passed both at the `vlogan` and `vcs` steps.

---

## Support for Overriding Parameter Values through Configuration

VCS supports overriding the parameter value through a configuration as defined in the SystemVerilog LRM. Configurations can be used either to override parameter values that are declared within a design or to override parameter values for a specific instance of a design.

## Example

The following example illustrates overriding of parameter values through configuration:

*Example 90 Example of parameter overriding through configuration*

```

config cfg;
 design rtlLib.top;
 default liblist rtlLib;
 localparam LP = 19;
 instance top.B1 use #(.P(LP)); // assign 19 to top.B1.P
 instance top.B2 use #(.P(3)); // assign 3 to top.B2.P
 instance top.B3 use #(.P()); // assign its default value to top.B3.P
 cell bot use #(.P(10)); // assign 10 to rest of instances of bot
endconfig : cfg

module top;
 bot #(11) B1(); // instance parameter value being override
 // inside configuration
 bot B2();
 bot B3();
 bot B4();
 bot B5();
 bot B6();
 defparam top.B4.P = 20; // defparam specified parameter value
 //being override inside configuration
endmodule

module bot;
 parameter P = 9;
 initial $display("%m", P);
endmodule

```

## Precedence Override Rules

Parameter overriding during elaboration is determined in the following order of priority (highest to lowest):

1. Parameter overriding from VCS elaboration command line (-pvalue)
2. Parameter overriding through a configuration using instance and cell rules
3. defparam using hierarchical path names
4. Instance based overriding

**Note:**

If multiple instance and cell rules are used, VCS applies the rule that appears first in configuration. It ignores multiple rules and generates a warning message.

## Limitations

This feature has the following limitations:

- Cross-module references (XMRs) for parameter overriding is not supported.
- For VCS, parameter overriding rules are not supported if the design hierarchy crosses the VHDL boundary.

## Support for Inclusion of Dynamic Types in Sensitivity List

VCS ignores the dynamic types while processing the implicit sensitivity lists of `always @*`, `always_comb`, and `always_latch` procedural blocks. Therefore, it issues the following warning:

Warning-[IDTS] Ignoring dynamic type sensitivity

<source\_file>, <line\_no> "obj"

Dynamic types used in `always_comb`, `always_latch`, `always (@*)` will be ignored for the inferred sensitivity list.

VCS supports the dynamic types in the implicit sensitivity list of `always @*` block as specified in the Section 9.2 of the *IEEE Standard SystemVerilog Specification 1800-2012*.

The following usages of arrays would be processed for implicit sensitivity list of `always@*`:

- Selects of dynamic arrays, queues, and associative arrays
- Built-in method calls of:
  - Dynamic array method `size()`
  - Queue built-in method `size()`
  - Associative array methods `num()`, `size()`, `exists()`
- If the base type of array is an unpacked structure, then the array would be ignored for the sensitivity list.

This support is enabled under the following compile-time option:

`-ntb_opts sensitive_dyn.`

## Usage Example

The following example shows the case of usage of dynamic array in an `always @*` procedural block:

*Example 91 Usage of Dynamic Array*

```
module top;
 int da1[];
 int da2[];
 int i = 0;
 always @(*) begin
 $display($stime,, "triggered, da1 size is %d", da1.size());
 da2[i] = da1[i];
 end
 initial begin
 da1 = new[2]; //trigger the always block
 da2 = new[2];
 #1;
 da1[0] = 10; //trigger the always block
 #1;
 i++; //trigger the always block
 #1;
 da1[1] = 20; //trigger the always block
 #0 $display($stime,, "da = %p", da2);
 end
endmodule
```

Execute the following command:

```
% vcs -sverilog -ntb_opts sensitive_dyn test.v -R
```

Output:

```
0 triggered, da1 size is 2
1 triggered, da1 size is 2
2 triggered, da1 size is 2
3 triggered, da1 size is 2
3 da = '{10, 20}
```

## Support for Assignment Pattern Expression in Non-Assignment Like Context

VCS supports assignment pattern expression that is used in aggregate data types such as unpacked arrays or unpacked structures in non-assignment like contexts, as specified in the Section 10.9 of the *IEEE Standard SystemVerilog LRM Std. 1800-2012*.

An assignment pattern can be used to construct or deconstruct a structure or an array by prefixing the pattern with the name of a data type to form an assignment pattern expression. An assignment pattern expression has a self-determined data type.

## Usage Example

The following example shows the case of usage of assignment pattern expression:

*Example 92 Usage of Assignment Pattern Expression*

```
module test;
 byte payload[];
 typedef byte byte_array[];
 initial begin
 payload = '{8'hab, 8'hcd, 8'hcd, 8'hef};
 if (payload[1:2] == byte_array'{8'hcd, 8'hcd})begin
 $display("Matches");
 end
 else
 $display("Failed");
 end
endmodule
```

## Limitations

The following are the limitations for this feature:

- Assignment pattern expressions are not supported for associative arrays.
- Multi-dimensional arrays of unpacked structure which do not have dynamic types as its members are not supported.
- Assignment pattern expressions that are used in left side expression are not supported.
- Selects of multi-dimensional arrays which result in to another multi-dimensional array are not supported as a member of assignment pattern expression.
- Assignment pattern expressions to a nested structure can have only constants, assignment pattern, and objects/variables of other data types as its members. Any other complex expressions such as function calls, expressions involving operators and so on are not supported.

## User-Defined Nettypes

SystemVerilog 1800-2012 supports a user-defined net datatype. You declare it with the `nettype` keyword. You can write a user-defined function to be the resolution function for the new net datatype. This resolution function is usually optional.

See section 6.6.7 “User-defined nettypes” in the SystemVerilog (1800-2012) LRM.

## The Resolution Function

The user-defined resolution function has the following characteristics:

- The return type of the function must match the datatype of the nettype.
- The function uses a dynamic array as a single input argument, but the function cannot modify (resize or write) a dynamic array input argument.
- The function is automatic and preserves no state information.
- Parameterized variants of class methods are possible.
- The SystemVerilog (1800-2012) LRM specifies that the resolution function can be a static class method, but there is a limitation against this in the I-2014.03 release.

**Note:**

A user-defined nettype that has multiple drivers must have a resolution function.

## Limitations

The following features are not supported in the declaration of user-defined nettypes:

- NLP activities on nettype nets or interconnect are ignored
- Nettypes across language boundaries (SystemC) and partial elaboration boundaries
- A static class method cannot be used as a resolution function.
- Struct members with unpacked dimensions
- Active driver analysis and VCD or EVCD dumping
- Resolution function that is an interface function or a function within a class scope
- \$deposit system task or VPI for deposit with nettypes

## Example of User-Defined Nettype

In the following code example package `n pkg` includes user-defined nettype `myNet` with a resolution function `resFn`.

```
package npkg;
typedef struct {
 real I;
 real V;
 logic active;
}wShape;
function automatic wShape resFn(wShape dr[]);
 int count = 0;
 begin
```

```

resFn.I = 0.0;
resFn.V = 0.0;
resFn.active = 0;
foreach (dr[i]) begin
 resFn.I += dr[i].I;
 resFn.V += dr[i].V;
 resFn.active = 1;
 count++;
end
resFn.I = resFn.I/count;
resFn.V = resFn.V/count;
end
endfunction
nettype wShape myNet with resFn;
endpackage

module model;
import npkg::*;
myNet w;
wShape v1, v2;
assign w = v1;
assign w = v2;
initial begin
 v1.active = 1;
 v2.active = 1;
 v1.I = 2.5;
 v1.V = 1.5;
 v2.V = 3.5;
 v2.I = 4.5;
end
endmodule

```

## Example of User-Defined Nettype in Arrays

In the following code example package `npkg` includes user-defined nettype `myNet` with a resolution function `resFn`.

```

`timescale 1 ns / 1 fs

package elec_pkg;

typedef real resistance_type;
typedef real voltage_type;
typedef struct {
 voltage_type v = 0;
 resistance_type r = 1e21;
} electrical_type;

function automatic electrical_type Kirchhoff_Law (input
 electrical_type s[]);
 Kirchhoff_Law = '{0.0, 0.0};
 foreach (s[i]) begin

```

```

 Kirchhoff_Law.v += s[i].v;
Kirchhoff_Law.r += s[i].r;
 end
endfunction

nettype electrical_type electrical with Kirchhoff_Law;
 nettype voltage_type voltage;
endpackage

module nfc_g2_bias_reader(o_ibg40u_tx3_2v5,o_atst_2v5);
 import elec_pkg::*;

 output voltage o_ibg40u_tx3_2v5;
 input electrical o_atst_2v5[3:0];

 electrical_type e1,e2;

 assign o_ibg40u_tx3_2v5 = o_atst_2v5[0].v;

 assign o_atst_2v5[0] = e2;
 assign o_atst_2v5[0] = e1;

 initial begin
 e1 = '{0.0,0.0};
 e2 = '{0.0,0.0};
 #5 e1 = '{1.2,1.2};
 e2 = '{1.4,1.4};
 #5 e1 = '{5.0,5.0};
 e2 = '{5.0,5.0};
 #5 $finish();
 end

initial $monitor($time,"Voltage is %g V ",o_ibg40u_tx3_2v5);

endmodule

```

## Example of Nettype MDAs of Type Real

The following code example includes nettype MDAs of type `real`.

```

typedef real T;
function automatic real resfn(input real drv[]);
 resfn=0;
 foreach(drv[i])
 resfn = resfn + drv[i];
endfunction

nettype T myNet with resfn;
module top;
 myNet mem[4][5];
 T v;
 assign mem[1][4] = v;

```

```
initial $display ($time,mem[1][4]);
endmodule
```

## Example of Nettype MDAs of Type Unpacked Struct

The following code example includes nettype MDAs of type unpacked struct.

```
typedef struct {logic r; real v;} T;
function automatic T resfn(input T drv[]);
 resfn='{0,0};
 foreach(drv[i]) begin
 resfn.r = resfn.r + drv[i].r;
 resfn.v = resfn.v + drv[i].v;
 end
endfunction

nettype T myNet with resfn;
module top;
 myNet mem[4][5];
 T v;
 assign mem[1][4] = v;
 initial $display ($time,,mem[1][4].r);
 initial begin
 //mem[0][3]= v;
 #10 v = '{1, 1.1};
 #10 v = '{1, 2.1};
 #10 v = '{1, 3.1};
 end
endmodule
```

## Support for Connecting Nettypes through Tranif Gates

VCS also allows you to connect user-defined nettypes through tranif gates. Therefore, you can have a back-to-back connection in a transistor network.

### *Example 93 Example of Nettypes Connected through Tranif Gates*

```
function automatic real resfn (input real drv[]);
 real temp;
 resfn=0;
 temp=drv[0];
 foreach(drv[i]) begin
 if(drv[i] > temp)
 temp=drv[i];
 end
 resfn=temp;
endfunction

nettype real real_net with resfn;

module test;
 bit ctrl;
```

```
real_net a[3:0];
assign a[0] = 1;
assign a[2] = 2;
tranif0 trf(a[1],a[0],ctrl);
endmodule
```

### **Key Points to Note**

- Nettypes that are connected through tranif gates must have a resolution function.
- The control signal of tranif gates must be of 2-state data type.
- The nets connected to the two bidirectional pins of a tranif gate must be of the same nettype.

### **Limitations**

The feature has the following limitations:

- Nettypes of data types other than real and unpacked structs are not supported.
- Force or release on nettypes connected through tran gates are not supported.

### **Generic Interconnect Nets**

SystemVerilog 1800-2012 supports interconnect nets. Interconnects are very generic without any type associated with them and can be connected to any datatype through a port. VCS treats them as connection points and automatically associates an appropriate type to this connection point, depending on what it's connected to.

You declare them with the `interconnect` keyword. Interconnect nets are generic in that they don't have a datatype. They are limited to connecting ports (on modules and interfaces) and terminals (on primitives including UDPs). You can't use them on the left or right side of procedural statements or in continuous assignment statements or procedural continuous assignment statements.

See section 6.6.8 “Generic interconnect” in the SystemVerilog (1800-2012) LRM.

You enable generic interconnect nets with the `-sv_interconnect` compile-time option.

## Limitations

The following features are not supported in the declaration of interconnect nets:

- Interconnects across language boundaries (SystemC) and partial elaboration boundaries
- Interconnect as part of interface or virtual interfaces
- Verilog-AMS is not supported in presence of interconnect

## Support for Associative Array With Unpacked Structure as Key

VCS enables you to use unpacked structure as key to associative array as specified in the Section 7.8.5 of the *IEEE Standards for System Verilog LRM 1800-2012*.

The following example shows how you can use associative array with an index of unpacked structure:

### *Example 94 Associative Array With Unpacked Structure as Key*

```
module test;
 typedef struct {byte B; int I[*];} Unpkt;
 int arr[Unpkt];
 Unpkt st0,st1;
 initial begin
 st0 = '{1,'{0:1,1:1}};
 st1 = '{2,'{0:2,1:2}};
 #1 arr[st0] = 32'd11;
 arr[st1] = 32'd22;
 $display("%p",arr);
 end
endmodule
```

## Limitation

The following is the limitation of this feature:

Assignment pattern expression for key is not supported.

## Specifying a SystemVerilog Keyword Set by LRM Version at Command Line

You can use the `-sv` compile-time option to specify a version of the IEEE Standard for SystemVerilog (also referred as LRM) in a simulation. This version is the language version used by the VCS and VCS compiler and the keyword set version applied to the design. Furthermore, the `-sv` option enables the SystemVerilog mode that is currently specified by the `-sverilog` option.

In every new version of the LRM, new language keywords can be introduced. These new keywords create potential compilation errors in designs or IPs that use the keywords as identifiers. VCS support the SystemVerilog '`begin_keywords`' and '`end_keywords`' compiler directives as defined in the IEEE Standard for SystemVerilog (IEEE Std P1800-2012), section 22.14 '`begin_keywords`', '`end_keywords`'. You can use these directives to specify a version of the LRM and apply that version's keyword set to the source code encapsulated by the directives. You can use the directives to resolve compilation errors caused by keywords introduced in newer versions of the LRM.

In addition, you can use the `-sv` compile-time option to specify a default LRM version of keyword set to be used in a compilation. This default LRM version applies to the source code that is not encapsulated by the SystemVerilog '`begin_keywords`' and '`end_keywords`' compiler directives.

## The `-sv` Compile-Time Option

You can use the `-sv` compile-time option to specify a default LRM version of keyword set to be used in a compilation. This default LRM version applies to source code that is not encapsulated by the SystemVerilog '`begin_keywords`' and '`end_keywords`' compiler directives,

The syntax of the `-sv` option is:

`-sv=SystemVerilog_LRM_Version`

where,

`SystemVerilog_LRM_Version`

Specifies a version of *IEEE Standard for SystemVerilog* and applies that version's keyword set to the design. The supported LRM versions are:

- 2012
- 2009
- 2005

### Note:

An error message is issued if the `-sv` option is specified with `-sverilog`.

For example,

**test.v**

```
class C;
 rand int implements;

constraint c1 {
 implements inside {2,3,5};
```

```

 }
endclass

module m;
 C c=new();
 initial begin
 c.implements = 10;
 $display("implements = %0d\n", c.implements);
 end
endmodule

```

### Compilation command

```
%vcs -sv=2009 test.v
```

The keyword `implements` was introduced in the *IEEE Standard for SystemVerilog* (IEEE Std P1800-2012). In the above example, the `-sv` option specifies the LRM version IEEE Std P1800-2009 at compile time. As a result, `implements` is handled as a regular identifier instead of a keyword. With the `-sv=2012` option, a compilation error is issued.

## The `begin\_keywords and `end\_keywords Compiler Directives

VCS supports the SystemVerilog `'begin_keywords` and `'end_keywords` compiler directives as defined in the *IEEE Standard for SystemVerilog* (IEEE Std P1800-2012), section 22.14 `'begin_keywords`, `'end_keywords`. You can use these directives to specify a version of the LRM and apply that version's keyword set to the source code encapsulated by the directives. You can use the directives to resolve compilation errors caused by keywords introduced in newer versions of the LRM.

For example,

```

`begin_keywords "1800-2009"
class C;
 rand int implements;

 constraint c1 {
 implements inside {2,3,5};
 }
endclass

module m;
 C c=new();
 initial begin
 c.implements = 10;
 $display("implements = %0d\n", c.implements);
 end
endmodule
`end_keywords

```

The keyword `implements` was introduced in the *IEEE Standard for SystemVerilog* (IEEE Std P1800-2012). In the above example, the `'begin_keywords` and `'end_keywords`

directives specify the LRM version IEEE Std P1800-2009. As a result, `implements` is handled as a regular identifier instead of a keyword. Without the directives, a compilation error is issued.

## Support for `.triggered` Property with Clocking Block Name

As defined in *IEEE SystemVerilog LRM*, clocking event of a clocking block is available directly by using the clocking block name, regardless of the actual clocking event used to declare the clocking block.

Besides using expression `@(clocking_block_name)` to access clocking block event, VCS also allows the usage of clocking block name with `.triggered` property.

**Note:**

The feature allows the usage of clocking block name with the `.triggered` property only. No other event like properties, such as assignment, passing-through-port, and so on are allowed.

## Usage Examples

The following examples show how you can use clocking block name with `.triggered` property.

*Example 95 Example of Usage of Clocking Block with `.triggered` Property*

```
interface ifc(input clk);
 ...
 logic [7:0] data;
 clocking cb@(posedge clk);
 ...
 output data;
 endclocking
 modport producer(clocking cb);
endinterface
class driver;
 virtual ifc.producer vif;
 task drive;
 wait(vif.cb.triggered);
 vif.cb.data <= pkt.data;
 ...
 endtask
 task check;

```

```

if (vif.cb.triggered || pkt.size == len) begin
 ...
end
endtask
endclass

```

## Support for Intra Assignment Delay With Non-Blocking Assignments in Program Block

VCS supports intra assignment delay with non-blocking assignment statements in program block with the `-ntb_opts re_nba_sched` compile time option, as specified in the section *9.4.5 of the IEEE Standard System Verilog Std. 1800-2012*.

### Limitations

This option is not supported in NLP flow.

## Support for Array Query Functions

This section contains enhancements related to Array Query functions. This section contains the following topics:

- [Array Query Functions With Associative Array](#)

### Array Query Functions With Associative Array

VCS supports the following array query function on associative array whose index is of integral type:

- `$left`
- `$right`
- `$high`
- `$low`
- `$increment`

The width of the index type must meet the following conditions:

- `width > 0` and `width <= 32` for signed type.
- `width > 0` and `width <=31` for unsigned type.

With associative array, these query functions returns the following values:

- `$left` returns 0.
- `$right` returns the highest possible index value.

- `$low` returns the lowest currently allocated index value but will return 'x if there are no elements currently allocated.
- `$high` returns the largest currently allocated index value but returns 'x if there are no elements currently allocated.
- `$increment` returns -1.

The following is an example of array query function with associative array. The new supported values are highlighted in the example.

*Example 96 Usage of Array Queries for Associative Array*

```
module top();
 int arr[int];
 initial begin
 $display($left(arr));
 end
endmodule
```

## Limitations

The limitations of this feature are as follows:

- Query functions `$high` and `$low` are not supported inside wait statement and event control statement.
- Array query function calls are not supported within with clause of coverbin.

## Support for Nested Randsequence

VCS supports randsequence inside randsequence with the `-nested_randseq` compile time option.

### Use Model

Use the following option at compile time:

```
% vcs -sverilog <filename.sv> -nested_randseq
```

#### Note:

In vcs three step flow, use this option both at the `vlogan` and at `vcs` steps.

### Usage Example

The following example shows the support for nested randsequence.

*Example 97 Nested randsequence usage*

```
module top;
 bit topa;
```

```

final begin
 topa = 0;
 randsequence(main)
 main : first|second done;
 first : add |dec;
 second : pop | push;
 done : { topa = 1; };
 add : { topa = 2; };
 pop : { topa = 3; };
 push : { topa = 4; };
 dec : {
 randsequence(main1) // Nested randsequence
 main1 : first;
 first : { topa = 5; };
 endsequence };
 endsequence
 if(!topa) $display("Failed");
 else $display("Passed");
end
endmodule

```

## Support for Typed Constructor Call as an Argument to Task or Function

VCS supports passing typed constructor call as an input type argument to task or function.

### Usage Example

Consider the following test case:

#### *Example 98 test.v*

```

module top;
class A;
 int x;
 function new(int v1);
 this.x=v1;
 endfunction
endclass
class B extends A;
 int y;
 function new(int v2);
 super.new(v2);
 this.y=v2+1;
 endfunction
endclass
function void init(A a);
 $display("%p",a);
endfunction
initial begin

```

```
 init(B::new(5));
end
endmodule
```

The output generated is as follows:

```
'{x:5, y:6}
```

## Limitations

The limitations of this feature are as follows:

- Typed constructor call is not supported in the default value of task or function argument.

## Support for Select of Unpacked Structure Array as an Argument to Task or Function

VCS supports passing select of unpacked structure array (a structure which contains unpacked array) as a ref type argument to task or function.

### Usage Example

Consider the following test case:

*Example 99 test.v*

```
typedef struct {int unsigned a; int unsigned b; int unsigned c;} st1_t;
typedef struct { st1_t x[2];} st2_t;
typedef struct {st2_t y[2]; } st3_t;
module tb;
 st3_t st_var;
 initial begin
 set_y(st_var.y[0]);
 $display(st_var.y[0]);
 end
 function set_y(ref st2_t e);
 e.x[0] = '{9,8,7};
 e.x[1] = '{9,8,7};
 endfunction
endmodule
```

The output generated is as follows:

```
'{x:'{{a:9, b:8, c:7}, '{a:9, b:8, c:7}} }
```

### Limitation

The limitation of this feature is as follows:

- Force on high-conn of a Function ref port is not supported.

## Support for Function Returning Unpacked Structure in Conditional Operator in a Continuous Assignment Statement

VCS supports function returning unpacked structure in conditional operator in RHS of a continuous assignment statement.

### Note:

The use of static or impure functions in the continuous assignment statement is not recommended as it may result in synthesis simulation mismatches.

### Usage Example

Consider the following testcase:

#### *Example 100 test.sv*

```
module test;
 typedef struct {int intVar;} stDef;
 int intVar1, intVar2;
 stDef retVal;
 bit trueOrFalse;
 function stDef funcDef1(input int arg1);
 stDef stObj1;
 stObj1 = '{arg1};
 return stObj1;
 endfunction:funcDef1
 function stDef funcDef2(input int arg2);
 stDef stObj2;
 stObj2 = '{arg2};
 return stObj2;
 endfunction:funcDef2
 assign retVal = trueOrFalse ? funcDef1(intVar1) : funcDef2(intVar2);
 always @(retVal.intVar)
 $display("retVal.intVar at time %0t = %d", $time, retVal.intVar);
 initial begin
 #1; intVar1 = 17; intVar2 = 25;
 #1; trueOrFalse = 1;
 #1; intVar1 = 01; intVar2 = 05;
 end
endmodule:test
```

Run the design using the following command:

```
% vcs -sverilog test.sv -full64 -R
```

The output generated is as follows:

```
retVal.intVar at time 1 = 25
retVal.intVar at time 2 = 17
retVal.intVar at time 3 = 1
```

## Support for Built-in Method Which Returns Work Library Name

The built-in method `get_worklib_name()` returns the work library name of the given class handle. The format of the string returned by the built-in method is as follows:

```
<lib name>.<package name>. <class name>
```

The syntax for `get_worklib_name()` is as follows:

```
string = <class_handle>.get_worklib_name();
```

where `class_handle` is the name of the class handle.

## Usage Example

Consider the following example:

### *Example 101*

```
pkg1.v

package pkg;
class c1;
int a;
endclass
endpackage

pkg2.v

package pkg2;
class c12 #(int i = 24) extends pkg::c1;
int a2;
endclass
endpackage

top.v

module top;
import pkg::*;
import pkg2::*;
c1 ci = new();
string s;

initial begin
 s = ci.get_worklib_name ();
 $display ("%s", s);
 ci = c12#(48)::new();
 s = ci.get_worklib_name ();
 $display ("%s", s);
end
endmodule
```

`synopsys_sim.setup` file:

```
WORK > DEFAULT
DEFAULT : ./work
LIB_A : ./lib_a
LIB_B : ./lib_b
```

Run the design using the following commands:

```
% vlogan -full164 -sverilog pkg1.v -work lib_a
% vlogan -full164 -sverilog pkg2.v -work lib_b
% vlogan -full164 -sverilog top.v -work work
% vcs top -full164 -R
```

The output generated is as follows:

```
\LIB_A.pkg .cl
\LIB_B.pkg2 .\cl2#(48)
```

## Limitations

Following are the limitations of this feature:

- If a parametrized class is defined in `generate` block, then the string returned by `get_worklib_name ()` for such class handle may not include generate block name in the hierarchy.
- If a class is extended using the AOP construct, then the string returned by `get_worklib_name ()` for such class handle does not include library name or package name in Partition Compile flow.
- If a class is defined inside the scope which contains any protected code, then the string returned by `get_worklib_name ()` for such class handle does not include the class name.
- If the string output returned by the built-in method is equal to or more than 4095 characters, then the returned string is truncated and the last three characters in the string are displayed as “...”.

## Support for Function call in Clocking Block Hierarchical Expression

VCS supports the user-defined function call in clocking block input hierarchical expression as described in *SystemVerilog LRM Std 1800™-2012 Section 14.5, Hierarchical expressions*.

As per LRM, any expression assigned to a signal in a clocking `input` or `inout` declaration shall be an expression that would be legal for connection to a module's input port.

The clocking block input hierarchical expression behaves the same as a continuous assignment to the clockvar.

## Usage Example

Consider the following example:

*Example 102 test.sv*

```
module top;
virtual class check;
 static function bit func1(input logic t);
 if($isunknown(t))
 $display("Error at %0t: X/Z loss during 4-state to 2-state
assignment in testbench",$realtime);
 return t;
 endfunction : func1
endclass : check
logic logic_var;
bit bit_var;
logic clk=0;
clocking cb @(posedge clk);
 input bit_var = check::func1(logic_var);
endclocking
always_ff @(cb) begin
 $display("%0t - value of logic_var is %0b value of bit_var is %0b",
$realtime,logic_var,cb.bit_var);
 case (logic_var)
 1'bx: logic_var <= 'z;
 1'b0: logic_var <= 0;
 1'b1: logic_var <= 1;
 default: logic_var <= 'x;
 endcase
end
initial forever #5 clk = !clk;
initial #45 $finish;
endmodule
```

Run the design using the following commands:

```
% vlogan -full64 -sverilog test.sv
% simv
```

The output generated is as follows:

```
Error at 0: X/Z loss during 4-state to 2-state assignment in testbench
5 - value of logic_var is x value of bit_var is 0
Error at 5: X/Z loss during 4-state to 2-state assignment in testbench
15 - value of logic_var is z value of bit_var is 0
25 - value of logic_var is 0 value of bit_var is 0
35 - value of logic_var is 1 value of bit_var is 1
Error at 35: X/Z loss during 4-state to 2-state assignment in testbench
```

## Limitations

Following are the limitations of this feature:

- The function call is not allowed in clocking block output hierarchical expression.
- The hierarchical expression cannot contain multiple function calls.
- Function used in hierarchical expression cannot return dynamic data such as string, class object, and so on.
- A function used in hierarchical expression cannot have arguments of dynamic type.

---

## Extensions to SystemVerilog

This section contains descriptions of Synopsys enhancements to SystemVerilog. This section contains the following topics:

- [Unique/Priority Case/IF Final Semantic Enhancements \(-xlrm uniq\\_prior\\_final Compile-Time Option\)](#)
- [Single-Sized Packed Dimension Extension](#)
- [Covariant Virtual Function Return Types](#)
- [Self Instance of a Virtual Interface](#)
- [Support for Shuffle Method for Multi-Dimensional Arrays](#)
- [Enhanced Clocking Block Behavior When Skew is negedge/posedge](#)
- [Support for Slice of String Variable](#)
- [Support for Randomization of Floating Point Variables](#)
- [Support for Unpacked Array Concatenation in the HighConn of Inout Port](#)
- [Support for Index Locator Methods for Multi-Dimensional Arrays](#)
- [Support for String Method Substr\(\) with a Single Argument](#)
- [Support for Assignment Pattern with Single-Bit Scalar Nets or Variables](#)
- [Support for Reading Value of any Clock Variable](#)

---

## Unique/Priority Case/IF Final Semantic Enhancements (-xlrn uniq\_prior\_final Compile-Time Option)

The behavior of the compliance checking keywords `unique` and `priority` for `case` and for `if...else if...else` selection statements as defined in the Conditional if-else statement Section 12.4 “Conditional if-else statement” in some cases can cause spurious warnings when used inside a module’s continuous assignment or `always` block. By default, VCS evaluate compliance with `unique` or `priority` on every update to the selection statement input.

To force `unique` and `priority` to evaluate compliance only on the stable and final value of the selection input at the end of a simulation timestep, VCS now provides a `-xlrn uniq_prior_final` compile-time option.

This can be useful, for example, when `always_comb` might trigger several times within a simulation time slot while its input values are getting stabilized. The `case` statements can get executed several times during the same time slot if it is valid for combinational blocks. While going through intermediate transitions, the `case` statement might get values that violate the `unique` or `priority` property and cause VCS to report multiple runtime warnings. When it is undesirable to receive intermediate warnings, the `-xlrn uniq_prior_final` compile time option can be used to evaluate compliance for only the final stable value of the input.

## Using Unique/Priority Case/If with Always Block or Continuous Assign

`-xlrn uniq_prior_final` behavior only applies to the use of `unique` and `priority` keywords when selection statements are used inside a module’s continuous assignment statements or `always` blocks. The option is not applicable to selection statements in a program block or an initial block.

The following two examples illustrate this behavior:

### *Example 103 unique case statement at the same timestep*

```
//test.sv:
module top;
reg cond;
bit [7:0] a = 0,b, v1, v2;
always_comb begin
if (cond) begin
unique case (a)
 v1: begin b = 0; $display(" Executing Case
with cond value 1 "); end
 v2: begin b = 1; $display(" Executing Case
with cond value 1 "); end
 endcase
end
```

```

else begin
 unique case (a)
 v1: begin b = 0; $display(" Executing Case
 with cond value 0 "); end
 v2: begin b = 1; $display(" Executing Case
 cond value 0 "); end
 endcase
end
end

initial begin
#1 cond = 1;
a=a+4; v1=4; v2=4;
$display("\n TIME %0d ns : cond value %0b, a value %0d",
 $time, cond, a);
#0 cond = 0;
a=a+1; v1++; v2++;
$display("\n TIME %0d ns: cond value %0b, a value %0d",
 $time, cond, a);
end
endmodule

```

### **Simulation output without -xlrn uniq\_prior\_final:**

```

%> vcs -sverilog test.sv -R

Executing Case with condition value 0
RT Warning: More than one conditions match in 'unique case' statement.
 "unique_case.sv", line 12, for top.
 Line 13 & 14 are overlapping at time 0.
Executing Case with cond value 0
RT Warning: More than one conditions match in 'unique case' statement.
 "unique_case.sv", line 12, for top.
 Line 13 & 14 are overlapping at time 0.

TIME 1 ns : cond value 1, a value 4
Executing Case with cond value 1
RT Warning: More than one conditions match in 'unique case' statement.
 "unique_case.sv", line 6, for top.
 Line 7 & 8 are overlapping at time 1.

TIME 1 ns: cond value 0, a value 5
Executing Case with cond value 0
RT Warning: More than one conditions match in 'unique case' statement.
 "unique_case.sv", line 12, for top.
 Line 13 & 14 are overlapping at time 1.

```

### **Simulation output with -xlrn uniq\_prior\_final compile-time option:**

```

%> vcs -sverilog test.sv -xlrn uniq_prior_final -R
Executing Case with cond value 0:
RT Warning: More than one conditions match in 'unique case' statement.
 "unique_case.sv", line 12, for top.

```

```

Line 13 & 14 are overlapping at time 0.

TIME 1 ns : cond value 1, a value 4
Executing Case with cond value 1

TIME 1 ns: cond value 0, a value 5
Executing Case with cond value 0
RT Warning: More than one conditions match in 'unique case' statement.
 "unique_case.sv", line 12, for top.
Line 13 & 14 are overlapping at time 1.

```

**Example 104 unique if inside always\_comb**

```

//test.sv
module top;
reg cond;
bit [7:0] a = 0,b;
always_comb begin

unique if (a == 0 || a == 1) $display ("A is 0 or 1");
else if (a == 2) $display ("A is 2");

end

initial begin
#100;
a = 1;
#100 a = 2;
#100 a = 3;
#0 a++;
#0 a++;
#0 a++;
#10 $finish;

end

endmodule

```

**Simulation output without -xIrm:**

```

%> vcs -sverilog test.sv -R

A is 0 or 1
A is 0 or 1
A is 0 or 1
A is 2
RT Warning: No condition matches in 'unique if' statement.
 "unique_if.sv", line 5, for top, at time 300.
RT Warning: No condition matches in 'unique if' statement.
 "unique_if.sv", line 5, for top, at time 300.
RT Warning: No condition matches in 'unique if' statement.
 "unique_if.sv", line 5, for top, at time 300.
RT Warning: No condition matches in 'unique if' statement.
 "unique_if.sv", line 5, for top, at time 300.

```

```
"unique_if.sv", line 5, for top, at time 300.
$finish called from file "unique_if.sv", line 17.
```

### Simulation output with -xlmr uniq\_prior\_final:

```
%> vcs -sverilog test.sv -xlmr uniq_prior_final -R
A is 0 or 1
A is 0 or 1
A is 0 or 1
A is 2
RT Warning: No condition matches in 'unique if' statement.
"unique_if.sv", line 5, for top, at time 300.
$finish called from file "unique_if.sv", line 17.
```

## Using Unique/Priority Inside a Function

With this enhancement, if unique/priority case statement is used inside a function, VCS not only points to the current case statement but also provides a complete stack trace of where the function is called. The following example illustrate this behavior:

#### *Example 105 unique case used with nested loop inside function*

```
//test.sv
module top;
 int i,j;
 reg [1:0][2:0] a, b, c;
 bit flag;

 function foo;
 for (int i=0; i<2; i++)
 for (int j=0; j<3; j++)
 unique case (a[i][j])
 0: b[i][j] = 1'b0;
 1: b[i][j] = c[i][j];
 endcase
 endfunction : foo

 always_comb begin
 for(i=0; i<4; i++) begin
 if (i==2)
 foo();
 end
 end

 initial begin
 a = 6'b00x011;
 end

endmodule : top
```

### Simulation output without the -xlr option:

```
%> vcs -sverilog test.sv -R
RT Warning: No condition matches in 'unique case' statement.
"unique_case_inside_func.sv", line 8, for top.foo, at time 0.

RT Warning: No condition matches in 'unique case' statement.
"unique_case_inside_func.sv", line 8, for top.foo, at time 0.
```

### Simulation output with -xlr uniq\_prior\_final:

```
%> vcs -sverilog test.sv -xlr uniq_prior_final -R
RT Warning: No condition matches in 'unique case' statement.

"unique_case_inside_func.sv", line 8, for top.foo, at time 0.
#0 in foo at unique_case_inside_func.sv:8
#1 in loop with j= 0 at unique_case_inside_func.sv:7
#2 in loop with i= 1 at unique_case_inside_func.sv:6
#3 in top at unique_case_inside_func.sv:16
#4 in loop with i= 2 at unique_case_inside_func.sv:14
```

#### Note:

The following limitations must be noted while using the `-xlr uniq_prior_final` option for loop indices:

- It must be written in `for` statement. The `while` and `do...while` are not supported.
- The loop bounds must be the compile-time constants.
- `for(i= lsb; i<msb; i++)`
- Here, `lsb` and `msb` must be compile-time constant, or becomes constant when upper loops get unrolled.
- No other drivers of the loop variable must be in the loop body.

VCS also supports `unique/prior final` in a `for` loop that cannot be unrolled at compile time. For example, if you have a `for` loop whose range could not be determined at compile time and if there are errors during the last evaluation of such a `for` loop, VCS still reports the error. However, loop index information will not be provided. Even if multiple failures occur in different iterations, VCS reports only the last one.

#### Note:

Use `unique/priority case/if` statement only inside the `always` block, continuous assign, or inside a function. If you use it in other places, the final semantic is ignored.

## System Tasks to Control Warning Messages

Two system tasks `$uniq_prior_checkon` and `$uniq_prior_checkoff` enable you to switch on/off runtime warning messages for unique/priority if/case statements. The following example illustrates the use model of these tasks to ignore violations:

*Example 106 System tasks to control warning messages*

```
//test.sv
module m;
 bit sel, v1, v2;

//Disable this initial block to display all RT warning messages
initial
begin
$display($time, " Priority checker OFF\n");
$uniq_prior_checkoff();
#1;
$display($time, " Priority checker ON\n");
$uniq_prior_checkon();
end

initial
begin
//violation with this set of values (warning disabled)
sel = 1'b1;
v1 = 1'b1;
v2 = 1'b1;
#1;
//violation with this set of values (warning enabled)
sel = 1'b0;
v1 = 1'b0;
v2 = 1'b0;
#1;
end
always_comb begin
unique case(sel)
 v1: $display($time, " Hello");
 v2: $display($time, " World");
endcase
end
endmodule
```

### Simulation Output:

```
%> vcs -sverilog test.sv -R

0 Priority checker OFF
0 Hello
0 Hello
1 Priority checker ON
1 Hello
RT Warning: More than one conditions match in 'unique case' statement.
```

```
"system_task_control_warning.sv", line 28, for m.
Line 29 & 30 are overlapping at time 1.
```

## LRM Compliant Behavior for the `atohex` Method

As per *IEEE Std 1800-2012 SystemVerilog LRM*, the width of the return value of the `atohex` method should be 32-bit. VCS incorrectly returns context-specific width (width as per LHS) for the `atohex` method. The behavior of the `atohex` method is made LRM compliant.

The following table shows the behavior difference.

| Testcase                                                                                                                                                                              | Old Return Value | New Return Value                                   |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------|----------------------------------------------------|
| <pre>module test; string txt_64; bit [63:0] b_64; initial begin     txt_64 =         "00000abcd88000000a";     b_64 = txt_64.atohex();     \$display("%h", b_64); end endmodule</pre> | 000abcd88000000a | ffffffff8000000a(32-bit value with sign extension) |

You can use the `-xlrn atox_context_width` option at compile time to revert back to the old behavior.

## Controlling Runtime Warning Messages Generated Using Unique/Priority If Constructs

The `-xlrn uniq_prior_observed` compile-time option allows the runtime warning message to appear in the observed region of the current time step in compliance with the IEEE SystemVerilog LRM Std 1800™-2012 Section Section 12.4.2.1, which states the following:

“A unique, unique0, or priority violation check is evaluated at the time the statement is executed, but violation reporting is deferred until the Observed region of the current time step. The violation reporting can be controlled by using assertion control system tasks.

*Once a violation is detected, a pending violation report is scheduled in the Observed region of the current time step. It is scheduled on a violation report queue associated with the currently executing process.*”

The failure messages are reported in the following order:

1. All `assert #0` and RT warning messages are interleaved among themselves.
2. All `assert final` messages are reported after step1 is done.

Consider the following messages:

```

`timescale 1ns/1ns
module top;
reg a,b,c;
always_comb begin
 Unique_case: unique case(1)
 a:$display("matched a at time ",,$time);
 b:$display("matched b at time ",,$time);
 default:;
 endcase
end
initial begin
 a = 0; b = 0;
 #5 a = 1; b = 1;
end
initial
$monitor("\n %t value of a : %b b : %b",$time,a,b);
Assobserved: assert #0 ($onehot({a,b}));
Assfinal: assert final ($onehot({a,b}));
p p1();
endmodule

program p();
initial begin
 #5 top.b = 0;
 #10 $finish;
end
endprogram

```

In this example, in the program block. `#5 top.b=0` is assigned a value 0 at time 0 and a value 1 at time 5. Similarly, in the initial block, `a` and `b` are assigned a value 0 at time 0 and a value 1 at time 5.

You can compile this example using the following command:

```
% vcs -sverilog -assert svaext -xlrn uniq_prior_observed test.v
% simv
```

VCS generates the following output:

```
Warning-[RT-MTOCMUCS] More than one condition match in statement test.v,
5
More than one condition matches are found in 'unique case' statement
inside top.Unique_case, at time 5ns.
```

Line number 6 and 7 are overlapping.

## Support for Unique0 in Conditional Statements

SystemVerilog provides support for `unique0` conditional statements based on the *IEEE SystemVerilog LRM Std 1800™-2012 Section 12.4.2*. The `unique0` conditional statements

can identify improperly coded case statements that can lead to bugs at a later stage. The conditional statement is used to make a decision about whether a statement must be executed. The syntax for a conditional statement is as follows:

```
conditional_statement ::=
[unique_priority] case (cond_predicate) statement_or_null
unique_priority ::= unique | unique0 | priority
cond_predicate ::=
expression_or_cond_pattern { && expression_or_cond_pattern }
expression_or_cond_pattern ::=
expression | cond_pattern
cond_pattern ::= expression matches pattern
endcase
```

When `unique0` is used, it issues a warning if two or more of the case conditions are true at the same time. Alternatively, `unique0` does not issue any violation when none of the conditions matches. The usage of `unique0` enhances the performance and productivity of conditional statements in SystemVerilog assertions.

## Usage Example

The following examples illustrate usage of `unique0`:

```
module top;
bit [2:0] a;
initial
begin
a=3;
end

always @(a)
begin
unique case(a) // values 3,5,6,7 cause a violation report
0,1: $display("0 or 1");
2: $display("2");
4: $display("4");
endcase

unique0 case(a) // values 3,5,6,7 do not cause a violation report
0,1: $display("0 or 1");
2: $display("2");
4: $display("4");
endcase
end
endmodule
```

If the keyword `unique0` is used, there shall be no violation if no condition is matched.

On the other hand, unique case issues a violation message when no condition matches as follows:

```
Warning-[RT-NCMUCS] No condition matches in statement uniq0_case.v, 11
No condition matches in 'unique case' statement. 'default' specification
is missing, inside top, at time 0s.
```

## Enhancements to the -xIrm uniq\_prior\_final Compile-Time Option

The -xIrm uniq\_prior\_final compile-time option and keyword argument tells you when:

- A case statement is modified by the unique, unique0, or priority keywords and there is a case item expression that is not unique
- A conditional if statement is modified by the unique, unique0, or priority keywords and both of the following:
  - the conditional expression is not met
  - there is no else statement for the if statement

When these conditions happens, VCS issues a violation report.

The violation reports, contains the following information:

- The hierarchical name of the module instance that contains the case or if statement as described above, or contains a call to a user-defined task or function that contains these statements.
- The value of the evaluation expression which must be true for a for loop statement to continue to iterate.

Also, a violation report is generated in the following scenarios:

- there is a call to a function that contains the case or if statements as described above
- the function called is a possible assignment expression for the conditional ?: operator or in the RHS expression in a continuous assignment.

The following code examples illustrate these enhancements.

*Example 107 unique case Statement in Multiple Module Instances*

```

function foo (input a);
 unique case (a)
 1'b1: ;
 1'b1: ;
 endcase
endfunction

module T (input wire a);
 always_comb
 foo(a);
 endmodule

module Top;
 reg a;
 T t1(a);
 T t2(a);

 initial
 begin
 a = 1'b1;
 end
endmodule

```

case item expressions  
are not unique

module T calls the function

multiple instances of module T

In [Example 107](#), the violation reports refers to the hierarchical names of the instances:

```

Warning-[RT-MTOCMUCS] More than one condition match in statement
 More than one condition matches are found in 'unique case' statement
 inside
 $unit::foo, at time 0s.

```

Line number 3 and 4 are overlapping.

```
#0 in foo at test.v:2
#1 in Top.t1 at test.v:10
```

```

Warning-[RT-MTOCMUCS] More than one condition match in statement
test.v, 2
 More than one condition matches are found in 'unique case' statement
 inside
 $unit::foo, at time 0s.

```

Line number 3 and 4 are overlapping.

```
#0 in foo at test.v:2
#1 in Top.t2 at test.v:10
```

VCS identifies both the instances and generates two violation reports.

*Example 108 for Loop Statement Iteratively Calling a Function with a unique case Statement*

```
function foo (input a);
 unique case (a)
 1'b1:;
 1'b1:;
 endcase
endfunction

module T (input wire a);
 int P;
 always_comb
 for (int i=0;i<P;i++)
 foo(a);
 initial begin
 P = 2;
 end
endmodule

module Top;
 reg a;
 T t1(a);
 initial begin
 a = 1'b1;
 end
endmodule
```

case item expressions  
are not unique

for loop that iteratively  
calls the function

In the `for` loop, the value of `i` must be less than `p`, then there are violation reports because the `unique case` statement does not have unique case item expressions.

In [Example 108](#), the violation report includes the value of `i` in the iteration.

```
Warning-[RT-MTOCMUCS] More than one condition match in statement
test1.v, 2
 More than one condition matches are found in 'unique case' statement
 inside
 $unit::foo, at time 0s.
```

Line number 3 and 4 are overlapping.

```
#0 in foo at test1.v:2
#1 in Top.t1 at test1.v:12
#2 in loop with i = 0 at test1.v:11
```

Warning-[RT-MTOCMUCS] More than one condition match in statement  
test1.v, 2  
More than one condition matches are found in 'unique case' statement  
inside  
\$unit::foo, at time 0s.

Line number 3 and 4 are overlapping.

```
#0 in foo at test1.v:2
#1 in Top.t1 at test1.v:12
#2 in loop with i = 1 at test1.v:11
```

*Example 109 unique case Statement in a Function Call in a Possible Assignment Expression with the Conditional Operator*

```
function foo (input a);
 unique case (a)
 1'b1: ; case item expressions
 1'b1: ; are not unique
 endcase
endfunction
```

```
module Top;
 reg a, b;
 reg cond;
```

function call is an assignment expression  
of the conditional operator

initial

In Example 109, the following is the violation report:

Warning-[RT-MTOCMUCS] More than one condition match in statement  
test2.v, 2  
More than one condition matches are found in 'unique case' statement  
inside  
\$unit::foo, at time 0s.

Line number 3 and 4 are overlapping.

```
#0 in foo at test2.v:2
#1 in Top at test2.v:18
```

## Limitations

This enhancement supports only `unique/priority` case

- `unique0 if` is not supported.

## Single-Sized Packed Dimension Extension

VCS has implemented an extension to a single-sized packed dimension SystemVerilog signals and multidimensional arrays (MDAs). This section provides examples of using this extension for a single-sized packed dimension and explains how VCS expands the single size.

You can use the extension for these basic data types: `bit`, `reg`, and `wire` (using other basic data types with this extension is an error condition) The following is an example:

```
bit [4] a;
```

VCS expands the packed dimension `[4]` into `[0:3]`.

For packed MDAs, for example:

```
bit [4][4] a;
```

VCS expands the packed dimensions `[4][4]` into `[0:3][0:3]`.

You can use this extension in several ways. The following is an example of using this extension in a user defined type:

```
typedef reg [8] DREG;
```

The following is an example of using this extension in a structure, union, and enumerated type:

```
struct packed {
 DREG [20][20] arr4;
} [2][2] st1;
union packed {
 DBIT [20][20] arr5;
} [2][2] unl;
enum logic [8] {IDLE, XX=8'bxxxxxxxx, S1=8'bzzzzzzz, S2=8'hfff} arr3;
```

The following is an example of a user-defined structure and union with a packed memory or MDA:

```
typedef bit [2][24] DBIT;
```

```

typedef reg [2][24] DREG;

typedef struct packed {
 DBIT [20][20] arr1;
} ST;

ST [2][2] st;

typedef union packed {
 DREG [20][20] arr2;
} UN;

UN [2][2] un;

```

You can also use this extension for specifying module ports. For example:

```

module mux2(input wire [3] a,
 input wire [3] b,
 output logic [3] y);

```

You can use this extension in the parameter list of a user-defined function or task. For example:

```
function automatic integer factorial (input [32] operand);
```

You can use this extension in the definition of a parameter. For example:

```
parameter reg [2][2][2] p2 = 8;
```

### Error Conditions

The following are error conditions for this extension:

- Using the dollar sign (\$) as the size. For example:

```

reg [8:$] a;
reg [$] b;

```

- Using basic data types other than bit, reg, and wire. For example:

```
typedef shortint [8] DREG;
```

## Covariant Virtual Function Return Types

VCS supports, as an extension to SystemVerilog, covariant virtual function return types.

A covariant return type allows overriding a superclass method with a return type that is a derived type of the superclass method's return type. Covariant return types minimize the need for dynamic casts (upcasting or downcasting).

**Example 110 Sample code for covariant function return types**

```
class Base;
 virtual function Base clone();
 Base b = new this;
 return b;
 endfunction
endclass

class Derived extends Base;
 virtual function Derived clone();
 Derived d = new this;
 return d;
 endfunction
endclass
```

Without covariant types, the signature of the `Derived::clone()` above would have to be the same as in the `Base` class, like the following:

```
class Derived extends Base;
 virtual function Base clone();
 Derived d = new this;
 return d;
 endfunction
endclass
```

This leads to code like the following for users of the class:

```
Derived d = new;
Base b = d.clone(); // automatic down-cast to Base
Derived d2;
if(!$cast(d2, b)) begin
 b = null;
 $error(...); // some exception
end
```

Instead, with covariant return types, the code is simplified to:

```
Derived d = new;
Derived d2 = d.clone();
```

## Self Instance of a Virtual Interface

You can create a self instance of a virtual interface that points to itself when it is initialized. For example:

```
interface intf;
 int data1;
 int data2;
 virtual intf vi;
 initial
 vi = interface::self();
```

```

endinterface

module top;
 intf i0();
 initial #1 i0.vi.data1 = 100;
 always @(i0.data1)
 $display("trigger success");
endmodule

```

In this example, the virtual interface named `vi` is initialized with the expression:

```
vi = interface::self();
```

The `interface::self()` expression enables you provide a string variable that is effectively the `%m` format specification of the interface instance that VCS returns for assignment to the virtual interface variable. You use the `interface::self()` expression to initialize virtual interface variables in methodologies like UVM and VMM. It enables you to write components that are configurable with a string is the `%m` of the virtual interface that the component drives or monitors.

The expression `interface::self()` must be entered precisely. Otherwise it is a syntax error. Also notice the required delay (in this case `#1`) in the initialization of virtual interface `vi`. This delay is required to prevent a race condition.

This implementation is in accordance with the IEEE SystemVerilog LRM Std 1800™-2012 Section 9.7 “Fine-grain process control” that specifies:

“The `self()` function returns a handle to the current process, that is, a handle to the process making the call.”

SVA-bind is supported with self instances of virtual interfaces.

The following conditions are required for a self instance of a virtual interface:

- The self instance must be defined in the scope.
- The virtual interface type in the interface declaration must be the same as the interface that includes itself.
- Within an interface, you can only use the virtual `interface::self()` expression in a context that is valid for initializing a virtual interface. Any other use of the `interface::self()` expression results in a compilation error.
- Within an interface, you can use the virtual `interface::self()` expression in a context that is valid for initializing a virtual interface. Any other use of the `interface::self()` expression results in a compilation error.

## UVM Example

The following is an example of a self-instance of a virtual interface:

```
/* interface definition */
interface bus_if; //ports.
//signal declaration.
...
 initial begin
 uvm_resource_db#(virtual bus_if)::set("*",
 $sformatf("%m"), interface::self());
 end
endinterface

/* instantiated bus interface in design. */
//Add "bus()" to module called "top".
bind top bus_if bus();

/*Example config_db usage: */
if(!uvm_config_db#(virtual bus_if)::get(this, "",
 "top.bus", bus))
 `uvm_error("TESTERROR", "no bus interface available");
else
 'uvm_info("build", "got bus_if", UVM_LOW)
```

OR

```
/*Example resource_db usage: */
if(!uvm_resource_db#(virtual bus_if)::read_by_type(get_full_name(),
bus, this))
 `uvm_error("TESTERROR", "no bus interface available");
else
 'uvm_info("build", "got bus_if", UVM_LOW)
```

## Support for Shuffle Method for Multi-Dimensional Arrays

Array ordering methods reorder the elements of any unpacked array. These can be fixed or dynamically sized arrays except for associative arrays. Among the ordering methods, the shuffle method randomizes the order of the elements in the array.

When the shuffle method is called through a multidimensional array element, the method randomizes the order of the elements of the target array, if this array is not an associative array.

If the array calling shuffle method is another multidimensional array, then shuffle method only randomizes the element order of the leftmost dimension. VCS does not re-order the elements of other sub-dimensions.

## Use Model

VCS supports the usage of shuffle method for multidimensional arrays through the following compile-time option:

```
-xlrn sv_mda_shuffle
```

## Usage Example

The following example shows how the shuffle method works on fixed sized multidimensional integer array:

```
module test;
 int mda[3][4][5];
 initial begin
 mda = {{'{{0, 1, 2, 3, 4}, {5, 6, 7, 8, 9}, {10, 11, 12,
13, 14}, {{15, 16, 17, 18, 19}}, {{20, 21, 22, 23, 24}, {{25, 26, 27, 28, 29}, {{30, 31, 32,
33, 34}, {{35, 36, 37, 38, 39}}, {{40, 41, 42, 43, 44}, {{45, 46, 47, 48, 49}, {{50, 51, 52,
53, 54}, {{55, 56, 57, 58, 59}}}}}};
```

When you execute `mda[0][1].shuffle()`, the output is,

```
{
'{{0, 1, 2, 3, 4}, {8, 6, 9, 5, 7}, {10, 11, 12, 13, 14}, {15, 16,
17, 18, 19}}
'{{20, 21, 22, 23, 24}, {25, 26, 27, 28, 29}, {30, 31, 32, 33, 34},
{35, 36, 37, 38, 39}}
'{{40, 41, 42, 43, 44}, {45, 46, 47, 48, 49}, {50, 51, 52, 53, 54},
{55, 56, 57, 58, 59}}
```

In the result, the element in `mda [0][1]` is re-ordered.

When you execute `mda[2].shuffle()`, the output is,

```
{
'{{0, 1, 2, 3, 4}, {5, 6, 7, 8, 9}, {10, 11, 12, 13, 14}, {15, 16,
17, 18, 19}}
'{{20, 21, 22, 23, 24}, {25, 26, 27, 28, 29}, {30, 31, 32, 33, 34},
{35, 36, 37, 38, 39}}
'{{55, 56, 57, 58, 59}, {45, 46, 47, 48, 49}, {50, 51, 52, 53, 54},
{40, 41, 42, 43, 44}}
```

In the result, only the element of `mda[2]` is re-ordered, and the element order of sub-dimension arrays, such as `mda[2][0]` is unaltered.

When you execute `mda.shuffle()`, the output is,

```
{
'{{40, 41, 42, 43, 44}, {45, 46, 47, 48, 49}, {50, 51, 52, 53, 54},
{55, 56, 57, 58, 59}}
'{{0, 1, 2, 3, 4}, {5, 6, 7, 8, 9}, {10, 11, 12, 13, 14}, {15, 16,
17, 18, 19}}
'{{20, 21, 22, 23, 24}, {25, 26, 27, 28, 29}, {30, 31, 32, 33, 34},
{35, 36, 37, 38, 39}}
}
```

This example shows how the shuffle method works on dynamically sized multidimensional integer arrays:

```
int mda[$][5] ={
{0, 1, 2, 3, 4}
{5, 6, 7, 8, 9}
{10, 11, 12, 13, 14}
{15, 16, 17, 18, 19}
}
```

When you execute `mda[2].shuffle()`, the output is:

```
{
{0, 1, 2, 3, 4}
{5, 6, 7, 8, 9}
{13, 11, 14, 10, 12}
{15, 16, 17, 18, 19}
}
```

When you execute `mda.shuffle()`, the output is:

```
{
{15, 16, 17, 18, 19}
{5, 6, 7, 8, 9}
{10, 11, 12, 13, 14}
{0, 1, 2, 3, 4}
}
```

## Enhanced Clocking Block Behavior When Skew is negedge/posedge

By default, VCS overrides the clocking event with the skew when the skew is specified as posedge/negedge. However, you can use the `-ntb_opts no_cb_edge_override`

option to avoid overriding the clocking event at input, output, and inout. The following is the behavior of this option at different clocking directions:

- *Input*: Value is sampled at the specified clocking skew delay before the clocking event and the update happens at the clocking event.
- *Output*: The output is updated at the specified clocking skew delay after the clocking event.

## Usage Example

The following example shows the usage of the `-ntb_opts no_cb_edge_override` option.

### *Example 111 test.v*

```
module top;
 reg clk=0;
 reg [3:0] write, data, sdata;
 always #5 clk = ~clk;

 clocking cb@(posedge clk);
 default input negedge output negedge;
 input data;
 output write;
 endclocking

 always @(cb.data) begin
 sdata = cb.data;
 end

 always @ (write) begin
 $display($time,, "Clocking Block Write");
 end
 always @ (cb.data) begin
 $display($time,, "Clocking Block Read");
 end

 initial begin
 @(posedge clk);
 #3 data = 4'ha; // t = 8s
 #3 data = 4'hb; // t = 11s
 @(posedge clk);
 #3 cb.write <= 4'hc; // t = 18s
 #3 cb.write <= 4'hd; // t = 21s
 end
endmodule
```

Run the example using the following commands:

```
% vcs -sverilog test.v -ntb_opts no_cb_edge_override
% simv
```

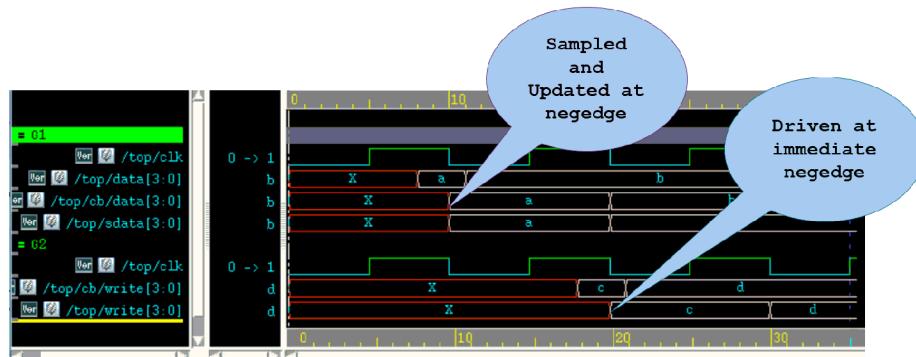
The following table shows the difference in the outputs with and without the `-ntb_opts no_cb_edge_override` option.

*Table 43 Outputs*

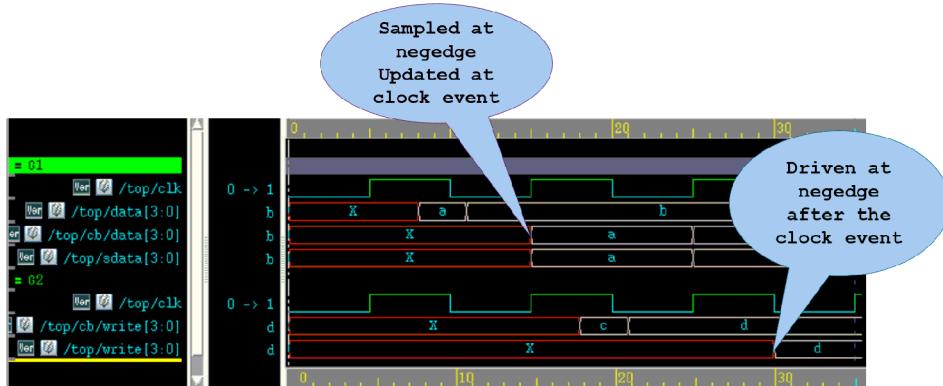
| With the Option        | Without the Option             |
|------------------------|--------------------------------|
| 15 Read25 Read30 Write | 10 Read20 Read20 Write30 Write |

The following figures shows the difference in behavior in the Wave window.

*Figure 164 Default Behavior*



*Figure 165 Behavior with the `-ntb_opts no_cb_edge_override` option*



## Support for Slice of String Variable

VCS supports string functions while accessing slice of string using the `-xlrn sv_string_slice` option. The following example shows the usage of slice operator on string.

*Example 112 test.v*

```
module top;
 string s0 = "Hello";
 initial begin
 $display("%s",s0[0:4]);
 $display("%s",s0[0:2]);
 end
endmodule
```

Run the example using the following commands:

- % vcs -sverilog -xlrn sv\_string\_slice test.v
- % simv

Following is the output:

```
Hello
Hel
```

## Support for Randomization of Floating Point Variables

VCS supports randomization of `real`, `realtime`, and `shortreal` variables in a constraint expression. This helps in generating random stimulus to your design in mixed-signal simulation.

### Use Model

To enable this feature, use the `-xlrn floating_pnt_constraint` compile-time option.

### Usage Example

Consider the following test case (`test.v`):

*Example 113 Randomization of Floating Point Variables*

```
program test;
 class cls;
 rand real x, y;
 constraint c1 { x > 1.78 + y; }
 constraint c2 { y < 56.78; }
 endclass
 initial begin
 cls obj = new;
```

```
 obj.randomize();
end
endprogram
```

Compile the test case as follows:

```
% vcs -sverilog test.v -xlm floating_pnt_constraint
```

To enable better linear distribution for randomization of floating point variable, use the `+ntb_solver_distribution=linear` runtime option.

To fine-tune the granularity, use the `+ntb_solver_linear_granularity=<floating point number>` runtime option. By default, the granularity is set to 1.0.

The command-line is as follows:

#### Compile Time

```
% vcs test.v -sverilog -xlm floating_pnt_constraint <other_options>
```

#### Simulation

```
% simv +ntb_solver_distribution=linear
+ntb_solver_linear_granularity=<floating point number>
```

## Support for Unpacked Array Concatenation in the HighConn of Inout Port

VCS supports unpacked array concatenation in the highconn of the inout ports of module, interface, program. This feature is supported for the following net types:

- Nets like `wire`, `wreal`, `supply0`, `supply1`, `voltage_r`, `wand`, `wor` and `Tri`.
- User defined net types that are legal to be connected in the highconn of the module.

## Use Model

Use the following option at compile time:

```
% vcs -sverilog <top_module> -xlm uac_inout_highconn <other_options>
```

#### Note:

In VCS 3-step flow, use this option at vcs step.

## Usage Example

Consider the following example:

*Example 114 test.v*

```
module top;
wire hc1[0:3];
sub inst({hc1[0:1], hc1[2:3]});
endmodule
module sub(lc1);
inout lc1[0:3];
real a[0:3];
assign lc1 = a;
endmodule
```

## Limitation

The limitation of this feature is as follows:

- Concatenations of different data types is not supported in highconn.

## Support for Index Locator Methods for Multi-Dimensional Arrays

VCS supports index locator methods for MDAs by using the `-x1rm sv_mda_find_index` option. The following locator methods are supported with the MDA's:

- `find_index()`
- `find_first_index()`
- `find_last_index()`

These methods return a queue with the indices of all items that satisfy the `with` clause expression. For MDA, the indices that are returned belong to the slowest varying dimension of MDA.

## Use Model

The use model is as follows:

```
% vcs -sverilog -x1rm sv_mda_find_index <filename>
```

### Note:

In VCS three step flow, use this option at `vcs` stage.

## Usage Example

Consider the following testcase:

*Example 115 test.v*

```
module test;
 int mda[$][2] = '{'1,2}, '{3,4}, '{5,6};
 int retQueue[$];
 initial
 begin
 retQueue = mda.find_first_index(item) with ((item[1] >= 4));
 $display("First index: %p", retQueue);
 retQueue = mda.find_index(item) with ((item[1] >= 4));
 $display(" Index: %p", retQueue);
 retQueue = mda.find_last_index(item) with ((item[1] >= 4));
 $display("Last index: %p", retQueue);
 end
endmodule:test
```

Use the following option at compile time:

```
% vcs -sverilog -xlrn sv_mda_find_index test.v
% simv
```

The output generated is as follows:

```
First index: '{1}
Index: '{1,2}
Last index: '{2}
```

## Limitation

Index locator methods for MDAs are not supported inside constraint block.

## Support for String Method Substr() with a Single Argument

The string method `Substr()` is defined in SystemVerilog LRM 1800-2017, Section 6.16.8, as follows:

```
function string substr(int i, int j);
 • str.substr(i, j) returns a new string that is a substring formed by
 characters in position i through j of str.
 • If i < 0, j < i, or j >= str.len(), substr() returns " " (the empty
 string).
```

The LRM does not specify a default value for argument `j`. When argument `j` is skipped in function call, VCS returns a substring formed by characters from position `i` to the end of the string and considers the second argument `j` value as `str.len() - 1` by default.

## Usage Example

Consider the following testcase:

*Example 116 test.v*

```
module test;
initial begin
automatic string str = "hello";
$display(str.substr(2));
end
endmodule
```

The output generated is as follows:

llo

## Support for Assignment Pattern with Single-Bit Scalar Nets or Variables

VCS supports assignment patterns for assignments to single-bit scalar nets or variables such as reg, logic, bit, and so on.

### Use Model

VCS supports the usage of assignment patterns with single-bit scalar nets or variables using the following compile-time option:

`-xlrn ap_for_scalar`

In the case of UUM flow, the switch must be passed at the VCS step.

## Usage Example

Consider the following testcase:

*Example 117 test.sv*

```
module top;
logic scalar1;
assign scalar1 = '{default:1};
initial begin
#1 $display("scalar1 = %b", scalar1);
end
endmodule
```

Run the design using the following command:

```
vcs -sverilog test.sv -full64 -xlrn ap_for_scalar -R
```

The output generated is as follows:

```
scalar1 = 1
```

## Limitations

This feature has the following limitations:

- Assignment pattern is not supported with single-bit enumeration type net or variable.
- Assignment pattern is not supported in the highconn of VHDL entity.
- Assignment pattern is not supported in parameter overrides.

## Support for Reading Value of any Clock Variable

VCS supports reading the value of any clocking variable whose clock direction is output.

### Use Model

VCS supports this feature using the following compile-time option:

```
-xlrn relax_cb_dir
```

### Usage Example

Consider the following testcase:

#### *Example 118 test.sv*

```
module top;
 parameter simulation_cycle = 100;
 bit SystemClock = 0;
 kdb_mst_bfm_interface bfm_ifc(SystemClock);
 test_top top_io(bfm_ifc);

 initial begin
 #(simulation_cycle/2) SystemClock = ~SystemClock;
 $display($realtime,,SystemClock);
 end
endmodule
```

Run the design using the following command:

```
%vcs -sverilog test.sv -full64 -xlrn relax_cb_dir
```

The output should not have any error messages.

# 17

## Aspect Oriented Extensions

---

Aspect-Oriented Programming (AOP) methodology complements the Object-Oriented Programming (OOP) methodology using a construct called aspect or an aspect-oriented extension (AOE) that can affect the behavior of a class or multiple classes. In AOP methodology, the terms “aspect” and “aspect-oriented extension” are used interchangeably.

Aspect-oriented extensions in SystemVerilog allow testbench engineers to design test cases more efficiently, using fewer lines of code.

AOP addresses issues or concerns that prove difficult to solve when using OOP to write constrained-random testbenches. These concerns include the following:

1. Context-sensitive behavior.
2. Unanticipated extensions.
3. Multi-object protocols.

In AOP, these issues are termed cross-cutting concerns as they cut across the typical divisions of responsibility in a given programming model.

In OOP, the natural unit of modularity is the class. Some of the cross-cutting concerns, such as multi-object protocols, cut across multiple classes and are not easy to solve using the OOP methodology. AOP is a way of modularizing such cross-cutting concerns. AOP extends the functionality of existing OOP derived classes and uses the notion of aspect as a natural unit of modularity. Behavior that affects multiple classes can be encapsulated in aspects to form reusable modules. As potential benefits of AOP are achieved better in a language where an aspect unit can affect behavior of multiple classes and therefore, can modularize the behavior that affects multiple classes, AOP ability in the SystemVerilog language is currently limited in the sense that an aspect extension affects the behavior of only a single class. It is useful to enable test engineers to design code that efficiently addresses concerns, such as context-sensitive behavior and unanticipated extensions.

AOP is used in conjunction with object-oriented programming. By compartmentalizing code containing aspects and cross-cutting concerns become easy to deal with. Aspects of a system can be changed, inserted or removed at compile time, and become reusable.

It is important to understand that the overall verification environment should be assembled using OOP to retain encapsulation and protection. Aspect-oriented extensions of Native

testbench should be used only for constrained-random test specifications with the aim of minimizing code.

Aspect-oriented extensions of Native testbench should not be used to do the following:

- Code base classes and class libraries
- Debug, trace, or monitor unknown or inaccessible classes
- Insert new code to fix an existing problem

For information on the creation and refinement of verification test benches, see the *Reference Verification Methodology User Guide*.

---

## Aspect-Oriented Extensions in SystemVerilog

In SystemVerilog, AOP is supported by a set of directives and constructs that need to be processed before compilation. Therefore, a SystemVerilog program with these aspect-oriented directives and constructs would need to be processed as per the definition of these directives and constructs in SystemVerilog to generate an equivalent SystemVerilog program that is devoid of aspect extensions, and consists of traditional SystemVerilog. Conceptually, AOP is implemented as pre-compilation expansion of code.

This chapter explains how aspect-oriented extensions in SystemVerilog are directives to SystemVerilog compiler as to how the pre-compilation expansion of code needs to be performed.

In SystemVerilog, an aspect extension for a class can be defined in any scope where the class is visible, except for within another aspect extension. That is, aspect extensions cannot be nested.

An aspect-oriented extension in SystemVerilog is defined using a new top-level *extends directive*. Terms “aspect” and “extends directive” have been used interchangeably throughout the document. Normally, a class is extended through derivation, but an extends directive defines modifications to a pre-existing class by doing *in-place* extension of the class. *In-place* extension modifies the definition of a class by adding new member fields and member methods, and changing the behavior of earlier defined class methods, without creating any new subclass(es). That is, aspect-oriented extensions change the original class definition without creating subclasses. These changes affect all instances of the original class that is extended by aspect-oriented extensions.

An *extends* directive for a class defines a scope in the SystemVerilog language. Within this scope exist the items that modify the class definition. These items within an *extends* directive for a class can be divided into the following three categories:

- Introduction

Declaration of a new property, or the definition of a new method, a new constraint, or a new coverage group within the *extends* directive scope adds (or *introduces*) the new symbol into the original class definition as a new member. Such declaration/definition is called an *introduction*.

- Advice

An *advice* is a construct to specify code that affects the behavior of a member method of the class by *weaving* the specified code into the member method definition. This is explained in more detail later. The advice item is said to be an advice *to* the affected member method.

- Hide list

Some items within an *extends* directive, such as a virtual method introduction, or an advice to virtual method may not be permissible within the *extends* directive scope depending upon the *hide permissions* at the place where the item is defined. A *hide list* is a construct whose placement and arguments within the *extends* directive scope controls the hide permissions. There could be multiple hide lists within an *extends* directive.

## Processing of Aspect-Oriented Extensions as a Precompilation Expansion

As a precompilation expansion, the Aspect-Oriented Extension code is processed by VCS to modify class definitions that it extends as per the directives in aspect-oriented extensions.

A *symbol* is a valid identifier in a program. Classes and class methods are symbols that can be affected by Aspect-Oriented Extensions. The Aspect-Oriented Extension code is processed which involves adding of introductions and *weaving* of advices in and around the affected symbols. Weaving is performed before actual compilation (and thereby, before symbol resolution). Therefore, under certain conditions, introduced symbols with the same identifier as some already visible symbol, can *hide* the already visible symbols. This is explained in more detail in [The hide\\_list Details](#). The preprocessed input program, now devoid of aspect-oriented extension, is then compiled.

Syntax:

```
extends_directive ::=
 extends extends_identifier (class_identifier) [dominate_list];
```

```

 extends_item_list
endextends

dominate_list ::=
dominates(extends_identifier {,extends_identifier});

extends_item_list ::=
 extends_item {extends_item}

extends_item ::=
 class_item
 | advice
 | hide_list

class_item ::=
 class_property
 | class_method
 | class_constraint
 | class_coverage
 | enum_defn

advice ::= placement procedure

placement ::=
before
| after
| around

procedure ::=
 | optional_method_specifiers task
 task_identifier(list_of_task_proto_formals);
 | optional_method_specifiers function function_type
 function_identifier(list_of_function_proto_formals)
endfunction

advice_code ::= [stmt] {stmt}

stmt ::= statement
 | proceed ;

hide_list ::=
hide([hide_item {,hide_item}]);

hide_item ::=
// Empty
| virtuals
| rules

```

*The symbols in bold are keywords and their syntax are as follows:*

extends\_identifier

Name of the aspect extension.

`class_identifier`

Name of the class that is being extended by the `extends` directive.

`dominate_list`

Specifies extensions that are *dominated* by the current directive. Domination defines the *precedence* between code woven by multiple extensions into the same scope. One extension can dominate one or more of the other extensions. In such a case, you must use a comma-separated list of `extends` identifiers.

`dominates(extends_identifier{,extends_identifier});`

A dominated extension is assigned lower precedence than an extension that dominates it. Precedence among aspects extensions of a class determines the order in which introductions defined in the aspects are added to the class definition. It also determines the order in which advices defined in the aspects are *woven* into the class method definitions thus, affecting the behavior of a class method. Rules for determination of precedence among aspects are explained later in [Precedence](#).

`class_property`

Refers to an item that can be parsed as a property of a class.

`class_method`

Refers to an item that can be parsed as a class method.

`class_constraint`

Refers to an item that can be parsed as a class constraint.

`class_coverage`

Refers to an item that can be parsed as `coverage_group` in a class.

`advice_code`

Specifies to a block of statements.

`statement`

Is a SystemVerilog statement.

`procedure_prototype`

A full prototype of the target procedure. Prototypes enable the advice code to reference the formal arguments of the procedure.

`opt_method_specifiers`

Refers to a combination of protection level specifier (local, or protected), virtual method specifier (virtual), and the static method specifier (static) for the method.

`task_identifier`

Name of the task.

`function_identifier`

Name of the function.

`function_type`

Data type of the return value of the function.

`list_of_task_proto_formals`

List of formal arguments to the task.

`list_of_function_proto_formals`

List of formal arguments to the function.

`placement`

Specifies the position at which the advice code within the advice is *woven* into the *target method* definition. Target method is either the class method, or some other new method that is created as part of the process of *weaving*, which is a part of precompilation expansion of code. The overall details of the process of “weaving” are explained in [Precompilation Expansion Details](#). The `placement` element could be any of the keywords, *before*, *after*, or *around*, and the advices with these placement elements are referred to as *before advice*, *after advice* and *around advice*, respectively.

The `proceed` statement

The `proceed` keyword specifies a SystemVerilog statement that can be used within advice code. The `proceed` statement is valid only within an *around* block and only a single `proceed` statement can be used inside the *advice code block* of an *around* advice. It cannot be used in a *before* advice block or an *after* advice block. The `proceed` statement is optional.

`hide_list`

Specifies the permission(s) for introductions to hide a symbol, and/or permission(s) for advices to modify local and protected methods. It is explained in detail in [The hide\\_list Details](#).

## Weaving Advice Into the Target Method

The target method is either the class method or some other new method that is created as part of the process of *weaving*. “Weaving” of all advices in the input program comprises several steps of *weaving of an advice into the target method*. Weaving of an advice into its target method involves the following:

A new method is created with the same method prototype as the target method and with the advice code block as the code block of the new method. This method is referred to as the *advice method*. The following table shows the rest of the steps involved in weaving of the advice for each type of placement element (*before*, *after*, and *around*).

*Table 44 Placement Elements*

| Element | Description                                                                                                                                                                                                                                                                                                                                                                 |
|---------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| before  | Inserts a new method-call statement that calls an advice method. The statement is inserted as the first statement to be executed before any other statements.                                                                                                                                                                                                               |
| after   | Creates a new method A with the target method prototype, with its first statement being a call to the target method. Second statement with A is a new method call statement that calls the advice method. All the instances in the input program where the target method is called are replaced by newly-created method calls to A. A is replaced as the new target method. |
| around  | All the instances in the input program where the target method is called are replaced by newly-created method calls to the advice method.                                                                                                                                                                                                                                   |

Within an *extends* directive, you can specify only one advice that can be specified for a given placement element and a given method. For example, an *extends* directive may contain a maximum of one *before*, one *after*, and *one around* advice each for a class method *Packet::foo* of a class *Packet*, but it may not contain two *before* advices for the *Packet::foo*.

### *Example 119 before* Advice

Target method:

```
class packet;
 task myTask();
 $display("Executing original code\n");
 endtask
endclass
```

**Advice:**

```
before task myTask ();
 $display("Before in aoel\n");
endtask
```

Weaving of the advice in the target method yields the following.

```
task myTask();
 mytask_before();
 $display("Executing original code\n");
endtask

task mytask_before ();
 $display("Before in aoel\n");
endtask
```

Note that the SystemVerilog language does not impose any restrictions on the names of newly-created methods during precompilation expansions, such as *mytask\_before*. Compilers can adopt any naming conventions for those methods that are created as a result of the *weaving* process.

**Example 120 after Advice**

**Target method:**

```
class packet;
 task myTask();
 $display("Executing original code\n");
 endtask
endclass
```

**Advice:**

```
after task myTask ();
 $display("Before in aoel\n");
endtask
```

Weaving of the advice in the target method yields the following.

```
task myTask_newTarget();
 myTask();
 myTask_after();
endtask

task myTask();
 $display("Executing original code\n");
endtask

task myTask_after ();
 $display("After in aoel\n");
endtask
```

As a result of weaving, all the method calls to `myTask()` in the input program code are replaced by method calls to `myTask_newTarget()`. Also, `myTask_newTarget()` replaces `myTask` as the target method for `myTask()`.

### *Example 121 around Advice*

Target method:

```
class packet;
 task myTask();
 $display("Executing original code\n");
 endtask
endclass
```

Advice:

```
around task myTask ();
 $display("Around in aoel\n");
endtask
```

Weaving of the advice in the target method yields the following:

```
task myTask_around();
 $display("Around in aoel\n");
endtask

task myTask();
 $display("Executing original code\n");
endtask
```

As a result of weaving, all the method calls to `myTask()` in the input program code are replaced by method calls to `myTask_around()`. Also, `myTask_around()` replaces `myTask()` as the target method for `myTask()`.

During weaving of an *around* advice that contains a `proceed` statement, the `proceed` statement is replaced by a method call to the target method.

### *Example 122 around Advice With proceed*

Target method:

```
class packet;
 task myTask();
 $display("Executing original code\n");
 endtask
endclass
```

Advice:

```
around task myTask ();
 proceed;
```

```
$display("Around in aoel\n");
endtask
```

Weaving of the advice in the target method yields the following:

```
task myTask_around();
myTask();
$display("Around in aoel\n");
endtask

task myTask();
$display("Executing original code\n");
endtask
```

As a result of weaving, all the method calls to `myTask()` in the input program code are replaced by method calls to `myTask_around()`. The `proceed` statement in the around code is replaced with a call to the target method `myTask()`. Also, `myTask_around()` replaces `myTask` as the target method for `myTask()`.

## Precompilation Expansion Details

Precompilation expansion of a program containing the aspect oriented-extensions code is done in the following order:

1. Preprocessing and parsing of all input code.
2. Identification of the symbols, such as methods and classes affected by extensions.
3. The precedence order of aspect extensions (and thereby, introductions and advices) for each class is established.
4. Addition of introductions to their respective classes as class members in their order of precedence. Whether an introduction can or cannot override or hide a symbol with the same name that is visible in the scope of the original class definition, is dependent on certain rules related to the `hide_list` parameter. For a detailed explanation, see [The hide\\_list Details](#).
5. Weaving of all advices in the input program are weaved into their respective class methods as per the precedence order.

These steps are described in more detail in the following sections.

- [Precedence](#)

## Precedence

Precedence is specified through `dominate_list` (see [dominate\\_list](#)). There is no default precedence across files; if precedence is not specified, the tool is free to weave code

in any order. Within a file, dominance established by *dominate\_list* always overrides precedence established by the order in which *extends* directives are coded. Only when the precedence is not established after analyzing the dominate lists of directives, is the order of coding used to define the order of precedence.

Within an *extends* directive, there is an inherent precedence between advices. Advices that are defined later in the directive have higher precedence than those defined earlier.

Precedence does not change the order between adding of introductions and weaving of advices in the code. Precedence defines the order in which introductions to a class are added to the class, and the order in which advices to methods belonging to a class are woven into the class methods.

#### **Example 123 Precedence Using *dominates***

```
// Beginning of file test.sv
class packet;
 // Other member fields/methods
 //...

 task send();
 $display("Sending data\n");
 endtask
endclass

program top ;
 initial begin
 packet p;
 p = new();
 p.send();
 end
endprogram

extends aspect_1(packet) dominates (aspect_2, aspect_3);

 after task send(); // Advice 1
 $display("Aspect_1: send advice after\n");
 endtask
endextends

extends aspect_2(packet);

 after task send() ; // Advice 2
 $display("Aspect_2: send advice after\n");
 endtask
endextends

extends aspect_3(packet);

 around task send(); // Advice 3
 $display("Aspect_3: Begin send advice around\n");
 endtask
endextends
```

```

 proceed;
 $display("Aspect_3: End send advice around\n");
 endtask

 before task send(); // Advice 4
 $display("Aspect_3: send advice before\n");
 endtask
endextends

// End of file test.sv

```

In [Example 123](#), multiple aspect extensions for a class named *packet* are defined in a single SystemVerilog file. As specified in the dominating list of `aspect_1`, `aspect_1` dominates both `aspect_2` and `aspect_3`. As per the dominating lists of the aspect extensions, there is no precedence order established between `aspect_2` and `aspect_3`. As `aspect_3` is coded later than `aspect_2`, `aspect_3` has higher precedence than `aspect_2`. Therefore, the precedence of these aspect extensions in the decreasing order of precedence is:

```
{aspect_1, aspect_3, aspect_2}
```

This implies that the advice(s) within `aspect_2` have lower precedence than advice(s) within `aspect_3`, and advice(s) within `aspect_3` have lower precedence than advice(s) within `aspect_1`. Therefore, `advice 2` has lower precedence than `advice 3` and `advice 4`. Both `advice 3` and `advice 4` have lower precedence than `advice 1`. Between `advice 3` and `advice 4`, `advice 4` has higher precedence as it is defined later than `advice 3`. It puts the order of advices in the increasing order of precedence as {2, 3, 4, 1}.

## Adding of Introductions

*Target scope* refers to the scope of the class definition that is being extended by an aspect. Introductions in an aspect are appended as new members at the end of its target scope. If an extension A has precedence over extension B, the symbols introduced by A are appended first.

Within an aspect extension, symbols introduced by the extension are appended to the target scope in the order they appear in the extension.

There are certain rules according to which an introduction symbol with the same identifier name as a symbol that is visible in the target scope, may or may not be allowed as an introduction. These rules are discussed later in the chapter.

## Weaving of advices

An input program may contain several aspect extensions for any or each of the different class definitions in the program. Weaving of advices needs to be carried out for each class method for which an advice is specified.

Weaving of advices in the input program consists of weaving of advices into each such class method. Weaving of advices into a class method A is unrelated to weaving of advices into a different class method B. Therefore, weaving of advices to various class methods can be done in any ordering of the class methods.

For weaving of advices into a class method, all the advices pertaining to the class method are identified and ordered in the order of increasing precedence in the list, L. This is the order in which these advices are woven into the class method, thereby, affecting the runtime behavior of the method. The advices in list L are woven in the class method as per the following steps (Target method is initialized to the class method):

1. Advice A that has the lowest precedence in L is woven into the target method as explained earlier. Note that the target method may either be the class method or some other method newly created during the weaving process.
2. Advice A is deleted from the list L.
3. The next advice on list L is woven into the target method. This continues until all the advices on the list have been woven into list L.

It would become apparent from the example provided later in this section that how the order of precedence of advices for a class method affects how advices are woven into their target method. Thus, the relative order of execution of advice code blocks. The *before* and *after* advices within an aspect to a target method are unrelated to each other. Their relative precedence to each other does not affect their relative order of execution when a method call to the target method is executed. The code block of the *before* advice executes before the target method code block, and the *after* advice code block executes after the target method code block. When an *around* advice is used with a *before* or *after* advice in the same aspect, code weaving depends upon their precedence with respect to each other. Depending upon the precedence of the *around* advice with respect to other advices in the aspect for the same target method, the *around* advice either may be woven before all or some of the other advices, or may be woven after all of the other advices.

As an example, weaving of advices 1, 2, 3, 4 specified in aspect extensions in [Example 123](#) leads to the expansion of code in the manner shown in [Example 124](#). Advices are woven in the order of increasing precedence {2, 3, 4, 1} as explained earlier.

#### **Example 124 After Weaving Advice-2 of Class packet**

```
// Beginning of file test.sv

program top ;
 packet p;
 p = new();
 p.send_Created_a();
endprogram

class packet;
 ...

```

```

// Other member fields/methods
...
task send();
 p$display("Sending data\n");
endtask

task send_Created_a();
 send();
 send_after_Created_b();
endtask

task send_after_Created_b();
 $display("Aspect_2: send advice after\n");
endtask

endclass

extends aspect_1(packet) dominates (aspect_2, aspect_3);
after task send(); // Advice 1
 $display("Aspect_1: send advice after\n");
endtask
endextends

extends aspect_3(packet);
around task send(); // Advice 3
 $display("Aspect_3: Begin send advice around\n");
 proceed;
 $display("Aspect_3: End send advice around\n");
endtask

before task send(); // Advice 4
 $display("Aspect_3: send advice before\n");
endtask
endextends

// End of file test.sv

```

**Example 124** shows how the input program looks like after weaving **advice 2** into the class method. Two new methods **send\_Created\_a** and **send\_after\_Created\_b** are created in the process. The instances of method calls to the target method **packet::send** are modified, such that the code block from **advice 2** executes after the code block of the target method **packet::send**.

**Example 125** shows how the input program looks like after weaving advice 3 into the class method. A new method **send\_around\_Created\_c** is created in this step. The instances of method call to the target method **packet::send\_Created\_a** are modified. The code block from **advice 3** executes **around** the code block of the **packet::send\_Created\_a** method. Also, note that the **proceed** statement from the advice code block is replaced by a call to **send\_Created\_a**. At the end of this step, **send\_around\_Created\_c** becomes the new target method for weaving of further advices to **packet::send**.

**Example 125 After Weaving Advice-3 of Class packet**

```
// Beginning of file test.sv

program top ;
 packet p;
 p = new();
 p.send_Created_a();
endprogram

class packet;
 ...
 // Other member fields/methods
 ...
 task send();
 p$display("Sending data\n");
 endtask

 task send_Created_a();
 send();
 send_after_Created_b();
 endtask

 task send_after_Created_b();
 $display("Aspect_2: send advice after\n");
 endtask

endclass

extends aspect_1(packet) dominates (aspect_2, aspect_3);
after task send(); // Advice 1
 $display("Aspect_1: send advice after\n");
endtask
endextends

extends aspect_3(packet);
around task send(); // Advice 3
 $display("Aspect_3: Begin send advice around\n");
 proceed;
 $display("Aspect_3: End send advice around\n");
endtask

before task send(); // Advice 4
 $display("Aspect_3: send advice before\n");
endtask
endextends

// End of file test.sv
```

[Example 126](#) shows how the input program looks like after weaving advice 4 into the class method. A new method, `send_before_Created_d`, is created in this step and a call to it is added as the first statement in the target method, `packet::send_around_Created_c`. Also, note that the outcome is different if advice 4 (before advice) is defined earlier than

advice 3 (around advice) within aspect\_3, as it affects the order of precedence of advice 3 and advice. In this scenario, advice 3 (around advice) weaves around the code block from advice 4 (before advice), unlike the current outcome.

**Example 126 After Weaving Advice-4 of Class packet**

```
// Beginning of file test.sv

program top;
 packet p;
 p = new();
 p.send_around_Created_c();
endprogram

class packet;
 ...
 // Other member fields/methods
 ...
 task send();
 $display("Sending data\n");
 endtask

 task send_Created_a();
 send();
 send_after_Created_b();
 endtask

 task send_after_Created_b();
 $display("Aspect_2: send advice after\n");
 endtask

 task send_around_Created_c();
 send_before_Created_d();
 $display("Aspect_3: Begin send advice around\n");
 send_after_Created_a();
 $display("Aspect_3: End send advice around\n");
 endtask

 task send_before_Created_d();
 $display("Aspect_3: send advice before\n");
 endtask
endclass

 extends aspect_1(packet) dominates (aspect_2, aspect_3);
after task send(); // Advice 1
 $display("Aspect_1: send advice after\n");
endtask
endextends

// End of file test.sv
```

**Example 127** shows the input program after weaving of all four advices {2, 3, 4, 1}. New methods `send_after_Created_e` and `send_Created_f` are created in the last step of weaving and the instances of method call to `packet::send_around_Created_c` are replaced by the method call to `packet::send_Created_f`.

**Example 127 After Weaving All Advices {2,3,4,1}of Class packet**

```
// Beginning of file test.sv

program top;
 packet p;
 p = new();
 p.send_Created_f();
endprogram

class packet;
 ...
 // Other member fields/methods
 ...
 task send();
 $display("Sending data\n");
 endtask

 task send_Created_a();
 send();
 send_Created_b();
 endtask

 task send_after_Created_b();
 $display("Aspect_2: send advice after\n");
 endtask

 task send_around_Created_c();
 send_before_Created_d();
 $display("Aspect_3: Begin send advice around\n");
 send_after_Created_a();
 $display("Aspect_3: End send advice around\n");
 endtask

 task send_before_Created_d();
 $display("Aspect_3: send advice before\n");
 endtask
 task send_after_Created_e();
 $display("Aspect_1: send advice after\n");
 endtask

 task send_Created_f();
 send_around_Created_c();
 send_after_Created_e()
 endtask
endclass
```

```
// End of file test.sv
```

When executed, the output of this program is as follows:

```
Aspect_3: send advice before
Aspect_3: Begin send advice around
Sending data
Aspect_2: send advice after
Aspect_3: End send advice around
Aspect_1: send advice after
```

[Example 128](#) shows the program in which `aoe1` dominates `aoe2`.

#### *Example 128 Around Advice With **dominates**-I*

```
// Begin file test.sv
class foo;
 int i;

 task myTask();
 $display("Executing original code\n");
 endtask
endclass

extends aoe1 (foo) dominates (aoe2);
 around task myTask();
 proceed;
 $display("around in aoe1\n");
 endtask
endextends

extends aoe2 (foo);
 around task myTask();
 proceed;
 $display("around in aoe2\n");
 endtask
endextends

program top;
 foo f;

 initial begin
 f = new();
 f.myTask();
 end
endprogram
// End file test.sv
```

When `aoe1` dominates `aoe2` and when the program is executed, its output is as follows:

```
Executing original code
around in aoe2
around in aoe1
```

[Example 129](#) shows the program in which `aoe2` dominates `aoe1`.

#### *Example 129 Around Advice With **dominates-II***

```
// Begin file test.sv
class foo;
 int i;
 task myTask();
 $display("Executing original code\n");
 endtask
endclass

extends aoe1 (foo);
 around task myTask();
 proceed;
 $display("around in aoe1\n");
 endtask
endextends

extends aoe2 (foo) dominates (aoe1);
 around task myTask();
 proceed;
 $display("around in aoe2\n");
 endtask
endextends

program top;
 foo f;
initial begin
 f = new();
 f.myTask();
end
endprogram
// End file test.sv
```

On the other hand, when `aoe2` dominates `aoe1` as in [Example 129](#), the output is as follows:

```
Executing original code
around in aoe1
around in aoe2
```

## Symbol Resolution Details

As introductions and advices defined within `extends` directives are preprocessed as a precompilation expansion of the input program, the preprocessing occurs earlier than the final symbol resolution stage within a compiler. Therefore, it is possible for aspect-oriented

extensions code to reference symbols that are added to the original class definition using aspect-oriented extensions. Because advices are woven after introductions are added to the class definitions, advices can be specified for introduced member methods and can reference introduced symbols.

An advice to a class method can access and modify the member fields and methods of the class object to which the class method belongs. An advice to a class function can access and modify the variable that stores the return value of the function.

Furthermore, members of the original class definition can also reference symbols introduced by aspect extensions using the extern declarations (?). Extern declarations can also be used to reference symbols introduced by an aspect extension to a class in some other aspect extension code that extends the same class.

An introduction that has the same identifier as a symbol that is already defined in the target scope as a member property or member method is not permitted.

## Examples

### *Example 130 before Advice on Class Task*

```
// Begin file test.sv
 class packet;
 task foo(integer x); //Formal argument is "x"
 $display("x=%0d\n", x);
 endtask
 endclass
 extends myaspect(packet);
 // Make packet::foo always print: "x=99"
 before task foo(integer x);
 x = 99; //force every call to foo to use x=99
 endtask
 endextends
 program top;
 packet p;

 initial begin
 p = new();
 p.foo(100);
 end
 endprogram
// End file test.sv
```

The extends directive in [Example 130](#) sets the x parameter inside the foo() task to 99 before the original code inside of foo() executes. Actual argument to foo() is not affected and is not set unless passed-by-reference using reference.

### *Example 131 after Advice on Class Function*

```
// Begin file test.sv
 class packet ;
 function integer bar();
```

```

 bar = 5;
 $display("Point 1: Value = %d\n", bar);
 endfunction
endclass
extends myaspect(packet);
 after function integer bar();
 $display("Point 2: Value = %d\n", bar);
 bar = bar + 1; // Stmt A
 $display("Point 3: Value = %d\n", bar);
 endfunction
endextends
program top ;
 packet p;

 initial begin
 p = new();
 $display("Output is: %d\n", p.bar());
 end
endprogram
// End file test.sv

```

An advice to a function can access and modify the variable that stores the return value of the function as shown in [Example 131](#). In this example a call to `packet::bar` returns 6 instead of 5 as the final return value is set by the Stmt A in the advice code block. When executed, the output of the program code is as follows:

```

Point 1: Value = 5
Point 2: Value = 5
Point 3: Value = 6
Output is: 6

```

## The `hide_list` Details

The `hide_list` item of an `extends_directive` specifies the permission(s) for introductions to hide symbols, and/or `advice` to modify local and protected methods. By default, an introduction does not have permission to hide symbols that are previously visible in the target scope, and it is an error for an extension to introduce a symbol that hides a global or super-class symbol.

The `hide_list` option contains a comma-separated list of options such as:

- The `virtuals` option permits the hiding (that is, overriding) of virtual methods defined in a super class. Virtual methods are the only symbols that may be hidden; global, and file-local tasks and functions may not be hidden. Furthermore, all introduced methods must have the same virtual modifier as their overridden super-class and overriding sub-class methods.

- The *rules* option permits the extension to suspend access rules and to specify advice that changes protected and local virtual methods; by default, extensions cannot change protected and local virtual methods.
- An empty option list removes all permissions, that is, it resets permissions to default.

In Example 132, the `print` method introduced by the `extends` directive hides the `print` method in the super class.

### Example 132 Change Permission Using `hide virtuals`

```

class pbase;
 virtual task print();
 $display("I'm pbase\n");
 endtask
endclass

class packet extends pbase;
 task foo();
 $display(); //Call the print task
 endtask
endclass

extends myaspect(packet);
 hide(virtuals); // Allows permissions to
 // hide pbase::print task

 virtual task print();
 $display("I.m packet\n.");
 endtask
endextends

program test;
 packet tr;
 pbase base;

 initial begin
 tr = new();
 tr.print();
 base = tr;
 base.print();
 end
endprogram

```

As explained earlier, there are two types of hide permissions:

1. Permission to hide virtual methods defined in a super class (option `virtuals`) is referred to as *virtuals-permission*. An *aspect item* is either an introduction, an advice, or a hide list within an aspect. If such permission is granted at an aspect item within an aspect, then the *virtuals-permission* is said to be *on* or the *status* of *virtuals-permission* is said to be on at that aspect item and at all the aspect items following that, until a hide list that forfeits the permission is encountered. If *virtuals-permission* is not on or the status

of `virtuals-permission` is not on at an aspect item, then the `virtuals-permission` at that item is said to be *off* or the *status* of `virtuals-permission` at that item is said to be off

2. Permission to suspend access rules and to specify advice that changes protected and local virtual methods (option "rules") is referred to as *rules-permission*. If within an aspect, at an aspect item, such permission is granted, then the `rules-permission` is said to be *on* or the *status* of `rules-permission` is said to be *on* at that aspect item and at all the aspect items following that, until a hide list that forfeits the permission is encountered. If `rules-permission` is not on or the status of `rules-permission` is not on at an aspect item, then the `rules-permission` at that item is said to be *off* or the *status* of `rules-permission` at that item is said to be off.

Permission for one of the above types of hide permissions does not affect the other. Status of `rules-permission` and `hide-permission` varies with the position of an aspect item within the aspect. Multiple `hide_list(s)` may appear in the extension. In an aspect, whether an introduction or an advice that can be affected by hide permissions is permitted to be defined at a given position within the aspect extension is determined by the status of the relevant hide permission at the position. A `hide_list` at a given position in an aspect can change the status of `rules-permission` and/or `virtuals-permission` at that position and all following aspect items until any hide permission status is changed again in that aspect using `hide_list`.

[Example 133](#) illustrates how the two hide permissions can change at different aspect items within an aspect extension.

#### *Example 133 Hide Permissions*

```
class pbase;
 virtual task print1();
 $display("pbase::print1\n");
 endtask

 virtual task print2();
 $display("pbase::print2\n");
 endtask
endclass

class packet extends pbase;
 task foo();
 rules_test();
 endtask

 local virtual task rules_test();
 $display("Rules-permission example\n");
 endtask
endclass

extends myaspect(packet);
// At this point within the myaspect scope,
// virtuals-permission and rules-permission are both off.
```

```

 hide(virtuals); // Grants virtuals-permission

 // virtuals-permission is on at this point within aspect,
 // and therefore can define print1 method introduction.
 virtual task print1();
 $display("packet::print1\n.");
 endtask

 hide(); // virtuals-permission is forfeited

 hide(rules); // Grants rules-permission

 // Following advice permitted as rules-permission is on
 // before local virtual task rules_test();
 $display("Advice to Rules-permission example\n");
 endtask

 hide(virtuals); // Grants virtuals-permission

 // virtuals-permission is on at this point within aspect,
 // and therefore can define print2 method introduction.
 virtual task print2();
 $display("packet::print2\n.");
 endtask
 endextends

program test;
 packet tr;

 initial begin
 tr = new();
 tr.print1();
 tr.foo();
 tr.print2();
 end
endprogram

```

## Examples

Introducing new members into a class:

[Example 135](#) shows how aspect-oriented extensions can be used to introduce new members into a class definition. `myaspect` adds a new property, constraint, coverage group, and method to the `packet` class.

### Example 134 Introducing New Member

```

class packet;
 rand bit[31:0] hdr_len;
endclass

extends myaspect(packet);
 integer sending_port;

```

```

event cg_trigger;

constraint con2 {
 hdr_len == 4;
}

covergroup cov2 @(cg_trigger);
 coverpoint sending_port;
endgroup

task print_sender();
 $display("Sending port = %0d\n", sending_port);
endtask
endextends

program test;
 packet tr;

initial begin
 tr = new();
 void'(tr.randomize());
 tr.sending_port = 1;
 tr.print_sender();
 -> tr.cg_trigger;
end
endprogram

```

As mentioned earlier, new members that are introduced should not have the same name as a symbol that is already defined in the class scope. So, aspect-oriented extensions defined in the manner shown in [Example 135](#) is not allowed, as the aspect `myaspect` defines `x` as one of the introductions when the symbol `x` is already defined in class `foo`.

*Example 135 Non-Permissible Introduction*

```

class foo;
 rand integer myfield;
 integer x;
endclass

extends myaspect(foo);
 integer x ;

 constraint con1 {
 myfield == 4;
 }
endextends

program test;
 foo tr;

initial begin

```

```

 tr = new();
 $display("Non-permissible introduction error....!");
 void'(tr.randomize());
 end
endprogram

```

### Examples of Advice Code

In [Example 139](#), the `extends` directive adds advices to the `packet::send` method.

#### *Example 136 before-after Advices*

```

// Begin file test.sv
class packet;
 task send();
 $display("Sending data\n.");
 endtask
endclass

extends myaspect(packet);
 before task send();
 $display("Before sending packet\n");
 endtask

 after task send();
 $display("After sending packet\n");
 endtask
endextends

program test;
 packet p;

 initial begin
 p = new();
 p.send();
 end
endprogram

// End file test.sv

```

When [Example 139](#) is executed, the output is as follows:

```

Before sending packet
Sending data
After sending packet

```

In [Example 139](#), `extends` directive `myaspect` adds advice to turn off constraint `c1` before each call to the `foo::pre_randomize` method.

#### *Example 137 Turn-off Constraint Using before Advice*

```

class foo;
 rand integer myfield;

```

```

constraint c1 {
 myfield == 4;
}
endclass

extends myaspect(foo);

before function void pre_randomize();
 c1.constraint_mode(0);
endfunction
endextends

program test;
 foo tr;

 initial begin
 tr = new();
 void'(tr.randomize());
 $display("myfield value = %d, constraint mode OFF (!= 4)!", tr.myfield);
 end
endprogram

```

In [Example 139](#), extends directive `myaspect` adds advice to set a property named `valid` to 0 after each call to the `foo::post_randomize` method.

*Example 138 Change Property Value After post-randomize()*

```

class foo;
 integer valid;
 rand integer myfield;

 constraint c1 {
 myfield inside {[0:6]};
 }
endclass

extends myaspect(foo);
after function void post_randomize();
 if (myfield > 6)
 valid = 0;
 else
 valid = 1;
endfunction
endextends

program test;
 foo tr;

 initial begin
 tr = new();
 void'(tr.randomize());

```

```

 $display("valid = %0d ", tr.valid);
 end
endprogram

```

[Example 139](#) shows an aspect extension that defines an around advice for the class method, `packet::send`. When the code in example is compiled and run, the around advice code is executed instead of original `packet::send` code.

*Example 139 Changing Test Functionality Using around Advice*

```

// Begin file test.sv
class packet;
 integer len;
 task setLen(integer i);
 len = i;
 endtask

 task send();
 $display("Sending data\n.");
 endtask
endclass

program test;
 packet p;

 initial begin
 p = new();
 p.setLen(5000);
 p.send();
 p.setLen(10000);
 p.send();
 end
endprogram

extends myaspect(packet);
 around task send();
 if (len < 8000)
 proceed;
 else
 $display("Dropping packet\n");
 endtask
endextends

// End file test.sv

```

This [Example 139](#) also demonstrates how the around advice code can reference properties, such as `len` in the `packet` object `p`. When executed the output of the above example is as follows:

```

Sending data
Dropping packet

```

# 18

## Using Constraints

---

This chapter explains VCS support for the following constraints features:

- Support for Array Slice in Unique Constraints
- Support for Object Handle Comparison in Constraint Guards
- Support for Pure Constraint Block
- Support for SystemVerilog Bit Vector Functions in Constraints
- Inconsistent Constraints
- Constraint Debug
- Constraint Guard Error Suppression
- Support for Array and Cross-Module References in std::randomize()
- Support for Cross-Module References in Constraints
- State Variable Index in Constraints
- Using DPI Function Calls in Constraints
- Using Foreach Loops Over Packed Dimensions in Constraints
- Randomized Objects in a Structure
- Support for Typecast in Constraints
- Strings in Constraints
- Support for Dynamic MDA Elements in the Randomize Call Arguments
- Support for Dynamic MDA in the Inside Expression Used in Constraints
- Support for Array Reduction Methods over MDAs in Constraints
- SystemVerilog LRM 1800™-2012 Update
- Enhancement to the Randomization of Multidimensional Array Functionality
- Supporting Random Array Index

- Supporting System Function Calls
- Supporting Foreach Loop Iteration over Array Select
- Support for Enumerated Type Methods in a Constraint Expression
- Improved Support for Function Evaluation in Constraints

## Support for Array Slice in Unique Constraints

You can specify slices of unpacked array variables in unique constraints as defined in the IEEE SystemVerilog LRM Std 1800™-2012 Section 18.5.5 “Uniqueness constraints”.

### *Example 140 Array Slice in Unique Constraint*

```
module top;

class c;
 rand bit[2:0] a[7];
 rand bit[2:0] b;

 constraint cst1 {
 unique { a[0:2], a[6], a[3:5], b };
 }
endclass: c

c c1;

initial
begin
 c1 = new;
 c1.randomize();
end

endmodule: top
```

In Example 140, the array slices, `a[0:2]` and `a[3:5]` are used in the specification of the unique constraint `cst1`.

## Limitation

Specifying loop variables of the `foreach` statement inside array slices is not supported with unique constraints.

### *Example 141 Loop Variables Inside an Array Slice*

```
class C;
 rand int x[5];

 constraint cst {
```

```

foreach (x[i]) {
 if (i > 0) unique { x[i-1+:2] } ;
}
}
endclass

program automatic test;
 C obj = new;
 initial obj.randomize;
endprogram

```

In [Example 141](#), the loop variable `i` is specified in the array slice `x[i-1+:2]`. The following error message is issued:

```

Error-[NYI-CSTR-AS] NYI constraint: array slice
test.sv, 6
$unit, "this.x[(i - 1)+:2]"
The expression 'this.x[(i - 1)+:2]' contains an array slice
that is not yet supported in unique constraint.
Please remove the array slice from the unique constraint,
or replace it with entire array or simple array select.

```

## Support for Object Handle Comparison in Constraint Guards

You can use the equal `==` and not equal `!=` operators to specify object handle comparisons in constraint guards as defined in the IEEE SystemVerilog LRM Std 1800™-2012 Section 18.5.13 “Constraint guards”. You can compare between two object handles or between an object handle and the value `null`. The object handles being compared must be instances of the same class. Using object handle comparison as constraint guards can prevent potential randomization errors caused by non-existent or incorrect object handles.

### *Example 142 Comparison Between Two Object Handles*

```

module top;

class A;
 rand int i;

 constraint cst0 {
 i > 0;
 i < 10;
 }
endclass: A

class C;
 rand bit[3:0] x;
 rand A a1,a2,a3;

 constraint cst1 {
 if (a1 == a3) { x == 3; }
 }

```

```

 if (a1 != a3) { x == 5; }

}

function new;
 a1 = new;
 a2 = new;

 a3 = a2;
endfunction

endclass: C

C c1;

initial
begin
 c1 = new;
 repeat(3) begin
 c1.randomize();
 $display("x = %p\ta1 = %p\ta2= %p", c1.x, c1.a1, c1.a2);
 end

 c1.a1 = c1.a2;

 repeat(3) begin
c1.randomize();
 $display("x = %p\ta1 = %p\ta2= %p", c1.x, c1.a1, c1.a2);
 end
end

endmodule: top

```

In [Example 142](#), a different constraint is applied to the variable `x` depending on the object handles `a1` and `a3`. If `a1` equals to `a3`, `x` is constrained to 3. If `a1` is not equal to `a3`, `x` is constrained to 5. The following is the result of the example:

```

x = 5 a1 = '{i:2} a2= '{i:5}
x = 5 a1 = '{i:8} a2= '{i:5}
x = 5 a1 = '{i:9} a2= '{i:1}
x = 3 a1 = '{i:1} a2= '{i:1}
x = 3 a1 = '{i:3} a2= '{i:3}
x = 3 a1 = '{i:7} a2= '{i:7}

```

#### *Example 143 Comparison Between an Object Handle and Null*

```

module top;

class A;
 rand int i;

 constraint cst0 {
 i > 0;
 i < 10;
 }

```

Chapter 18: Using Constraints  
Support for Object Handle Comparison in Constraint Guards

```

 }
endclass: A

class C;
 rand bit[3:0] x;
 rand A a1;
 A a2;

 constraint cst1 {
if (a2 != null) { x == 3; }

 function new;
 a1 = new;
 endfunction
endclass: C

C c1;

initial
begin
 c1 = new;
 repeat(3) begin
 c1.randomize();
 $display("x = %p\ta1 = %p\ta2 = %p", c1.x, c1.a1, c1.a2);
 end

 c1.a2 = new;
 repeat(3) begin
 c1.randomize();
 $display("x = %p\ta1 = %p\ta2 = %p", c1.x, c1.a1, c1.a2);
 end
end

endmodule: top

```

In [Example 143](#), the 4-bit variable `x` is constrained to value 3 when the object handle `a2` is not **null**. The following is the result of the example:

```

x = 6 a1 = '{i:5} a2 = null
x = 1 a1 = '{i:5} a2 = null
x = 9 a1 = '{i:1} a2 = null
x = 3 a1 = '{i:1} a2 = '{i:0}
x = 3 a1 = '{i:3} a2 = '{i:0}
x = 3 a1 = '{i:7} a2 = '{i:0}

```

---

## Limitations

The following are the limitations with this functionality:

- Object handles returned by function calls cannot be used for comparison in constraint guards.
- An array of objects cannot be used for comparison in constraint guards. Comparing singleton members is supported. The following constraint specification results in a “Not Yet Implemented” runtime error.

```
constraint cst1 {
 foreach (a_array[j]) {
 if (a_array[j] != null) { x inside { [2:5] }; }
 }
}
```

### Error message:

```
Error-[NYI] Not Yet Implemented
orig_null.sv, 18
Feature is not yet supported: objects in object (in)equality must
currently refer to singleton members (no array elements, function
calls, null, ...)
```

---

## Support for Pure Constraint Block

You can specify pure constraint block as defined in the IEEE SystemVerilog LRM Std 1800™-2012 Section 18.5.2 “Constraint inheritance”. A pure constraint block, specified with the `pure` keyword defines a constraint prototype in a virtual class. The constraint implementation is provided when a non-virtual class is derived from the virtual class. The constraint that overrides a pure constraint block can be declared using a constraint block, a constraint prototype or an external constraint of the same name in the body of the derived class.

### *Example 144 Pure Constraint Implementation in a Derived Class*

```
virtual class B;
 pure constraint t1;
 pure constraint q1;
endclass

class C extends B;

 rand int z,z1;
 randc int x,y;
 rand bit a,b;

 constraint t1{
```

```

 x inside {2,3,5};
 y inside {3,4,7};
 z==x+y;
 }

 constraint q1{
 z1==(a+b);
 }
endclass

module m;
 C c=new();
 initial begin
 repeat (3) begin
 c.randomize();
 $display("x = %0d y = %0d z = %0d\n", c.x, c.y, c.z);
 $display("a = %0d b = %0d z1 = %0d\n", c.a, c.b, c.z1);
 end
 end
endmodule

```

In Example 144, the pure constraint block is declared in the virtual base class `B`. The constraints of the pure constraint block are implemented in class `C` that is derived from class `B`. The following is the result of the example:

```

x = 5 y = 7 z = 12
a = 1 b = 0 z1 = 1
x = 2 y = 3 z = 5
a = 0 b = 1 z1 = 1
x = 3 y = 4 z = 7
a = 0 b = 0 z1 = 0

```

#### *Example 145 Pure Constraint Implementation in a Hierarchy of Derived Classes*

```

virtual class A;
 pure constraint q;

 virtual function obj ();
 endfunction
endclass

virtual class B extends A;
 pure constraint s;
 int i;

 function obj ();
 i=10;
 endfunction
endclass

class C extends B;
 rand bit [7:0]x,y,z;
 rand bit [3:0]a;

```

```

constraint q {
 x < 8'h80; y < 8'h80; z < 8'h80;
 z == x+y;
}

constraint s{
 a inside { 0, 10 };
}
endclass

module top;
C c=new;
initial begin
repeat(2) begin
 c.randomize();
 c.obj();
 $display("x = %0d y = %0d z = %0d", c.x, c.y, c.z);
 $display("a = %0d\n", c.a);
end
end
endmodule

```

In [Example 145](#), the pure constraint blocks are declared in the virtual base class `A` and the virtual class `B` that is derived from class `A`. The constraints of the pure constraint blocks are implemented in class `C` that is derived from class `B`. The following is the result of the example:

```

x = 91 y = 20 z = 111
a = 10

x = 13 y = 45 z = 58
a = 10

```

#### *Example 146 Pure Constraint Error With Missing Constraint Implementation*

```

virtual class A;
 rand int i;
 pure constraint t;
endclass

class B extends A ;
 rand bit [7:0]x,y,z;
endclass

module m;
 B b=new;

 initial begin
 b.randomize();
 end
endmodule

```

In [Example 146](#), the pure constraint block is declared in the virtual base class A. The derived class B does not include the constraint implementation for the pure constraint block. As a result, the following error message is issued:

```
Error-[CSTR-PCNI] 'pure' constraint not implemented
test.v, 6
$unit
 'pure' constraint 't' has not been implemented in class 'B'
```

*Example 147 Pure Constraint Error With Missing Virtual Declaration*

```
class A;
 pure constraint c1;
endclass

class B;
 rand int x;
 randc int y;

 constraint c1 {
 x == 12;
 y inside {2,3,5};
 }
endclass

program test;
 B c ;
 initial begin
 c=new();
 c.randomize(x,y);
 end
endprogram
```

In [Example 147](#), the pure constraint block is declared in the base class A. Pure constraint blocks must be declared inside a virtual class. As a result, the following error message is issued:

```
Error-[CSTR-IUPC] Illegal use of pure constraint
test.v, 2
$unit
 Only 'virtual' (i.e. abstract) class may contain 'pure' constraints.
```

*Example 148 Pure Constraint Error With Mismatched Prototypes*

```
virtual class A;
 pure static constraint q1;
endclass;

virtual class B extends A;
 pure static constraint v1;
endclass;

class C extends B;
```

```

rand bit [3:0]a[3];
rand bit [3:0]b;
rand int dyn_arr[];

constraint q1 {
 foreach (dyn_arr [k])
 { dyn_arr [k] > 10; }

 dyn_arr.sum() == 500;
}

constraint v1 { unique {a, b}; }
endclass

module m;
 C c=new();
 initial begin
 c.dyn_arr =new[10];
 c.randomize();
 end
endmodule

```

In [Example 148](#), the pure constraint blocks are declared in the virtual base class `A` and the virtual class `B` that is derived from class `A`. The constraints of the pure constraint blocks are implemented in class `C` that is derived from class `B`. However, the constraint declarations in class `C` do not include the `static` keyword that is specified in the pure constraint blocks. As a result, the following error messages are issued:

```

Error-[CSTR-PCPM] 'pure' constraint prototype mismatch
negative_pureconstraint3.v, 14
$unit
 The declaration of constraint 'q1' in class 'C' must match its
 corresponding
 'pure' constraint declaration in its base class hierarchy.

```

```

Error-[CSTR-PCPM] 'pure' constraint prototype mismatch
negative_pureconstraint3.v, 21
$unit, "constraint v1 { unique {this.a, this.b}; }"
 The declaration of constraint 'v1' in class 'C' must match its
 corresponding
 'pure' constraint declaration in its base class hierarchy.

```

## Support for SystemVerilog Bit Vector Functions in Constraints

You can specify the following SystemVerilog bit vector functions in constraints:

- [\\$countones Function](#)
- [\\$onehot Function](#)

- \$onehot0 Function
- \$countbits Function
- \$bits Function

The \$countones, \$onehot, \$onehot0, \$countbits and \$bits are defined as bit vector system functions in the IEEE SystemVerilog LRM Std 1800™-2012. These functions are supported in the constraint context as an operator to the expression in its argument and can be used to create an iterated constraint expression, similar to the use of array reduction methods in constraints.

The following example explains the differences between handling \$countones, \$onehot, \$onehot0, and \$countbits as functions and as expressions in the constraint context:

```
rand bit [3:0] vector;
constraint cst { $countones (vector) == 4; }
```

As defined in IEEE SystemVerilog LRM Std 1800™-2012 Section 18.5.12, the semantic restrictions related to a function call in constraints require the solver to solve the random variable as a function argument first and then use the return value of the function as a state variable. In this example, if \$countones is treated as a system function, the random variable, vector, is used as a function argument. The random variable is unconstrained and it can be assigned any value between 4'b0000 and 4'b1111. For example, vector is randomized to a value 4'b1010. The function \$countones returns a value of 2 based on the vector value of 4'b0101. A constraint solver failure is issued because the return value of the function 2 is not equal to 4. In this case, there is a 31/32 chance that a constraint solver failure is issued. This is unlikely the intent.

If \$countones is treated as an operator to the expression in its argument, it can be used to create iterative expression involving the bits of the expression. In this case, you can constrain how many bits of the expression is 1'b1. The constraint \$countones (vector) == 4 is considered as follows:

```
vector[0] + vector[1] + vector[2] + vector[3] == 4
```

The solver generates a solution for the random variable vector: 4'b1111. This is the most likely the intent when \$countones, \$onehot, \$onehot0, and \$countbits are used in the constraint context and is the behavior supported by VCS.

For more information on the bit vector functions, see the IEEE SystemVerilog LRM Std 1800™-2012 Section 20.9 “Bit vector system functions”.

## \$countones Function

The `$countones` system function returns an integer equal to the number of bits in the expression having value 1. You can use `$countones` as a constraint expression to specify the number of bits inside the `$countones` expression to be randomized to value 1.

### *Example 149 Using \$countones in Constraint Specification*

```
module top;

class c;
 rand bit[7:0] b;

 constraint cst1 {
 $countones (b) == 3;
 }
endclass: c

c c1;

initial
begin
 c1 = new;
 repeat(5) begin
 c1.randomize();
 $display("b = %b", c1.b);
 end
end

endmodule: top
```

In [Example 140](#), the 8-bit variable `b` is constrained to generate a random value with exactly 3 bits having value 1. The following is the result of the example.

```
b = 10000011
b = 01010001
b = 10010010
b = 01100001
b = 10001100
```

## \$onehot Function

The `$onehot` function returns true if one and only one bit in the expression have value 1. You can use `$onehot` as a constraint expression to specify one and only one bit inside the `$onehot` expression to be randomized to value 1.

### *Example 150 Using \$onehot in Constraint Specification*

```
module top;

class c;
```

```

rand bit[7:0] b;

constraint cst1 {
 $onehot (b);
}
endclass: c

c c1;

initial
begin
 c1 = new;
 repeat(5) begin
 c1.randomize();
 $display("b = %b", c1.b);
 end
end

endmodule: top

```

In [Example 143](#), the 8-bit variable `b` is constrained to generate a random value with one and only one bit having value 1. The following is the result of the example.

```

b = 00000001
b = 01000000
b = 00001000
b = 10000000
b = 01000000

```

## \$onehot0 Function

The `$onehot0` function returns true if zero or one bit in the expression have value 1. You can use `$onehot0` as a constraint expression to specify zero or one bit inside the `$onehot0` expression to be randomized to value 1.

### *Example 151 Using \$onehot0 in Constraint Specification*

```

module top;

class c;
 rand bit[2:0] b;

 constraint cst1 {
 $onehot0 (b);
 }
endclass: c

c c1;

initial
begin

```

```

c1 = new;
repeat(5) begin
 c1.randomize();
 $display("b = %b", c1.b);
end
endmodule: top

```

In [Example 151](#), the 3-bit variable `b` is constrained to generate a random value with zero or one bit having value 1. The following is the result of the example.

```

b = 100
b = 000
b = 100
b = 001
b = 010

```

## \$countbits Function

The `$countbits` function counts the number of bits that have a specific set of values (0, 1, x, z) in a bit vector. The syntax is as follows:

```
$countbits (expression, control_bit {, control_bit})
```

The control bit is a 1-bit logic that can have '0, '1, 'x, or 'z values. If a value with a width greater than 1 is passed, only the least significant bit (LSB) is used. If any individual value appears more than once in the control bits, it is treated exactly as if it had appeared once.

The `$countbits` function returns an integer that is equal to the number of bits in the expression whose values match one of the control bit entries.

For example,

- `$countbits (expression, '1)` returns the number of bits in the expression having value 1.
- `$countbits (expression, '1, '0)` returns the number of bits in the expression having values 1 or 0.

### Note:

As SystemVerilog constraints support only two-state values. The use of 'x or 'z as control bits results in an error message when used in the constraint context.

The expression argument is of a `bit-stream` type. For more information on the bit vector functions, see IEEE SystemVerilog LRM Std 1800™-2012 Section 20.9 “Bit vector system functions”.

**Example 152 Using \$countbits in Constraint Specification**

```

program automatic test;
 class cls;
 rand bit [7:0] driver_port1, driver_port2;

 constraint c1 {
 //randomize 'driver_port1' in such a way that number of
 "ones" or
 // no of active driver ports should be two
 $countbits(driver_port1, 1) == 2;

 // randomize 'driver_port2' in such a way that number of
 "zeros" should be four
 $countbits(driver_port2, 0) == 4;
 }
 endclass

 cls obj = new;

 initial begin
 obj.randomize;
 assert($countbits(obj.driver_port1, 1) == 2);
 assert($countbits(obj.driver_port2, 0) == 4);
 end
endprogram

```

## \$bits Function

The `$bits` system function returns the number of bits required to hold an expression as a bit stream. The return type is `integer`. The syntax is as follows:

```
$bits (expression | data_type)
```

The value returned by `$bits` is determined without actual evaluation of the expression that it encloses. An error message is issued if you enclose a function that returns a dynamically sized data type. The `$bits` return value is valid during elaboration only if the expression contains fixed-size data types.

The `$bits` system function returns `0` when called with a dynamic sized expression that is empty. An error message is issued,

- When you use the `$bits` system function directly with a dynamically sized data type identifier.
- When you use the `$bits` system function on an object of an interface class type.

For more information on the bit vector functions, see IEEE SystemVerilog LRM Std 1800™-2012 Section 20.6.2 “Expression size system function”.

**Example 153 Using \$bits in Constraint Specification**

```

program automatic test;

 class cls1 #(parameter width1 = 8, width2 = 32);

 rand bit [width1-1:0] r1;
 rand bit [width2-1:0] r2;
 rand bit [7:0] r3;

 constraint c1 {
 r2[$bits(r1)-1:0] == '1; // randomize r2, such that LSB bits
($bits(r1) no of bits) to all ones
 r3 == $bits(r1); // randomize r3, such that r3 should be
equal to $bits(r1) or width of r1
 }
 endclass

 cls1 #() obj1 = new; // object obj1 holds default parameter values
8(w1), 32(w2)
 // so, $bits(r1) will be '8'

 cls1 #(4, 16) obj2 = new; // object obj2 holds over- ridden
parameter values 4(w1), 16(w2)
 // so, $bits(r1) will be '4'

 initial begin
 obj1.randomize;
 assert(obj1.r3 == 8);
 assert(obj1.r2[7:0] == '1);

 obj2.randomize;
 assert(obj2.r3 == 4);
 assert(obj2.r2[3:0] == '1);

 end
endprogram

```

## Inconsistent Constraints

VCS correctly identifies inconsistent constraints while trying to find the minimal set causing the inconsistency. VCS supports two options to find inconsistent constraints: binary search and linear search. You can use two new options to set larger timeout values. The default timeout values for each iteration of the constraint solver are 100 seconds for the binary search and 10 seconds for the linear search. You can set larger timeout values in seconds. For example:

```
% simv +ntb_binary_debug_solver_cpu_limit=200
% simv +ntb_linear_debug_solver_cpu_limit=20
```

**Note:**

If the constraint solver timeout value is too low, VCS may not be able to find the minimal set of conflicting constraints. If the solver timeout value is too high, performance may degrade while finding a conflict. Therefore, setting optimal timeout values is important.

Inconsistent constraints are non-fatal by default. VCS continues to run after a constraint failure. Use the `+ntb_stop_on_constraint_solver_error=0|1` option, where `1` enables stop on first error and `0` disables stop on first error to control how VCS handles these inconsistencies. For example, to make VCS stop the simulation on the first constraint failure, use the following command line:

```
simv +ntb_stop_on_constraint_solver_error=1
```

When VCS detects inconsistent constraints, the default printing mode only displays the failure subset. For example:

The constraint solver failed when solving the following set of constraints:

```
rand integer y; // rand_mode = ON
rand integer z; // rand_mode = ON
rand integer x; // rand_mode = ON
constraint c // (from this) (constraint_mode = ON)
{
 (x < 1) ;
 (x in { 3 , 5 , 7 : 11 }) ;
}
```

You can use the `+ntb_enable_solver_trace_on_failure=0|1|2|3` runtime option as follows:

- 
- |   |                                                                                                       |
|---|-------------------------------------------------------------------------------------------------------|
| 0 | Print a one-line failure message with no details.                                                     |
| 1 | Print only the failure subset (this is the default).                                                  |
| 2 | Print the entire constraint problem and the failure subset.                                           |
| 3 | Print only the failure problem. This is useful when the solver fails to determine the minimum subset. |
- 

## Constraint Debug

Generally, there are two kinds of constraint debug scenarios. In the first scenario, VCS solves the random variables but the you wish to get a better understanding of how the random variables are solved. This is about debugging the solved values. In the second scenario, VCS either times out when solving or solves after a long time. This is about performance debug.

The following sections describe the VCS features that can help with these kind of constraint debugs.

- [Partition](#)
  - [Randomize Serial Number](#)
  - [Solver Trace](#)
  - [Test Case Extraction](#)
  - [Using multiple +ntb\\_solver\\_debug arguments](#)
  - [Summary for the +ntb\\_solver\\_debug Option](#)
  - [Support for Save and Restore Stimulus](#)
- 

## Partition

Whether it is `std::randomize` or the randomization of a class object, it generally involves one or more state and random variables. Constraints are used to describe relationships between these variables. An important concept of constrained randomization is the notion of partitions. In other words, a randomize call is partitioned into one or more smaller constraint problems to solve. At runtime, VCS groups all the related random variables involved in each randomization into one or more partitions. If there are no constraints between two random variables, they are not solved in the same partition. Here is an example to illustrate this concept:

```
class myClass;
 rand int x;
 rand int y;
 rand int z;
 rand byte a;
 rand byte b;
 bit c;
 constraint m {
 x > z;
 c -> a == b;
 }
 constraint n {
 y > 0;
 }
myClass obj = new;
obj.randomize(); // 1st randomize() call
obj.randomize() with {x!=y}; // 2nd randomize() call
```

For the first randomize call, the following constraints are used to solve the five random variables: `x`, `y`, `z`, `a`, and `b` and VCS creates three partitions for these random variables.

```
x > z; // from the constraint block m
c -> a == b; // from the constraint block m
y > 0; // from the constraint block n
```

The random variables `x` and `z` are grouped in one partition because of a constraint (`x > z`) relating the two together.

The random variables `a` and `b` are grouped in another partition because of the constraint (`c -> a == b`).

There are no constraints between `y` and any other random variable. So, `y` is on a third partition of its own.

Because the random variables from different partitions are not constrained together, they do not have to be solved in any particular order.

For the second `randomize()` call, a new constraint is added in the inline constraint (that is `randomize()` with). The following are the four constraints for the same 5 random variables.

```
x > z; // from the constraint block m
c -> a == b; // from the constraint block m
y > 0; // from the constraint block n
x != y; // from the inline constraint
// - randomize() with ..
```

For this second `randomize` call, two partitions are created.

The first partition has the random variables: `x`, `y`, and `z` because the following constraints relate all three together: (`x > z`), (`y > 0`), and (`x != y`).

The second partition has the random variables `a` and `b` because of the (`c -> a == b`) constraint.

## Randomize Serial Number

Each randomization in a simulation is assigned a serial number starting with 1. For example, if there are ten `randomize` calls (`std::randomize` or randomization of class objects) in a simulation, they are numbered from 1 to 10.

By default, the `randomize` serial numbers are not printed at runtime. To display the `randomize` serial numbers during simulation, you need to run the simulation with the `+ntb_solver_debug=serial` option.

```
% simv +ntb_solver_debug=serial
```

After each randomization completes, VCS prints the `randomize` serial number along with some runtime and memory data for the `randomize()` call.

Using a randomize serial number provides a mechanism to focus the constraint debug on a specific `randomize()` call. If the randomize serial number is used together with the partition number, it is the specified partition within the specified randomize call that becomes the focus for the constraint debug.

To specify the *n<sup>th</sup>* partition of the *m<sup>th</sup>* randomize call, the notation `m.n` is used.

## Solver Trace

To get more insight to how VCS solves a randomize call, you can enable solver trace reporting by using the `+ntb_solver_debug=trace` runtime option. The following is an example of the solver trace:

```
// Part.sv
class C;
 rand byte x, y, z, m, n, p, q;

 constraint imply {
 x > 3 -> y > p; // C1
 z < bigadd (x, q); // C2
 n != 0; // C3
 }

 function byte bigadd (byte a, b);
 return (a + b);
 endfunction

endclass

program automatic test;
 C obj = new;
 initial begin
 repeat (5) begin
 obj.randomize() with { m == z; }; // C4
 end
 end
endprogram
```

For this example, let us determine the partitions that are created by the solver.

The SystemVerilog LRM mandates that function arguments must be solved first in order to compute the function that is used to constraint other random variables. In other words, separate partitions must be created for `(x, q)` and then for `z`.

- The constraint expression `C1` relates the random variables `x, y, p` together. So they are solved together in one partition.
- The constraint expression `C2` using function call in constraint requires that `z` is solved in a different partition from `x` and `q`.

- Since the random variable  $q$  is not related to any other random variables,  $q$  is solved in a partition on its own.
  - Similarly, the random variable  $n$  is not related to any other random variables,  $n$  is solved in another partition on its own.
  - The constraint expression  $c4$  is an inline constraint relating the two random variables  $m$  and  $z$  together. Therefore,  $m$  and  $z$  be solved together in one partition.
  - Given the above descriptions, you can see four partitions are created.
    - Partition 1 to solve  $x, y, p$  together
    - Partition 2 to solve  $n$  alone
    - Partition 3 to solve  $q$  alone
    - Partition 4 to solve  $z$  and  $m$  together

To compile and run this example and enable solver trace for the third randomize call, use the following command:

```
% vcs -sverilog part.sv
% simv +ntb_solver_debug=trace +ntb_solver_debug_filter=3
```

Part of the solver trace shows the partition information. The following is a part of the solver trace from the above command.

```

rand bit signed [7:0] q; // rand_mode = ON

...
Solving Partition 4 (mode = 2)

bit signed [7:0] fv_3 /* this .C::bigadd(x , q) */ = -127;
rand bit signed [7:0] z; // rand_mode = ON
rand bit signed [7:0] m; // rand_mode = ON

```

It is required to specify the `randomize()` calls and/or partitions to report the solver trace details. For example, the following command reports the solver trace for the second `randomize()` call and all partitions within this `randomize()` call of the simulation.

```
% simv +ntb_solver_debug=trace +ntb_solver_debug_filter=2
```

The following command reports the solver trace for the third partition of the fifth `randomize()` call of the simulation.:

```
% simv +ntb_solver_debug=trace +ntb_solver_debug_filter=5.3
```

If the solver trace is to be enabled for multiple `randomize` calls, you can specify the list of random serial and partition numbers (optionally) in a comma-separated list for the `+ntb_solver_debug_filter` option. For example, consider the following `randomize()` calls and their partitions:

- Serial number 2, all partitions of this second `randomize()` call
- Serial number 5, just the third partition of this fifth `randomize()` call
- Serial number 10, all partitions of this tenth `randomize()` call
- Serial number 15, just the 30th partition of this 15th `randomize()` call.

The following command reports the solver traces for these `randomize()` calls and their partitions:

```
% simv +ntb_solver_debug=trace \ +ntb_solver_debug_filter=2,5.3,10,15.30
```

The following command reports the solver traces for the `randomize()` calls or partitions listed in a text file, such as `serial_trace.txt` is the file name.

```
% simv +ntb_solver_debug=trace \
+ntb_solver_debug_filter=file:serial_trace.txt
```

The following command reports the solver traces for all `randomize()` calls in the simulation. Be aware that this may produce a lot of data if there are many `randomize()` calls in the simulation.

```
% simv +ntb_solver_debug=trace +ntb_solver_debug_filter=all
```

or

```
% simv +ntb_solver_debug=trace_all
```

The `+ntb_solver_debug_filter` is not needed on the second `simv` command line.

**Note:**

Reporting solver traces for all `randomize()` calls can generate very large data files. Using the `+ntb_solver_debug=trace` and `+ntb_solver_debug_filter=serial_num|file` options and arguments limit the solver trace reports to the ones on which you want to focus the constraint debug.

## Constraint Profiler

To debug any performance related issues, profiling is required to identify the top consumers of time/memory. VCS provides a constraint profiler feature that can be enabled by using the `+ntb_solver_debug=profile` runtime option and keyword argument.

```
% simv +ntb_solver_debug=profile
```

This `simv` command line runs the simulation and collects runtime and memory data on each of the `randomize()` calls in the simulation. The randomize calls/partitions that take the most time and memory are listed out in a constraint profile report in the file `simv.cst/html/profile.xml`, where `simv` is the name of the simulation executable.

To view the constraint profile report in `simv.cst/html/profile.xml`, open the file with the Firefox or Chrome Web browser. You cannot view this file in Internet Explorer on Windows.

The random serial numbers for the randomize calls and/or partitions that take the most time are listed in the `simv.cst/serial2trace.txt` file.

**Note:**

The unified profiler also does constraint profiling. The Unified profiler is an LCA feature, for more information see the *VCS LCA Features Guide*.

## Test Case Extraction

The solver trace shows the list of variables and constraints for each of the partitions. By wrapping this data inside a SystemVerilog class in a `program` block, you can create a standalone test case to compile and simulate to shorten the debug time. If you wish to try different things to better understand the solver behavior and also wish fix the constraint issue, you can do it on this extracted test case instead of the original design to save compile and runtime.

To enable test case extraction, you can enable the solver trace reporting by using the `+ntb_solver_debug=extract` runtime option and keyword argument. You must specify the specific `randomize()` calls to extract the test cases for using the `+ntb_solver_debug_filter` option.

For example, test case extraction is enabled for the second `randomize` call, that is, `randomize serial number = 2`:

```
% simv +ntb_solver_debug=extract +ntb_solver_debug_filter=2
```

This extracts a test case for each of the partitions of the `randomize()` call. Extracted test cases are saved in the `simv.cst/testcases` directory, where `simv` is the name of the simulation executable. The extracted test cases follow this naming convention:

```
extracted_r_serial#_p_partition#.sv
```

Once extracted, you can follow the commands to compile and run the standalone test case. For example, to simulate the extracted test case for the third partition of the second `randomize()` call of the original design:

```
cd simv.cst/testcases
% vcs -sverilog extracted_r_2_p_3.sv -R
```

Similar to reporting solver traces for a single partition or for multiple `randomize()` calls and their partitions, you can enable test case extraction for these too. For example:

```
% simv +ntb_solver_debug=extract \
+ntb_solver_debug_filter=5.3
% simv +ntb_solver_debug=extract \
+ntb_solver_debug_filter=2,5.3,10,15.30
% simv +ntb_solver_debug=extract \
+ntb_solver_debug_filter=file:serial_trace.txt
```

#### Note:

You can only extract test cases from a partition. If VCS fails before any partition is created, test case extraction does not work.

When VCS encounters a `randomize()` call that has no solution or has constraint inconsistencies, VCS automatically extracts a test for it and saves the extracted test case using the following naming convention:

```
% simv.cst/testcases/extracted_r_serial#_p_partition#_inconsistency.sv
```

When VCS fails to solve a `randomize()` call due to solver time outs, test case extraction is also automatically enabled for it and VCS saves the extracted test case using the following naming convention:

```
% simv.cst/testcases/extracted_r_serial#_p_partition#_timeout.sv
```

## Using multiple +ntb\_solver\_debug arguments

To use multiple `+ntb_solver_debug` arguments, such as `serial`, `trace`, `extract`, and `profile`, you can use plus (+) to combine them. For example:

```
% simv +ntb_solver_debug=serial+trace+extract \
+ntb_solver_debug_filter=3,4
```

## Summary for the +ntb\_solver\_debug Option

The runtime option `+ntb_solver_debug` provides you with many constraint debug features to debug constraints in batch mode.

### **+ntb\_solver\_debug=serial**

The serial number assignment to the randomizations in a simulation provides a method to identify the `randomize()` calls to be debugged next. Once identified, you can use this runtime option with appropriate arguments to report the trace and extract test cases. The constraint profiler also uses the same identification method to provide feedback, such as which specific `randomize()` calls to optimize for best performance improvements.

### **+ntb\_solver\_debug=trace**

This enables solver trace reporting for the specified `randomize()` calls. This helps you to understand how VCS solves the random variables for given `randomize` calls. The `+ntb_solver_debug_filter` option is required to specify a list of `randomize()` calls for which to enable the solver trace.

### **+ntb\_solver\_debug=profile**

This enables constraint profiling for the simulation at runtime. The profile report provides important information to you, such as which `randomize` calls should be targeted for improving constraint performance to bring down the total simulation runtime or memory.

### **+ntb\_solver\_debug=extract**

This enables test case extraction for the specified `randomize` calls. This creates standalone test cases for you to compile and run outside the original design. This should help in quicker turnaround time to experiment possible fixes as it is faster to compile and run a smaller test case. The `+ntb_solver_debug_filter` option is required to specify a list of `randomize` calls for which to enable test case extraction.

### **+ntb\_solver\_debug=verbose**

The `+ntb_solver_debug=verbose` option provides you more detailed information in the solver output. This information includes the file name and the line number for all the individual constraint conditions inside the constraint block from the solver output.

The sample solver output is as follows:

```
Solver failed when solving following set of constraints
rand bit[7:0] r1; // rand_mode = ON
constraint cons1 // (from this) (constraint_mode = ON) (test.v:4)
{
 (r1 == 8'h1); // (test.v:5)
 (r1 == 8'h2); // (test.v:6)
}
```

---

## Support for Save and Restore Stimulus

While migrating from a previous release to a new release, you may like VCS to generate same stimulus or output in the new release as it is generated for the previous release, especially during the early phases of migration. But, VCS can generate different stimulus and output across releases due to:

- procedural flows controlled by non-deterministic functions, such as date, time, and so on. This completely depends on the design.
- randomize function calls, such as `obj.randomize()` or `std::randomize()`. This is due to the change in the random behavior of constraint solvers from previous release to the new release.

Therefore, VCS supports saving the stimulus of the design in one release and applying or restoring the saved stimulus on the same design in the new release. This helps you to get same stimulus across all releases.

## Use Model

The following is the use model for saving and restoring the same stimulus for the same design across all releases:

```
% simv +ntb_solver_replay=[save|restore]
 [+ntb_solver_replay_path=<path>]
```

Where,

`+ntb_solver_replay=save`

Saves the stimulus of a simulation.

`+ntb_solver_replay=restore`

Restores the stimulus of a simulation for the same design.

`+ntb_solver_replay_path=<path>`

This is optional. It allows you to specify the file path to save and read the stimulus. Without this option, the `+ntb_solver_replay=save` and the `+ntb_solver_replay=restore` options save and restore the stimulus to or from `./simv.replay` respectively.

## Limitations

The feature has the following limitations:

- in save and restore phases, you should use the same set of compile-time and runtime options (including random seed).
- You should compile or run the design in the same environment or flow. For example, if you use the partition compile flow while saving the stimulus, you should use the same partition compile flow to restore the saved stimulus.
- For any changes in the setup from save to restore phase, the correct result is not guaranteed.

## Constraint Guard Error Suppression

If a guard expression is false, and if there are no other errors during randomization, VCS suppresses errors in the implied expressions of guard constraints. For example, the following is a sample error message that VCS now suppresses:

```
Error-[CNST-NPE] Constraint null pointer error test_guard.sv, 27
Accessing null pointer obj.x in constraints.
Please make sure variable obj.x is allocated.
```

Guarded constraints are defined in the IEEE SystemVerilog LRM Std 1800™-2012 Section 13.4; especially sections 13.4.5, 13.4.6, and 13.4.12.

The VCS constraint solver does not distinguish between implication (see IEEE SystemVerilog LRM Std 1800™-2012 Section 13.4.5) and `if-else` constraints (IEEE SystemVerilog LRM Std 1800™-2012 Section 13.4.6). They are equivalent representations in the VCS constraint solver. These are called guarded constraints in this document.

Hence, the two formats shown in [Example 154](#) are equivalent inside the VCS constraint solver.

### *Example 154 Guarded Expressions*

```
if (a | b | c)
{
 obj.x == 10;
}
```

-or-

```
(a | b | c) -> (obj.x == 10);
```

In [Example 154](#), the expression inside the `if` condition (or the left side of the implication operator) is the guard expression. The remaining part of the expression (the right side of the implication operator) is the implied expression.

**Note:**

If there are other types of errors or conflicts, VCS does not guarantee suppression of those errors in the implied expression of the guard constraint.

The LRM states that the implication operator (or the `if-else` statement) should be at the top level of each constraint. Therefore, a constraint may have at most one guard (or one implication operator).

## Error Message Suppression Limitations

The constraint guard error message suppression feature has some limitations, as explained in the following sections:

- [Flattening Nested Guard Expressions](#)
- [Pushing Guard Expressions into Foreach Loops](#)

### Flattening Nested Guard Expressions

If there are multiple nested guards for a constraint, VCS combines them into one guard. For example, given the following code:

```
if (a)
{
 if (b)
 {
 if (c)
 {
 obj.x == 10;
 }
 }
}
```

VCS flattens the guard expression into the following equivalent code:

```
if (a && b && c)
{
 obj.x == 10;
}
```

In the above example, if `a` is false and `b` has an error (for example, a null address error), VCS still issues the error message.

## Pushing Guard Expressions into Foreach Loops

VCS pushes constraint guards into foreach loops. For example, if you have:

```
if (a | b | c)
{
 foreach (array[i])
 {
 array[i].obj.x == 10;
 }
}
```

VCS transforms it into the following equivalent code:

```
foreach (array[i])
{
 if (a | b | c)
 {
 array[i].obj.x == 10;
 }
}
```

In the above example, if `a | b | c` is false and `array` has an error (for example, a null address error), VCS still issues the error message.

## Support for Array and Cross-Module References in std::randomize()

VCS allows you to use Cross-Module References (XMRs) in class constraints and inline constraints, in all applicable contexts. Here, XMR means a variable with static storage (anything accessed as a global variable).

VCS `std::randomize()` support allow the use of arrays and XMRs as arguments.

VCS supports all types of arrays:

- fixed-size arrays
- associative arrays
- dynamic arrays
- multidimensional arrays
- smart queues

**Note:**

VCS does not support multidimensional, variable-sized arrays.

Array elements are also supported as arguments to `std::randomize()`.

VCS supports all types of XMRs:

- class XMRs
- package XMRs
- interface XMRs
- module XMRs
- static variable XMRs
- any combination of the above

You can use arrays, array elements, and XMRs as arguments to `std::randomize()`.

## Syntax

```
integer fa[3];
success= std::randomize(fa);
success= std::randomize(fa[2]);
success= std::randomize(pkg::xmr);
```

## Example

```
module test;
integer i, success;
integer fa[3];
initial
begin
 foreach(fa[i]) $display("%d %d\n", i, fa[i]);
 success = std::randomize(fa);
 foreach(fa[i]) $display("%d %d\n", i, fa[i]);
end
endmodule
```

When `std::randomize()` is called, VCS ignores any `rand` mode specified on class member arrays or array elements that are used as arguments. This is consistent with how `std::randomize()` is specified in the SystemVerilog LRM. This means that for purposes of `std::randomize()` calls, all arguments have `rand` mode ON, and none of them are `randc`.

## Error Conditions

If you specify an argument to a `std::randomize()` array element, which is outside the range of the array, VCS prints the following error message:

```
Error-[CNST-VOAE] Constraint variable outside array error
```

Random variables are not allowed as part of an array index.

If you specify an XMR argument in a `std::randomize()` call, and that XMR cannot be resolved, VCS issues an error message.

## Support for Cross-Module References in Constraints

You can use Cross-Module References (XMRs) in class constraints and inlined constraints. You can refer to XMR variables directly or by specifying the full hierarchical name, where appropriate. You can use XMRs for all data types, including scalars, enums, arrays, and class objects.

VCS supports all types of XMRs:

- class XMRs
- package XMRs
- interface XMRs
- module XMRs
- static variable XMRs
- any combination of the above

### Syntax

```
constraint general
{
 varxmr1 == 3;
 pkg::varxmr2 == 4;
}

c.randomize with { a.b == 5; }
```

### Examples

The following is an example of a module XMR:

```
// xmr from module
module mod1;
 int x = 10;
class cls1;
 rand int i1 [3:0];
 rand int i2;
constraint constr
{
 foreach(i1[a]) i1[a] == mod1.x;
}
endclass
```

```
cls1 c1 = new();
initial
begin
 c1.randomize() with {i2 == mod1.x + 5;};
end
endmodule
```

The following is an example of a package XMR:

```
package pkg;
 typedef enum {WEAK,STRONG} STRENGTH;
 class C;
 static rand STRENGTH stren;
 endclass

 pkg::C inst = new;
endpackage

module test;
 import pkg::*;
 initial
 begin
 inst.randomize() with {pkg::C::stren == STRONG;};
 $display("%d", pkg::C::stren);
 end
endmodule
```

### Functional Clarifications

XMR resolution in constraints (that is, choosing to which variable VCS binds an XMR variable) is consistent with XMR resolution in procedural SystemVerilog code. VCS first tries to resolve an XMR reference in the local scope. If the variable is not found in the local scope, VCS searches for it in the immediate upper enclosing scope, and so on, until it finds the variable.

If you specify an XMR variable that cannot be resolved in any parent scopes of the constraint/scope where it is used, VCS errors out and prints an error message.

### XMR Function Calls in Constraints

VCS supports XMR function calls in class constraints, inlined constraints, and std::randomize. You can refer to XMR functions with or without specifying the full hierarchical name. XMR functions can return and have as arguments all supported data types, including scalar data types, enums, arrays, and class objects.

---

## State Variable Index in Constraints

VCS supports the use of state variables as array indexes in constraints and inline constraints, in all applicable contexts. These state variables must evaluate to the same type required by the index type of the array to which they are addressed.

**Note:**

String-type state variables in array indexes are not supported.

VCS supports the set of expressions (operators and constructs) that also work with loop variables as array indices in constraints. The set of supported expressions is restricted in the sense that they must evaluate in the constraint framework.

---

## Runtime Check for State Versus Random Variables

VCS supports state variables for array indexes, but not random variables. Therefore, the tool performs runtime checks for the randomness of the variable. The randomness may be affected if the variable is aliased (due to object hierarchy, module hierarchy, or XMR). When this runtime check finds a random variable being used as an array index, the tool issues an error message.

To differentiate random variables versus state variables, VCS uses the following scheme:

- For `randomize` with a list of arguments (`std::randomize` or `obj.randomize`), variables or objects in the argument list are considered to be random. Variables or objects outside the list (and not aliased by the random objects) are considered to be state variables.
- For `randomize` without a list of arguments (`obj.randomize`) variables declared as non-random, or declared as random but with `rand mode OFF`, variables are considered to be state variables.

---

## Array Index

The variable (or supported expression) used for an array index must be an integral data type. If the value of the expression or the state variable evaluates out of bounds, comes to a negative index value and references a non-existent array member or contains `X` or `Z`, VCS issues a runtime error message.

---

## Using DPI Function Calls in Constraints

VCS supports calling DPI functions directly from constraints. These DPI function calls must be pure and cannot have any side effects, as per the SystemVerilog LRM. For more

information on DPI function call contexts (pure and non-pure), see IEEE SystemVerilog LRM Std 1800™-2012 Section 35.

The following are some examples of valid import DPI function declarations that you can call from constraints:

```
import "DPI-C" pure function int func1();
import "DPI-C" pure function int func2(int a, int b);
```

[Example 155](#) shows a pure DPI function in C.

#### *Example 155 Pure DPI Function in C*

```
#include <svdpi.h>

int dpi_func (int a, int b) {
 return (a+b); // Result depends solely on its inputs.
}
```

[Example 156](#) shows how to call a pure DPI function from constraints.

#### *Example 156 Invoking a Pure DPI Function from Constraints*

```
import "DPI-C" pure function int dpi_func(int a, int b);
class C;
 rand int ii;
 constraint cstr {
 ii == dpi_func(10, 20);
 }
endclass

program tb;
initial begin
 C cc;
 cc = new;
 cc.randomize();
end
endprogram
```

## Invoking Non-pure DPI Functions from Constraints

VCS issues an error message when it detects a call to any context DPI function or other import DPI function for which the context is not specified or the import property is not specified as `pure`. VCS issues this error even if the DPI function actually has no side effects. To prevent this kind of error, explicitly mark the DPI function import declaration with the `pure` keyword.

For example, running [Example 157](#) with the C code shown in [Example 155](#) results in an error because the import DPI function is not explicitly marked as `pure`.

**Example 157 Invoking a DPI Function Not Marked *pure* from Constraints.**

```
import "DPI-C" function int dpi_func(int a, int b);
// Error: Only functions explicitly marked as
// pure can be called from constraints

class C;
 rand int ii;
 constraint cstr {
 ii == dpi_func(10, 20);
 }
endclass

program tb;
initial begin
 C cc;
 cc = new;
 cc.randomize();
end
endprogram
```

Similarly, running [Example 158](#) with the C code shown in [Example 155](#) results in an error because `context` import DPI functions cannot be called from constraints.

**Example 158 Invoking a context DPI Function from Constraints**

```
import "DPI-C" context function int dpi_func(int a, int b);

// Error: Calling 'context' DPI function
// from constraint is illegal.

class C;
 rand int ii;
 constraint cstr {
 ii == dpi_func(10, 20);
 }
endclass

program tb;
initial begin
 C cc;
 cc = new;
 cc.randomize();
end
endprogram
```

Calling an import DPI function that is explicitly marked `pure` (as shown in [Example 155](#)) has undefined behavior if the actual implementation of the function does things that are not pure, such as:

- Calling DPI exported functions/tasks.
- Accessing SystemVerilog data objects other than the function's actual arguments (for example, via VPI calls).

For example, [Example 159](#) has undefined behavior (and may even cause a crash).

*Example 159 Non-pure DPI Function in C*

```
#include <stdio.h>
#include <stdlib.h>
#include "svdpi.h"

int readValueOfFile(char * file) {
 int result = 0;
 char * buf = NULL;
 FILE * fp = fopen(file, "r");

 // Read the content of the file in 'buf' here...
 ...

 if (buf) return strlen(buf);
 else return 0;
}

int dpi_func () {

 char * str = getenv("ENV_VAL_OF_A");
 int a = str ? atoi(str) : -1;
 int b = readValueOfFile("/some/file");
 int c;

 svScope scp = svGetScopeFromName("$unit");
 if (scp == NULL) {
 fprintf(stderr, "FATAL: Cannot set scope to $unit\n");
 exit(-1);
 }
 svSetScope(scp);

 c = export_dpi_func();
 return (a+b+c);
}
```

[Example 160](#) shows a DPI function marked `pure` that is actually doing non-pure activities. This results in an error.

### **Example 160 DPI Function Marked `pure` but Non-pure Activities**

```
import "DPI-C" pure function int dpi_func();
export "DPI-C" function export_dpi_func;

function int export_dpi_func();
 return 10;
endfunction

class C;
 rand int ii;
 constraint cstr {
 ii == dpi_func();
 }
endclass

program tb;
initial begin
 C cc;
 cc = new;
 cc.randomize();
end
endprogram
```

So, make sure that DPI functions called from the constraints explicitly use the `pure` keyword. Also make sure that the DPI function corresponding foreign language implementation is indeed pure (that is, it has no side effects).

## **Using Foreach Loops Over Packed Dimensions in Constraints**

VCS supports foreach loops over the following kinds of packed dimensions in constraints:

- [Memories with Packed Dimensions](#)
- [MDAs with Packed Dimensions](#)

You do not need to set any special compilation or runtime switches to make this work. VCS supports foreach loop variables for the entire packed dimensions of an array. For more information, see the section [The foreach Iterative Constraint for Packed Arrays](#).

### **Memories with Packed Dimensions**

You can use foreach loops over memories with single or multiple packed dimensions, as shown in the following examples.

#### **Single Packed Dimension**

```
class C;
 rand bit [5:2] arr [2];
```

```
constraint Cons {
 foreach(arr[i,j])
 arr[i][j] == 1;
}
endclass
```

## Multiple Packed Dimensions

```
class C;
 rand bit [3:1][5:2] arr [2];
 constraint Cons {
 foreach(arr[i,j,k])
 arr[i][j][k] == 1;
 }
endclass
```

## MDAs with Packed Dimensions

You can use foreach loops over MDAs with single or multiple packed dimensions, as shown in the following examples.

### Single Packed Dimension

```
class C;
 rand bit [5:2] arr [2][3];
 constraint Cons {
 foreach(arr[i,j,k])
 arr[i][j][k] == 1;
 }
endclass
```

### Multiple Packed Dimensions

```
class C;
 rand bit [-1:1][5:2] arr [2][3];
 constraint Cons {
 foreach(arr[i,j,k,l])
 arr[i][j][k][l] == 1;
 }
endclass
```

### Just Packed Dimensions

```
class C;
 rand integer arr1;
 rand bit [-1:0][5:2] arr2;
 constraint cons_1 {
```

```
// foreach over the packed dimensions;
foreach (arr1[i]) {
 if (i > 15) {
 arr1[i] dist {0:= 10, 1:= 90};
 } else {
 arr1[i] dist {0:= 90, 1:= 10};
 }
}
constraint Cons2 {
 foreach(arr2[i,j]) {
 arr2[i][j] == 1;
 }
}
endclass
```

VCS does not create implicit constraints that guarantee the array indexed by the variable (or expression) is valid. You must properly constrain or set the variable value so that the array is correctly addressed.

VCS also supports associative array indices. The indexes of these arrays may be integral data types or strings if the associative array is string-indexed. However, you cannot use expressions for associative arrays.

## The `foreach` Iterative Constraint for Packed Arrays

VCS has implemented `foreach` loop variables for the entire packed dimensions of an array in the constraint context.

In previous releases, a `foreach` loop for the dimensions of a multidimensional array in the constraint context required that at least one of the dimensions be unpacked. That restriction is removed, a multi-dimensional packed array in the constraint context is now fully supported.

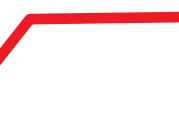
The following code example illustrates this implementation.

**Figure 166 The foreach Iterative Constraint for Packed Arrays**

```

program prog;
class my_class;
 rand reg [2][2][2][2] arr;
 constraint constr {
 foreach (arr[i,j,k,l]) {
 (i==0) -> arr [i][j][k][l] == 1;
 (i==1) -> arr [i][j][k][l] == 0;
 }
 }
endclass
endprogram

```



all dimensions packed

In previous releases at least one of the dimensions of MDA array needed to be unpacked.

This code example results in the following error message in previous releases:

```

Error-[NYI-UFAIFE] NYI constraint: packed dimensions
doc_ex.sv,9
prog, "this.arr"
 arr has only packed dimensions and no unpacked dimensions.
 Foreach over packed dimensions is supported if the object has at least
one
 unpacked dimension.

1 error

```

Now, entirely packed arrays in the constraint context are not an error condition and do not result in this error message.

## Randomized Objects in a Structure

VCS has implemented randomized objects in a structure. The following code example illustrates this implementation.

*Figure 167 Randomized Object in a Structure*

```

program test;

 class packet;
 randc int addr = 1;
 int crc;
 rand byte data [] = {1,2,3,4};
 endclass
 class packet_test;
 typedef struct {
 rand packet p1;
 } header;
 header hd;
 endclass

 function new();
 this.hd.p1 = new;
 endfunction

 packet_test pt = new;

 initial begin
 pt.randomize(hd);
 end
endprogram

```



**randomized object  
in a structure**

In previous releases declaring this class in a structure with the `rand` type-modifier keyword resulted in the following error message:

```

Error-[SV-NYI-CRUDST] Rand class object under structure
code_ex_rand_struct.sv, 10
"p1"
 Rand class objects which defined under structure is not yet supported.

1 error

```

This code example compiles and runs without any errors since `rand` class objects inside a structure are implemented.

## Support for Typecast in Constraints

You can use a cast (‘`’ operator on constraints as defined in the IEEE SystemVerilog LRM Std 1800™-2012 Section 6.24, “Casting”.

## Syntax

The following is the syntax for casting constraints.

```
constant_cast ::= casting_type ' (constant_expression)
cast ::= casting_type ' (expression)
casting_type ::= simple_type | constant_primary | signing
simple_type ::= integer_type | ps_type_identifier |
 ps_parameter_identifier
integer_type ::= bit | logic | reg | byte | shortint | int | longint |
 integer | time
```

## Description

The following are the details about typecasting constraints.

- VCS only supports variables and constants of integral types. As a result, typecasting constraints are applied only to integral or equivalent types (see IEEE SystemVerilog LRM Std 1800™-2012 Section 6.22.2).
- Built-in integer types can be cast to each other.
- Packed arrays, *structs*, *unions* and *enum* types can be cast to built-in integral types if their base types are integral types.
- For casting an enumerated type, if an enumerated value is outside the defined range of the *enum* type, then VCS issues a constraint inconsistency message (through the Solver).
- Casting from a non-integral types (*real*, *string*, etc.) to integral types is not supported. VCS issues a compile-time error message.
- Casting from user-defined types (*class*, unpacked *struct*, unpacked *unions*, etc.) to integral types is not supported. VCS issues a compile-time error message.

## Examples

```
// First example.
typedef struct packed signed {
 bit [7:0] a;
 byte b;
 shortint c;
} p1;
rand p1 p;
rand int q;
constraint c1 { int'(p) == q; };
```

For example, *p* is a 32-bit packed signed *struct* and is randomized to the value of 'h815F\_8D74. The members of this packed *struct* are *p.a* = 8'h81, *p.b* = 8'h5F, *p.c*

= 8'h8D74 (or -29324 as it is signed). Because of the equality constraint, *q* takes on the value of 8'h815F\_8474 or -2124444300 as *int* is a signed data type.

```
// Second example.
class C;
 typedef enum {
 red=0,
 yellow=5,
 green
 } light_t;

 rand light_t x;
 rand int y;

 constraint c0 { x == light_t'(y); }
endclass

C obj = new;
obj.randomize();
```

Note, in the previous example (“Second example”), *y* is always 0, 5, or 6. The following would produce a constraint solver failure message:

```
obj.randomize() with { y == 10; };
```

Following is a third example.

```
// Third example.
class cfg;
 rand bit en[16];
 constraint only_enable_4_GOOD {
 // Intent: only 4 of the 16 of the 1-bit elements are 1'b1
 en.sum() with (int'(item)) == 4;
 }
endclass
program automatic test;
 cfg obj = new;
 initial obj.randomize();
endprogram
```

According to the LRM, *sum()* returns a single value of the same type as the array element type; or, if specified, the expression in the ‘with’ clause. In the previous case, *sum()* will return the type ‘*int*’ because of the *with* clause, and not a single ‘bit’ as the type of the individual array elements. Without the use of the *with* clause and casting the item from a ‘bit’ to an ‘*int*’ type, there would have been a constraint failure as *sum()* would have returned a value of type ‘*bit*’, and no 1-bit value would be equal to 4.

```
// Fourth example.
class A;
 rand int x;
 constraint c {
 x >= 0;
```

```

 x < ((41'(2 ** 40)) - 1024);
 }
endclass

program automatic test;
 A obj = new;
 initial obj.randomize();
endprogram

```

The cast to increase the size of the expression ( $2^{**40}$ ) from 30 to 41 is needed so that  $2^{**40}$  is treated as the value 41'h100000000000. Otherwise, the expressions in the second constraint will be evaluated as 32-bit values, and  $(2^{**40})$  would have been evaluated to 32'h0.

## Strings in Constraints

You can compare string type state variables and string literals with the logical equality == and inequality != operators.

All other uses of strings in constraints, such as string concatenation, string methods (for example `str.substr()`), string replication and casting to or from strings are unsupported.

The following is an example of the supported use of strings in constraints:

```

bit [31:0] p = "string_lit";
string p2 = "string_var";
rand int flg;

constraint constr0 {
 if (p == "string_lit" && p2 != "string_var") flg == 1;
 else flg == 10;
}

```

## Support for Dynamic MDA Elements in the Randomize Call Arguments

VCS supports the randomization of dynamic MDA elements specified in the arguments of the randomized calls. That is, for randomize calls `std::randomize()` or `obj.randomize()` with an array type variable, VCS considers the variable and its array elements as random variables. For example, if the randomize call `std::randomize(xvar)` is used as follows:

```

class C;
 int xvar[] [4];
endclass;
C obj = new;
initial begin

```

```
std::randomize(xvar) with {xvar.size == 10;};
end
```

Then, `xvar`'s size and its elements are considered random variables and randomized.

This feature allows you to use dynamic MDA elements in the `std::randomize()/obj.randomize()` call.

You can use the following array type variables in the

`std::randomize()/obj.randomize()` call:

- Dynamic MDA of basic data types (int, bit, reg, logic, byte, or real)
- Dynamic MDA of struct data type
- Dynamic MDA of enum data type

## Supported Dynamic MDA Types

- The dynamic MDA can be any type of array with at least 2 dimensions, including variable/fixed size array, associated array, or queue. For example:
  - `int array[][][2];`
  - `int array[2][${}];`
  - `int array[]["string"];`
  - `int array[][[2][${}][string]];`

The dynamic dimension must be unpacked dimension.
- The dynamic MDA element type can be an integer, packed vector, enumeration, real, or struct type. For example:
  - `int array[][][];`
  - `bit[7:0] array[][][];`
  - `bit[8][4] array[][][];`
  - `enum {a, b, c} array[][][];`
  - `real array[][][];`
  - `struct type array[][][];`

## Key Points to Note

- For an array of a struct type, if the element of the struct is a state type, then it is kept as the state variable even if the array is in the randomize call.
- For the nested struct type as follows:

```
typedef struct {
 int a;
 rand int b;
} ST1;
typedef struct {
 ST1 a;
 rand ST1 b;
} ST
ST array[][];
```

During randomization, only the array elements with struct members such as `array[0][0].b.b` are considered as rand type. Array element like `array[0][0].a.b` or `array[0][0].b.a` is considered as state type.

## Supported Usage Scenarios

Consider the following dynamic MDA variable:

```
int array[2][3][];
```

The following usages are supported for this array variable:

- Using the whole array in `std::randomize`:

```
int array[2][3][];
std::randomize(array) with { ... };
```

- Using the whole array in `obj.randomize`:

```
class cls;
 int array[2][3][];
endclass

cls obj = new;

obj.randomize(array) with { ... };
```

- Using the subarray (constant index or state index variable) in `std::randomize()` as follows. This usage is not supported for `obj.randomize`.

```
std::randomize(array[0], array[idx]) with { ... };
```

- Using the array in XMR context:

```
std::randomize(a.b[0].array) with { ... };
obj.randomize(a.b.array) with { ... };
```

## Limitations

The following are the limitations of this feature:

- Dynamic MDA of class/struct type is not supported.
- Random index in dynamic MDA is not supported.

## Support for Dynamic MDA in the Inside Expression Used in Constraints

VCS supports specifying dynamic multi-dimensional arrays in the `inside` expression used in constraint context.

For example, VCS supports the following dynamic MDA used in the `inside` expression:

```
module test;
int array[2][];
int x;
initial begin
 std::randomize(x) with { x inside {array}; };
 $display(x);
end
endmodule
```

## Supported Dynamic MDA Types

- The dynamic MDA can be any type of array with at least 2 dimensions, including variable/fixed size array, associated array, or queue. For example:

- `int array[][][2];`
- `int array[2][${}];`
- `int array[][],string];`
- `int array[][],2][${}][string];`

The dynamic dimension must be unpacked dimension.

- The dynamic MDA element type can be an integer, packed vector, enumeration, real, or struct type. For example:

- int array[][];
  - bit[7:0] array[][];
  - bit[8][4] array[][];
  - enum {a, b, c} array[][];
  - real array[][];

---

## Supported Usage Scenarios

The following usages are supported for this feature:

- Using in `std::randomize`:

```
std::randomize(x) with { x inside {array}; };
```

- Using in the constraint block with class:

```
class cls;
 rand int x;
 int array[][];
 constraint cs {
 x inside {array};
 }
endclass
```

- Using in other constraint expression:

```
(x inside {array}) -> a == b;
(x1 inside {array1}) && (x2 inside {array2})
```

- Using with constant/variable index:

```
x inside {array[2]};
x inside {array[i]};
x inside {array[a+b]};
```

In the variable index case, the index variable can only be a state variable, not a rand variable.

- Using in the XMR context:

```
x inside {a.b.array}
```

## Limitations

The following are the limitations of this feature:

- Dynamic MDA of struct and class types are not supported.
- Random index in dynamic MDA is not supported.

## Support for Array Reduction Methods over MDAs in Constraints

The array reduction method is applied to any unpacked array of integral values to reduce the array to a single value. In procedural statements, the unpacked array is expanded level by level and MDA is handled with a nested array reduction method call. This behavior complies with SystemVerilog LRM.

The behavior of the array reduction method over MDA in constraints is consistent with the array reduction method used in procedural statements. That is, the array item used in the array reduction method is expanded one level for unpacked dimensions for one method call, and a nested array reduction method call is required for MDA.

Consider the following code:

*Example 161 test.v*

```
program test;
 class cls;
 rand bit [3:0] mda[4][4];
 constraint cons{
 mda.sum(item1) with (item1.sum(item2) with (item2.index != 0 ?
item2 : 0)) == 100;
 }
 function void post_randomize;
 $display("mda = %p",mda);
 assert(mda.sum(item1) with (item1.sum(item2) with (item2.index !=
0 ? item2 : 0)) == 100);
 endfunction
 endclass

 initial begin
 cls obj = new();
 obj.randomize;
 end
endprogram
```

Compile command line:

```
% vcs -sverilog -full64 test.sv -R
```

Following is the output:

```
mda = '{ {'h3, 'h3, 'h4, 'he}, {'hf, 'h4, 'hd, 'hd}, {'h1, 'h7, 'hd,
 'hc}, {'ha, 'h8, 'h8, 'h1}}
```

The following table describes the behavior of array methods in constraints using the `with` clause:

| Method                  | Description                                                                                                                                                                                                              |
|-------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <code>.sum()</code>     | Returns the sum of all the array elements. If a <code>with</code> clause is specified, returns the sum of the values yielded by evaluating the expression for each array element.                                        |
| <code>.product()</code> | Returns the product ( <code>*</code> ) of all the array elements. If a <code>with</code> clause is specified, returns the product of the values yielded by evaluating the expression for each array element.             |
| <code>.and()</code>     | Returns the bitwise AND ( <code>&amp;</code> ) of all the array elements. If a <code>with</code> clause is specified, returns the bitwise AND of the values yielded by evaluating the expression for each array element. |
| <code>.or()</code>      | Returns the bitwise OR ( <code> </code> ) of all the array elements. If a <code>with</code> clause is specified, returns the bitwise OR of the values yielded by evaluating the expression for each array element.       |
| <code>.xor()</code>     | Returns the bitwise XOR ( <code>^</code> ) of all the array elements. If a <code>with</code> clause is specified, returns the bitwise XOR of the values yielded by evaluating the expression for each array element.     |

## Limitations

The following are the limitations of this feature:

- Array comparison is not supported outside the `with` clause even when it is in a constraints context.
- Struct type array comparison is not supported.
- Variable-sized MDA comparison is not supported.

## SystemVerilog LRM 1800™-2012 Update

The SystemVerilog constructs in this update are as follows:

- [Using Soft Constraints in SystemVerilog](#)
- [Unique Constraints](#)

---

## Using Soft Constraints in SystemVerilog

Input stimulus randomization in SystemVerilog is controlled by user-specified constraints. If there is a conflict between two or more constraints, the randomization fails.

To solve this problem, you can use soft constraints. Soft constraints are constraints that VCS disables if they conflict with other constraints.

VCS uses a deterministic, priority-based mechanism to disable soft constraints. When there is a constraint conflict, VCS disables any soft constraints in reverse order of priority (that is, the lowest priority soft constraint is disabled first) until the conflict is resolved. The following sections explain how to use soft constraints with VCS:

- [Using Soft Constraints](#)
- [Soft Constraint Prioritization](#)
- [Soft Constraints Defined in Classes Instantiated as rand Members in Another Class](#)
- [Soft Constraints Inheritance Between Classes](#)
- [Soft Constraints in AOP Extensions to a Class](#)
- [Soft Constraints in View Constraints Blocks](#)
- [Discarding Lower-Priority Soft Constraints](#)

## Using Soft Constraints

Use the `soft` keyword to identify the soft constraints. Constraints that are not defined as soft constraints are hard constraints. The following [Example 162](#) shows a soft constraint:

*Example 162 Soft Constraint*

```
class A;
 rand int x;
constraint c1 {
 soft x > 2; // soft constraint
}
endclass
```

[Example 163](#) shows a hard constraint.

*Example 163 Hard Constraint*

```
class A;
 rand int x;
constraint c1 {
 x > 2; // hard constraint
}
endclass
```

## Soft Constraint Prioritization

VCS determines the priorities of soft constraints according to the set of rules described in this section. In general, VCS assigns increasing priorities to soft constraints as they climb the following list:

- Class parents in the inheritance graph
- Class members
- Soft constraints in the class itself
- Soft constraints in any `extends` blocks applied to a class

In this schema, soft constraints in any `extends` blocks applied to a class are assigned the highest priority.

The following notation is used to describe the priority of a given soft constraint (SC):

`priority(SCx)`: If the following is true:

`priority(SC2) > priority(SC1)`

Then VCS disables constraint SC1 before constraint SC2 when there is a conflict.

### Within a Single Class

VCS assigns soft constraints declared within a class, thus increasing the priority by order of declaration. The soft constraints that appear later in the class body have higher priority than the soft constraints that appear earlier in the class body.

For example, in [Example 164](#), `priority(SC2) > priority(SC1)`.

*Example 164 SC2 Higher Priority than SC1*

```
class A;
 rand int x;
constraint c1 {
 soft x > 10; // SC1
 soft x > 5; // SC2
}
endclass
```

In [Example 165](#), `priority(SC2) > priority(SC1)`.

*Example 165 SC2 Higher Priority than SC1*

```
class A;
 rand int x;
constraint c1 {
 soft x > 10; // SC1
}
constraint c2 {
```

```
soft x > 5; // SC2
}
endclass
```

## Soft Constraints Defined in Classes Instantiated as rand Members in Another Class

VCS assigns soft constraints declared within rand members of classes increasing the priority by order of member declaration. In [Example 166](#), the soft constraints contributed by `C.objB` are of higher priority than the soft constraints contributed by `C.objA`, because `C.objB` is declared after `C.objA` within class `C`.

[Example 166](#) also shows why some soft constraints are dropped, instead of honored, because of the relative priorities assigned to soft constraints are as follows:

- `// objC.x = 4` because SC6 is honored.
- `// objC.objA.x = 4` because `priority(SC4) > priority(SC1)`.

Here, SC4 is honored and SC1 is dropped. If SC1 were not dropped, it might have caused a conflict because `objA.x` cannot be 4 (`objC.x` in SC4) and 2 (SC1) at the same time.

- `// objC.objB.x = 5` because `priority(SC5) > priority(SC3) > priority(SC2)`.

Here, SC5 is honored and SC3 is dropped (otherwise, SC3 might conflict with SC5). SC2 is honored because it does not conflict with SC5 by honoring SC2, `objC.objB.x = 5`.

### *Example 166 SC3 Higher Priority than SC2 and SC1*

```
class A;
 rand int x;
 constraint c1 { soft x == 2; } // SC1
endclass

class B;
 rand int x;
 constraint c2 { soft x == 5; } // SC2
 constraint c3 { soft x == 3; } // SC3
endclass

class C;
 rand int x;
 rand A objA;
 rand B objB;
 constraint c4 { soft x == objA.x; } // SC4
 constraint c5 { soft objA.x < objB.x; } // SC5
 constraint c6 { soft x == 4; } // SC6
function
 new(); objA = new; objB = new;
```

```

endfunction
endclass

program test;
 C objC;
 initial begin
 objC = new;
 objC.randomize();
 $display(objC.x); // should print "4"
 $display(objC.objA.x); // should print "4"
 $display(objC.objB.x); // should print "5"
 end
endprogram

```

For array members where objects are allocated prior to randomization, priorities are assigned in increasing order by position in the array, where the soft constraints in element N have lower priority than the soft constraints in element N+1.

For array members where the objects are allocated during randomization, all soft constraints in allocated objects and their base classes and member classes have the same priority.

## Soft Constraints Inheritance Between Classes

Soft constraints in an inherited class have a higher priority than soft constraints in its base class. For example, in [Example 167](#), priority(SC2) > priority(SC1).

### *Example 167 SC2 Higher Priority than SC1*

```

class A;
 rand int x;
 constraint c1 {
 soft x > 2; // SC1
 }
endclass

class B extends A;
 constraint c1 {
 soft x > 3; // SC2
 }
endclass

```

## Soft Constraints in AOP Extensions to a Class

As defined in "Aspect Oriented Extensions", constraint blocks in AOP extensions modify the class definition as an AOP Introduction directive. When there are multiple AOP extensions to the same class, the AOP precedence must be considered. The AOP precedence defines the order in which introductions to a class are added to the class. First, symbols introduced by an AOP extension with a higher precedence are appended to the class. Subsequently, same AOP extensions are appended to the class in the order they appear in the extension.

For example,

```
class A;
 rand int x;
 constraint c1 {
 soft x == 1; // SC1
 }
endclass

extends A_aop1(A);
 constraint c2 {
 soft x == 2; // SC2
 }
endextends
```

This means that the new constraint block `c2` is added as a new symbol in the original class definition as a new member. As a result, class `A` gets modified as follows:

```
class A;
 rand int x;
 constraint c1 {
 soft x == 1; // SC1
 }
 constraint c2 { // from A_aop1(A)
 soft x == 2; // SC2
 }
endclass
```

After the AOP introduction to the class definition, VCS assigns priorities to multiple soft constraints in the modified class. In this case, as the constraint block `c2` is declared later in the modified class, the priority of the soft constraint `SC2` is higher than that of the soft constraint `SC1`, or  $\text{priority}(\text{SC2}) > \text{priority}(\text{SC1})$ . As a result, `x` is randomized to 2 in the example.

Consider another example, where soft constraints are added on the same random variable in multiple AOP extensions to the same class.

For example:

```
class A;
 rand int x;
 constraint c1 {
 soft x == 1; // SC1
 }
endclass

extends A_aop2(A);
 constraint c2 {
 soft x == 2; // SC2
 }
 constraint c3 {
 soft x == 3; // SC3
 }
endextends
```

```

}
endextends

extends A_aop4(A);
constraint c4 {
 soft x == 4; // SC4
}
endextends

extends A_aop5(A);
constraint c5 {
 soft x == 5; // SC5
}
endextends

```

The last AOP extension `A_aop5` of the class `A` has the highest precedence as it is declared last. As the introduction is added to the class in the order of the precedence, this becomes the content of the modified class as follows:

```

class A;
 rand int x;
 constraint c1 {
 soft x == 1; // SC1
 }

 // A_aop5(A) - highest AOP precedence gets added to the class first
 constraint c5 {
 soft x == 5; // SC5
 }

 // A_aop4(A) - next highest AOP precedence gets added to the class next
 constraint c4 {
 soft x == 4; // SC4
 }

 // A_aop2(A) - lowest AOP precedence gets added to the class last
 // but within the same AOP extension, the constraint blocks c2 and c3
 // are added in the order of declaration.
 constraint c2 {
 soft x == 2; // SC2
 }
 constraint c3 {
 soft x == 3; // SC3
 }
endclass

```

VCS then computes the priorities for the soft constraints based on this modified class: as follows:

`priority(SC3) > priority(SC2) > priority(SC4) > priority(SC5) > priority(SC1)`

In this example, `x` is randomized to 3. It is noted that constraint blocks from the highest AOP precedence are added to the class first. Soft constraints from the highest AOP precedence may have an overall lower soft constraint priority.

The following is the original class definition for the class `A`:

```
class A;
 rand int x;
 constraint c1 {
 soft x == 1; // SC1
 }
endclass
```

Few modified examples with the `dominate_list` along with the resulting soft priority assignments and solver results for the random variable `x` are listed in the [Table 45](#).

## Soft Constraints in View Constraints Blocks

VCS assigns soft constraints within a view constraint block increasing priority by order of declaration. Soft constraints that appear later have higher priority than those that appear earlier. For example, in [Example 168](#), `priority(SC3) > priority(SC2) > priority(SC1)`.

### *Example 168 SC3 Higher Priority than SC1*

```
class A;
 rand int a;
 rand int b;
 constraint c1 {
 soft a == 2; // SC1
 }
endclass

A objA;
objA.randomize () with {
 soft a > 2; // SC2
 soft b == 1; // SC3
}
```

## Discarding Lower-Priority Soft Constraints

You can use a `disable soft` constraint to discard lower-priority soft constraints, even when they are not in conflict with other constraints (see [Example 169](#)).

*Table 45 Examples with the Dominates\_list*

| Dominates_list variations                                                                                                                                                                                                                                                     | Resulting AOP precedence and soft constraint priority                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             | Solver result for x |
|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------|
| <pre>extends A_aop2(A) dominates (A_aop5); constraint c2 { soft x == 2; // SC2 } constraint c3 { soft x == 3; // SC3 } endextends extends A_aop4(A); constraint c4 { soft x == 4; // SC4} endextends extends A_aop5(A); constraint c5 { soft x == 5; // SC5} endextends</pre> | <ul style="list-style-type: none"> <li>Precedence set by dominates: <math>A_{aop2} &gt; A_{aop5}</math></li> <li>No specific precedence set between <math>A_{aop2}</math> and <math>A_{aop4}</math>. Therefore, the order of declaration depends on the precedence order between the two as follows: <math>A_{aop4} &gt; A_{aop2}</math></li> <li>Overall precedence order is set as: <math>A_{aop4} &gt; A_{aop2} &gt; A_{aop5}</math></li> </ul> <p>This means the soft constraint from <math>A_{aop5}</math> is appended last to the class A, making this soft constraint (SC5) the highest soft priority.</p> | 5                   |
| <pre>extends A_aop2(A) dominates (A_aop4); constraint c2 { soft x == 2; // SC2 } constraint c3 { soft x == 3; // SC3 } endextends extends A_aop4(A); constraint c4 { soft x == 4; // SC4} endextends extends A_aop5(A); constraint c5 { soft x == 5; // SC5} endextends</pre> | <ul style="list-style-type: none"> <li>Precedence set by dominates: <math>A_{aop2} &gt; A_{aop4}</math></li> <li>No specific precedence set between <math>A_{aop2}</math> and <math>A_{aop5}</math>. Therefore, the order of declaration depends on the precedence order between the two as follows: <math>A_{aop5} &gt; A_{aop2}</math></li> <li>Overall precedence order is set as <math>A_{aop5} &gt; A_{aop2} &gt; A_{aop4}</math></li> </ul> <p>This means the soft constraint from <math>A_{aop4}</math> is appended last to the class A, making this soft constraint (SC4) the highest soft priority.</p>  | 4                   |

*Table 45 Examples with the Dominates\_list (Continued)*

| Dominates_list variations                                                                                                                                                                                                                                                   | Resulting AOP precedence and soft constraint priority                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                            | Solver result for x |
|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------|
| <pre>extends A_aop2(A); constraint c2 { soft x == 2; // SC2 } constraint c3 { soft x == 3; // SC3 } endextends extends A_aop4(A) dominates (A_aop5; constraint c4 { soft x == 4; // SC4} endextends extends A_aop5(A); constraint c5 {soft x == 5; // SC5} endextends</pre> | <ul style="list-style-type: none"> <li>Precedence set by dominates: A_aop4 &gt; A_aop5.</li> <li>No specific precedence set between A_aop2 and A_aop5. Therefore, the order of declaration depends on the precedence order between the two as follows: A_aop5 &gt; A_aop2.</li> <li>Overall precedence order is set as: A_aop4 &gt; A_aop5 &gt; A_aop2.</li> </ul> <p>This means the soft constraints from A_aop2 is appended last to the class A. Within the same A_aop2 extension, the order of introduction of new constraint blocks (c2 and c3) is the same as the order of declaration. Thus, it makes the soft constraint from the constraint block c3(SC3) the highest soft priority.</p> | 3                   |

#### *Example 169 Discarding Lower-Priority Soft Constraints*

```
class A;
rand int x;
constraint A1 {soft x == 3;}
constraint A2 {disable soft x;} // discard soft constraints
constraint A3 {soft x inside {1, 2);}
endclass
initial begin
A a= new();
a.randomize();
end
```

In [Example 169](#), constraint A2 tells the solver to discard all soft constraints of lower priority on random variable x. This results in constraint A1 being discarded. Now, only the last constraint (A3) needs to be honored. This example results in random variable x taking the values 1 and 2.

A disable soft constraint causes lower-priority soft constraints to be discarded even when they are not in conflict with other constraints. This feature allows you to introduce fresh soft constraints that replace default values specified in preceding soft constraints (see [Example 170](#)).

*Example 170 Specifying Fresh Soft Constraints*

```
class B;
rand int x;
constraint B1 {soft x == 5;}
constraint B2 {disable soft x; soft x dist {5, 8};}
endclass
initial begin
B b = new();
b.randomize();
end
```

In [Example 170](#), the `disable soft` constraint preceding the `soft dist` in block `B2` causes the lower-priority constraint on variable `x` in block `B1` to be discarded. Now, the solver assigns the values 5 and 8 to `x` with equal distribution (the result from the fresh constraint: `soft x dist {5,8}`).

Compare the behavior of [Example 170](#) with [Example 171](#), where the `disable soft` constraint is omitted.

*Example 171 Specifying Additional Soft Constraints*

```
class B;
rand int x;
constraint B1 {soft x == 5;}
constraint B3 {soft x dist {5, 8};}
endclass
initial begin
B b = new();
b.randomize();
end
```

In [Example 171](#), the `soft dist` constraint in block `B3` can be satisfied with a value of 5, so the solver assigns `x` the value 5. If you want the distribution weights of a `soft dist` constraint to be satisfied regardless of the presence of lower-priority soft constraints, you should first use a `disable soft` constraint to discard those lower-priority soft constraints.

*Example 172 Usage of Random Variable in Guard Expression*

```
program tb;
class cls;
rand int x;
rand bit cond;

constraint C1 {soft x == 3;}
constraint C2 {(cond) -> disable soft x;}
constraint C3 {x inside {1, 2};}
endclass

initial begin
cls obj = new;
obj.randomize;
```

```
 end
endprogram
```

In Example 172, `cond` is a random variable that is used in the guard expression of disable soft constraint. VCS displays the following error message:

```
Error-[CNST-NYI-DSCNS] Disable soft constraint not supported
test.v, 12
 The variable 'cond' is not a state variable.
 Only state variables can be used in guard expressions of disable soft
constraints.
```

## Unique Constraints

VCS has implemented unique constraints as specified in IEEE SystemVerilog LRM Std 1800™-2012 Section 18.5.5 “Uniqueness constraints”.

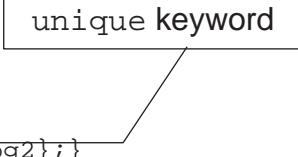
A unique constraint, specified with the `unique` keyword, specifies that a group of random variables after randomization have different (or unique) values.

*Figure 168 Unique Constraint*

```
module mod;
 class myclass;
 rand logic [1:0] log0, log1, log2;
 constraint constr1 { unique {log0,log1,log2}; }
 endclass

 myclass mcl = new;

 initial
 begin
 #25 if (mcl.randomize() == 1)
 $display("\n\n at %t log0=%0b log1=%0b log2=%0b \n",
 $time,mcl.log0,mcl.log1,mcl.log2);
 end
endmodule
```



The `$display` system task displays the following:

```
at 25 log0=10 log1=11 log2=1
```

None of the random variables has the same value after randomization.

The SystemVerilog LRM has the following limitations on the random variables in the unique constraint:

- The random variables must be either:
  - a scalar equivalent types or
  - arrays whose leaf elements are equivalent types
- The random variables cannot be `randc` variables.

If the constraint solver cannot find unique values for the variables in the group, such as if the three variables in [Figure 168](#) were scalar variables, so they could not have unique values, VCS displays the following error message:

```
Error-[CNST-CIF] Constraints inconsistency failure
exp3.sv, 13
 Constraints are inconsistent and cannot be solved.
 Please check the inconsistent constraints being printed
```

## Enhancement to the Randomization of Multidimensional Array Functionality

VCS supports MDA with variable size dimensions. As defined in the IEEE SystemVerilog LRM Std 1800™-2012 Section 7.4.2 “Unpacked arrays”, unpacked or dynamic arrays can be made up of any data type. Arrays can have more than one dimension. These array of arrays are called multidimensional arrays. Therefore, you can extend the semantics of a single-dimensional array to MDA.

### Syntax

```
rand int m [][];
m.size() constrains the first dimension (say, number of rows)
m[0].size() constrains the second dimension for the first row
m[1].size() constrains the second dimension for the second row
```

### *Example 173 Example of MDA for Variable Size Dimensions*

```
rand int m [][];
rand int m_length;
constraint cst { m.size() == m_length; m_length inside {[1:3]}; }
constraint cst2 {
 (m_length == 1) -> m[0].size == 5;
 (m_length == 2) -> { m[0].size == 3; m[1].size == 4; }
 (m_length == 3) -> { m[0].size == 2; m[1].size == 5; m[2].size == 1; }
}
```

- If `m_length` is set to 1, `m_data` structure may look like: [0] \* \* \* \* \*
- If `m_length` is set to 2, `m_data` structure may look like [0] \* \* \* [1] \* \* \* \*
- If `m_length` is set to 3, `m_data` structure may look like [0] \* \* [1] \* \* \* \* [2] \*

You can combine the dynamic array dimension with the fixed or associative array and queue dimensions. Any unpacked dimension in an array declaration can be a dynamic array dimension. For example:

```
rand int data1[][][4][];
rand int data2[$][2];
rand int data3[string][];
```

#### *Example 174 Example of MDA for Array Method Calls and Set Membership*

```
program tb;
 class cls;
 rand bit [7:0] mda1[][$];
 rand bit [7:0] a, b, c;

 constraint cons1 {
 // size constraints
 mda1.size == 2;
 foreach (mda1[i,])
 mda1[i].size inside {3,4,5,6};

 // array methods on rand MDA members
 foreach (mda1[i,])
 mda1[i].sum with (int'(item)) <= 100;

 // rand MDA members with the 'setmembership' operator
 a inside {mda1[0], 1, 2, 3};
 b inside {mda1[0][0], 4, 5, 6};
 }
 endclass

 initial begin
 cls obj = new;
 obj.randomize;
 end
endprogram
```

## Limitation

The feature has the following limitation:

- Randomization of MDA is not supported with `std::randomize` calls.

## Supporting Random Array Index

You can use random variable in array index expressions. You can use the random array index in fixed-size array, dynamic array, and smart queue arrays.

The following example shows the usage of random array index:

*Example 175 Usage of random array index*

```
program test;
 class cls;
 rand bit [7:0] indx1, indx2;
 rand bit [7:0] arr1[7:0], arr2[];

 constraint cons1 {
 indx1 inside {0,2,4,6};

 // rand variable 'indx1' used as the array index
 arr1[indx1] == 11;
 }

 constraint cons2 {
 indx2 inside {0, 1};
 arr2.size inside {[2:10]};

 // rand variable 'indx2' used as the array index
 arr2[indx2] == 22;
 }
 endclass

 initial begin
 cls obj = new;
 obj.randomize;
 end
endprogram
```

## Random Array Index Enhancements

VCS supports the following usages in random array index expression:

- Function call in an array index expression
- Dynamic MDA with loop variable in an array index expression
- Random array indexed sub-array in the unique/inside constraint

These enhancements help you with ease of migration, to write efficient code, and to avoid bugs. The following sections describe these enhancements in detail:

- [Function Call in Array Index Expression](#)
- [Dynamic MDA With Loop Index Variable in Array Index Expression](#)
- [Random Array Indexed Sub-Array In Unique/Inside Constraints](#)

## Function Call in Array Index Expression

You can use a function call in an array index expression.

*Example 176 test.v*

```
module test;
 class cls;
 rand int arr[1:5];
 int l1 = 1;
 int l2 = 1;
 function int foo_idx3(int a, int b);
 return (a) * (b);
 endfunction;
 constraint c1{arr[foo_idx3(l1, l2)] == 15;};
 endclass
 cls obj;
 initial begin
 obj = new();
 obj.randomize();
 $display("arr[1] = %d", obj.arr[obj.foo_idx3(obj.l1, obj.l2)]);
 end
endmodule
```

Execute the following commands:

```
% vcs -sverilog test.v
% simv
```

Following is the simulation output:

```
arr[1] = 15
```

### Limitation

- Rand variables as function arguments are not supported.

## Dynamic MDA With Loop Index Variable in Array Index Expression

You can use dynamic MDA with loop index variable in an array index expression.

**Example 177 test.v**

```
module mdl;
 class cls;
 rand int arr1[4:0];
 rand int dyn_arr[] [0:4];
 constraint c1{ (dyn_arr.size) == 5; }
 constraint c2{
 foreach (dyn_arr[i,j]){
 arr1[dyn_arr[i][j]] inside {[6:12]};}}
 endclass

 cls obj;
 initial begin
 obj = new();
 obj.randomize();
 $display("arr1 = %p",obj.arr1);
 end
endmodule
```

Execute the following commands:

```
% vcs -sverilog test.v
% simv
```

Following is the simulation output:

```
arr1 = '{12, 12, 8, 7, 8}
```

## Random Array Indexed Sub-Array In Unique/Inside Constraints

You can now use random array indexed sub-array in the unique/inside constraint expressions.

**Example 178 test.v**

```
module mdl;
 class cls;
 rand bit [3:0] arr[3:0][2:0];
 rand int x,r1,r2;
 constraint cons1{x inside {arr[r1+r2]};}
 constraint cons2{unique{arr[r1],arr[r2]};}
 endclass

 cls obj;
 initial begin
 obj = new();
 obj.randomize();
 $display("x = %d",obj.x);
 $display("arr[r1+r2] = %p",obj.arr[obj.r1+obj.r2]);
 $display("arr[r1] = %p",obj.arr[obj.r1]);
 $display("arr[r2] = %p",obj.arr[obj.r2]);
```

```
 end
endmodule
```

Execute the following commands:

```
% vcs -sverilog test.v
% simv
```

Following is the simulation output:

```
x = 4
arr[r1+r2] = '{'ha, 'h0, 'h4}
arr[r1] = '{'hf, 'hd, 'he}
arr[r2] = '{'ha, 'h0, 'h4}
```

### Limitation

- This feature is supported only for MDA of fixed size array.

### Limitation

The feature has the following limitation:

- Nested indices are not supported.

## Supporting System Function Calls

VCS supports usage of the following system function calls:

- [\\$size\(\) System Function Call](#)
- [\\$clog2\(\) System Function Call](#)

### \$size() System Function Call

VCS supports usage of \$size() system function call in the constraint code.

#### *Example 179 Example for the usage of \$size()*

```
program test;
 class cls;
 rand bit [7:0] arr1[1:0];
 rand bit [7:0] arr2[] [2:0][3:0];
 rand bit [7:0] size1, size2, size3;

 constraint cons_size {
 arr2.size == 2;
 // returns 'arr1' dimension size as '2'
```

```

 size1 == $size(arr1);

 // returns 'arr2' second dimension size as '3'
 size2 == $size(arr2[0]);

 // returns 'arr2' third dimension size as '4'
 size3 == $size(arr2[0], 2);
 }
endclass

initial begin
 cls obj = new;
 obj.randomize;
 $display("size1 = %0d, size2 = %0d, size3 = %0d",
 obj.size1, obj.size2, obj.size3);
end
endprogram

```

## \$clog2() System Function Call

VCS supports usage of `$clog2()` system function call or integer math function in the constraint scope as per IEEE SystemVerilog LRM Std 1800™-2012 Section 20.8.1 “Integer math functions”.

This significantly improves the verification productivity. You can use the `$clog2()` system function call directly in the constraint code to get the minimal number of bits required to store the value and to generate random stimulus based on the number of bits accordingly.

## Usage Example

Consider the following example for the usage of `$clog2()`:

```

program tb;
 class cls ;
 rand bit [15:0] a, b, c;

 constraint cons {
 c == a >> $clog2(b);
 b inside {3, 5, 14, 28, 46};
 a == 16'habcd;
 }
 endclass
 initial begin
 cls obj = new;
 repeat(3) begin
 obj.randomize;
 $display("a = 'h%4h, b = 'd%-4d, $clog2(b) = 'd%0d, c =
 'h%4h",
 obj.a, obj.b, $clog2(obj.b), obj.c);
 end
 end

```

```
 end
endprogram
```

To run the example, use the following command:

```
% vcs -sverilog clog2.v
% ./simv
```

It generates the following output:

```
a = 'habcd, b = 'd14 , $clog2(b) = 'd4, c = 'h0abc
a = 'habcd, b = 'd3 , $clog2(b) = 'd2, c = 'h2af3
a = 'habcd, b = 'd28 , $clog2(b) = 'd5, c = 'h055e
```

## Supporting Foreach Loop Iteration over Array Select

VCS supports foreach loop iteration over array select, such as `foreach (arr [0] [i] )`.

### *Example 180 Example for foreach loop iteration over array select*

```
program tb;
 class cls;
 rand bit [7:0] arr[1:0][7:0];

 constraint cons_array {
 // foreach iteration over the array select 'arr[0]'
 foreach (arr [0] [i]) {
 if (i <= 3) {
 arr[0][i] inside {10, 20, 30, 40};
 } else {
 arr[0][i] inside {50, 06, 70, 80};
 }
 }
 }
 endclass

 initial begin
 cls obj = new;
 obj.randomize;
 end
endprogram
```

## Support for Enumerated Type Methods in a Constraint Expression

VCS supports enumerated type methods in a constraint expression. The following enumerated type methods are supported:

- `first()`
- `last()`
- `next()`
- `prev()`
- `num()`

### Usage Example

Consider the following test case (`test.v`):

*Example 181 Usage of Enumerated Type Methods in a Constraint Expression*

```
program test;
 typedef enum {V, I, B, G, Y, O, R} colors_e;

 class C1;
 colors_e c0, c1 = I;
 rand colors_e rc;
 constraint c_in_range { rc == c1.next(); }
 endclass

 initial begin
 C1 obj1 = new();
 obj1.randomize();
 end
endprogram
```

Compile the test case as follows:

```
% vcs -sverilog test.v
```

### Limitation

The feature has the following limitation:

- Enumerated type methods can be used only on non-random `enum` variable inside constraints.

## Improved Support for Function Evaluation in Constraints

Evaluation of functions in constraints involves two-step process. In the first step the return value of the function is determined by solving the function arguments including its dependencies and evaluating the function.

In the second step, return value of the function is treated as a state value to satisfy rest of the constraint expressions involving this function call. This is in compliance with the *IEEE SystemVerilog Std 1800-2012*.

The current static approach may result in constraint inconsistency failures (CIFs) for scenarios where there are complex constraints on the arguments of the function and the function output.

In order to improve the quality of results, VCS has implemented the extended behavior where in case of conflict failures, the solver performs backtracking search to find function argument values that are consistent with the rest of the constraint problem. Thus, the solver finds a solution if it exists and avoids the constraint inconsistency failures.

To support this behavior, you can use the `+ntb_func_eval_in_solver=1` runtime option. The default value is `0` (existing flow). Following is the use model:

```
% simv +ntb_func_eval_in_solver=1 <other_options>
```

For example,

```
program tb;
 class cls;
 rand bit [7:0] r1, r2, r3;

 function bit [7:0] getSum(bit [7:0] r1, r2);
 return (r1 + r2);
 endfunction

 constraint cons1 {
 r3 == getSum(r1, r2);
 r3 inside {1, 3, 5, 7};
 }
 endclass

 initial begin
 cls obj = new;
 obj.randomize;
 end
endprogram
```

In the above example, the random variables `r1` and `r2` are evaluated in the first phase without having any dependency on `r3`.

The solver finds a solution, if the function return value `getSum(.,.)` is inside the given range {1, 3, 5, 7}. Otherwise, the solver generates a conflict.

**Note:**

The test may pass or fail based on the allocated random values `r1` and `r2`.

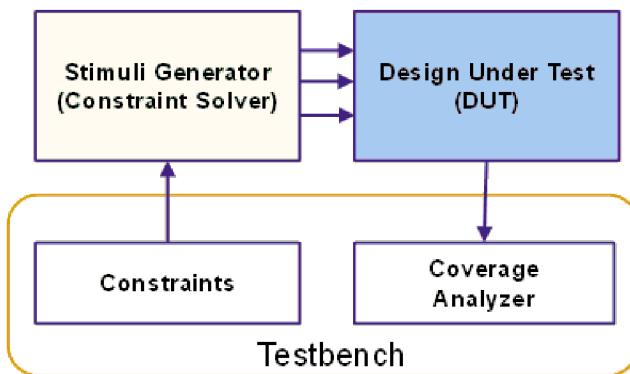
If the runtime switch is enabled, the solver searches different random values for the function call arguments to enter the feasible solution space and avoid conflicts.

# 19

## VCS Intelligent Coverage Optimization (ICO)

Constraint Solver (CS) generates input stimuli for a Design Under Test (DUT) at each stage during simulation by solving a set of user-defined testbench (TB) constraints at the current state, as shown in the following figure:

*Figure 169 Constraint Random Stimuli Quality*

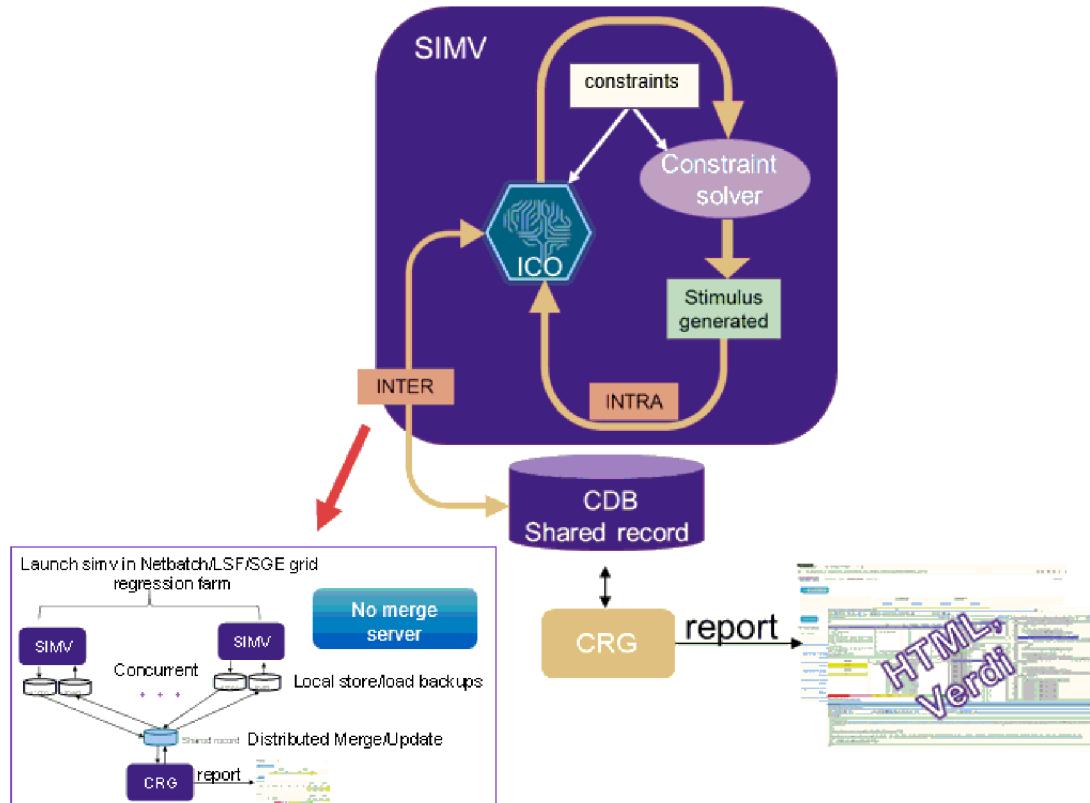


The quality of distribution of the generated stimuli is important as it can affect the number of verification cycles needed to meet coverage goals. The distribution of the stimuli is skewed by some preferred ordering of variables, which are induced by the constraints such as `solve before`, `dist`, relational and solver's affinity to certain variable ordering due to tight time budget. By default, there is no active feedback mechanism to rectify the skewed distribution.

Intelligent Coverage Optimization (ICO) using Adaptive Input Generator addresses the limitation of achieving the user intended stimuli distribution by using Reinforcement

Learning (RL) to maximize diversity, and thereby achieving improved stimuli quality.  
Consider the following figure:

*Figure 170 Stimuli Generation in ICO*



ICO computes the compensated distribution on-the-fly and biases the solver with the compensated distribution to nullify the skewness in the distribution and to maximize the diversity in the stimuli distribution, in both intra- and inter-simulation runs. This helps to reduce the repeated generation of the same stimuli in the regression, and thereby helps in improved and faster coverage closure. It also helps to expose volume/rare bugs and hard-to-hit scenarios.

In addition, ICO provides testbench (TB) stimuli visibility, diagnostics, and root cause analysis capabilities to help you debug TB for potential over-constraints and under-constraints causing low coverage, skewed distribution, and TB failures.

ICO can be used in all the stages (early, mid, and late stages) of a project. It can also be used as early as when the coverage model is not defined. It fits seamlessly in the VCS regression environment. There is no need to change the functional coverage model. It does not require an additional merge server.

For more details about VCS ICO, refer to the *VCS Intelligent Coverage Optimization (ICO) User Guide*.

# 20

## Extensions for SystemVerilog Coverage

---

The extensions for SystemVerilog coverage include the following:

- [Support for Reference Arguments in `get\_coverage\(\)` and `get\_inst\_coverage\(\)`](#)
- 

### Support for Reference Arguments in `get_coverage()` and `get_inst_coverage()`

The SystemVerilog LRM provides several predefined methods for every covergroup, coverpoint, or cross. For details, see Section 19.8, “Predefined coverage methods” in the *SystemVerilog LRM IEEE Std. 1800-2012*. Two of these predefined methods, `get_coverage()` and `get_inst_coverage()`, support optional arguments.

You can use the `get_coverage()` and `get_inst_coverage()` predefined methods to query on coverage during the simulation run, so that you can respond to the coverage statistics dynamically.

The `get_coverage()` and `get_inst_coverage()` methods both accept, as optional arguments, a pair of integer values passed by reference.

---

#### **`get_coverage()` method**

The numerator and denominator assigned by the `get_coverage()` method depend on the scope.

In the covergroup scope, `get_coverage()` assigns the weighted sum of the coverage of merged coverpoints and crosses to its first argument.

In the coverpoint or cross scope, the first argument to `get_coverage()` is the number of covered bins in the merged coverpoint or cross, and the second argument is the total number of bins.

In all cases, weighted sums are rounded to the nearest integer and the second argument is set to the sum of weights.

---

## get\_inst\_coverage() method

When the optional arguments are entered with the method in the coverpoint scope or cross scope, the `get_inst_coverage()` method assigns the value of the covered bins to the first argument, and assigns the number of bins for the given coverage item to the second argument. These two values correspond to the numerator and the denominator used for calculating the coverage score (before scaling by 100).

In covergroup scope, the `get_inst_coverage()` method assigns the weighted sum, rounded to the nearest integer, of coverpoint and cross coverage to the first argument and assigns the sum of the weights of the coverpoint or cross items to the second argument.

# 21

## OpenVera-SystemVerilog Testbench Interoperability

---

The primary purpose of OpenVera-SystemVerilog interoperability in VCS Native Testbench is to enable you to reuse OpenVera classes in new SystemVerilog code without rewriting OpenVera code into SystemVerilog.

This chapter describes the following topics:

- [Scope of Interoperability](#)
- [Importing OpenVera Types Into SystemVerilog](#)

Using the SystemVerilog package import syntax to import OpenVera data types and constructs into SystemVerilog.

- [Data Type Mapping](#)

The automatic mapping of data types between the two languages as well as the limitations of this mapping (some data types cannot be directly mapped).

- [Connecting to the Design](#)

Mapping of SystemVerilog modports to OpenVera where they can be used as OpenVera virtual ports.

- [Notes to Remember](#)
- [Usage Model](#)
- [Limitations](#)

## Scope of Interoperability

The scope of OpenVera-SystemVerilog interoperability in VCS Native Testbench is as follows:

- Classes defined in OpenVera can be used directly or extended in SystemVerilog testbenches.
- Program blocks must be coded in SystemVerilog. The SystemVerilog interface can include constructs, such as modports and clocking blocks, to communicate with a design.
- OpenVera code must not contain program blocks, bind statements, or predefined methods. It can contain classes, enums, ports, interfaces, tasks, and functions.
- OpenVera code can use virtual ports for sampling, driving, or waiting on design signals that are connected to the SystemVerilog testbench.

## Importing OpenVera Types Into SystemVerilog

OpenVera has two user-defined types: enums and classes. These types can be imported into SystemVerilog by using the SystemVerilog package `import` syntax:

```
import OpenVera::openvera_class_name;
import OpenVera::openvera_enum_name;
```

It allows you to use `openvera_class_name` in the SystemVerilog code in the same way as a SystemVerilog class. This includes the ability to perform the following:

- Create objects of the `openvera_class_name` type.
- Access or use properties and types defined in `openvera_class_name` or its base classes.
- Invoke methods (virtual and non-virtual) defined in `openvera_class_name` or its base classes.
- Extend `openvera_class_name` to SV classes.

However, this does not import the names of base classes of `openvera_class_name` into SystemVerilog (that requires an explicit import). For example:

```
// OpenVera
 class Base {
 .
 .
 .
 task foo(arguments) {
```

```

 .
 .
 }
 virtual task (arguments) {
 .
 .
 .
 }
class Derived extends Base {
 virtual task vfoo(arguments) {
 .
 .
 .
 }
}

// SystemVerilog
import OpenVera::Derived;
Derived d = new; // OK
initial begin
 d.foo(); // OK (Base::foo automatically
// imported)
 d.vfoo(); // OK
end
Base b = new; // not OK (do not know that Base is a
//class name)

```

The previous example would be valid if you add the following line before the first usage of the `Base` class name.

```
import OpenVera::Base;
```

Continuing with the previous example, SystemVerilog code can extend an OpenVera class as shown below:

```

// SystemVerilog
import OpenVera::Base;
class SVDerived extends Base;
 virtual task vmt()
 begin
 .
 .
 .
 end
endtask
endclass

```

**Note:**

- If a derived class redefines a base class method, the arguments of the derived class method must exactly match the arguments of the base class method.
- Explicit import of each data type from OpenVera can be avoided by a single `import OpenVera::*.`

```
// OpenVera
 class Base {
 integer i;
 .
 .
 .
 }
 class wrappedBase {
 public Base myBase;
 }
// SystemVerilog
 import OpenVera::wrappedBase;
 class extendedWrappedBase extends wrappedBase;
 .
 .
 .
endclass
```

In this example, `myBase.i` can be used to refer to this member of `Base` from the SV side. However, if SV also needs to use objects of the `Base` type, then you must include the following:

```
import OpenVera::Base;
```

## Data Type Mapping

This section describes how various data types in SystemVerilog are mapped to OpenVera and vice versa:

- *Direct mapping:* Many data types have a direct mapping in the other language and no conversion of data representation is required. In such cases, the OpenVera type is equivalent to the SystemVerilog type.
- *Implicit conversion:* In other cases, VCS performs implicit type conversion. The rules of inter-language implicit type conversion follows the implicit type conversion rules specified in the SystemVerilog LRM. To apply SystemVerilog rules to OpenVera, the OpenVera type must be first mapped to its equivalent SystemVerilog type. For example, there is no direct mapping between OpenVera `reg` and SystemVerilog `bit`. However, `reg` in OpenVera can be directly mapped to `logic` in SystemVerilog. The same implicit conversion rules between SystemVerilog `logic` and SystemVerilog `bit` can be applied to OpenVera `reg` and SystemVerilog `bit`.

- *Explicit translation:* In the case of mailboxes and semaphores, the translation must be explicitly performed by users. This is because in OpenVera, mailboxes and semaphores are represented by `integer ids` and VCS cannot reliably determine if an `integer` value represents a mailbox *id*.

This section discusses the following topics:

- [Mailboxes and Semaphores](#)
- [Events](#)
- [Strings](#)
- [Enumerated Types](#)
- [Integers and Bit-Vectors](#)
- [Arrays](#)
- [Structs and Unions](#)

## Mailboxes and Semaphores

Mailboxes and semaphores are referenced using object handles in SystemVerilog whereas in OpenVera, they are referenced using integral *ids*. VCS supports the mapping of mailboxes between these two languages.

For example, consider a mailbox created in SystemVerilog. To use it in OpenVera, you need to get *id* for the mailbox. The `get_id()` function, available as a VCS extension to SystemVerilog, returns this value:

```
function int mailbox::get_id();
```

The `get_id()` function is used as follows:

```
// SystemVerilog
 mailbox mbox = new;
 int id;

 .
 .
 .
 id = mbox.get_id();

 .
 .
 .
 foo.vera_method(id);

// OpenVera
class Foo {
 .
 .
 .
 task vera_method(integer id) {
 .
 }
}
```

```

 void = mailbox_put(data_type mailbox_id,
 data_type variable);
 }
}

```

Once OpenVera gets an *id* for a mailbox/semaphore, it can save it into any *integer* type variable. Note that if *get\_id* is invoked for a mailbox, the mailbox can no longer be garbage collected because VCS has no way of knowing when the mailbox ceases to be in use.

Typed mailboxes (currently not supported), when they are supported in SystemVerilog can be passed to OpenVera code using the same method as untyped mailboxes above. However, if the OpenVera code attempts to put an object of incompatible type into a typed mailbox, a simulation error occurs.

Bounded mailboxes (currently not supported), when they are supported in SystemVerilog can be passed to OpenVera code using the same method as above. OpenVera code trying to do *mailbox\_put* into a full mailbox results in a simulation error.

To use an OpenVera mailbox in SystemVerilog, you need to get a handle to the mailbox object using a system function call. The system function, *\$get\_mailbox*, returns this handle:

```
function mailbox $get_mailbox(int id);
```

The function is used as follows:

```
// SystemVerilog
.
.
.
mailbox mbox;
int id = foo.vera_method(); // vera_method returns an
 // OpenVera mailbox id
mbox = $get_mailbox(id);
```

Analogous extensions are available for semaphores:

```
function int semaphore::get_id();
function semaphore $get_semaphore(int id);
```

## Events

The OpenVera event data type is equivalent to the SystemVerilog event data type. Events from either language can be passed (as method arguments or return values) to the other language without any conversion. The operations performed on events in a given language are determined by the language syntax.

An event variable can be used in OpenVera in `sync` and `trigger`. An event variable `event1` can be used in SystemVerilog as follows:

```
event1.triggered //event1 triggered state property
->event1 //trigger event1
@(event1) //wait for event1
```

---

## Strings

OpenVera and SystemVerilog strings are equivalent. Strings from either language can be passed (as method arguments or return values) to the other language without any conversion. In OpenVera, `null` is the default value for `string`. In SystemVerilog, the default value is the empty string (""). It is illegal to assign `null` to `string` in SystemVerilog. Currently, Native Testbench-OpenVera (NTB-OV) treats "" and `null` as distinct constants (equality fails).

---

## Enumerated Types

SystemVerilog enumerated types have arbitrary base types and are not generally compatible with OpenVera enumerated types. A SystemVerilog enumerated type is implicitly converted to the base type of the enum (an integral type) and then, the bit-vector conversion rules (section 2.5) are applied to convert to an OpenVera type. This is illustrated in the following example:

```
// SystemVerilog
 typedef reg [7:0] formal_t; // SV type equivalent to
 // 'reg [7:0]' in OV
 typedef enum reg [7:0] { red = 8'hff, blue = 8'hfe,
 green = 8'hfd } color;
 // Note: the base type of color is 'reg [7:0]'
 typedef enum bit [1:0] { high = 2'b11, med = 2'b01,
 low = 2'b00 } level;
 color c;
 level d = high;
 Foo foo;
 ...
 foo.vera_method(c); // OK: formal_t'(c) is passed to
 // vera_method.
 foo.vera_method(d); // OK: formal_t'(d) is passed to
 // vera_method.
 // If d == high, then 8'b00000011 is
 // passed to vera_method.

// OpenVera
class Foo {
 ...
 task vera_method(reg [7:0] r) {
 ...
```

```
 }
 }
```

The above data type conversion does not involve a conversion in data representation. An enum can be passed by reference to the OpenVera code but the formal argument of the OpenVera method must exactly match the enum base type (for example, 2-to-4 value conversion, sign conversion, padding or truncation are not allowed for arguments passed by reference; they are OK for arguments passed by value).

Enumerated types with 2-value base types are implicitly converted to the appropriate 4-state type (of the same bit length). For the conversion of bit vector types, see the discussion in Section 2.5.

OpenVera enum types can be imported to SystemVerilog using the following syntax:

```
import OpenVera::openvera_enum_name;
```

It is used as follows:

```
// OpenVera
 enum OpCode { Add, Sub, Mul };

// System Verilog
 import OpenVera::OpCode;
 OpCode x = OpenVera::Add;

// or the enum label can be imported and then used
// without OpenVera::

 import OpenVera::Add;
 OpCode y = Add;
```

SystemVerilog enum methods such as `next`, `prev`, and `name` can be used on imported OpenVera enums.

Enums contained within OpenVera classes are illustrated in the following example:

```
class OVclass{
 enum Opcode {Add, Sub, Mul};
}

import OpenVera::OVclass;
OVclass::Opcode SVvar;
SVvar=OVclass::Add;
```

## Integers and Bit-Vectors

The mapping between SystemVerilog and OpenVera integral types are shown in the following table:

| <b>SystemVerilog</b> | <b>OpenVera</b>    | <b>2/4 or 4/2 value conversion?</b> | <b>Change in sign?</b> |
|----------------------|--------------------|-------------------------------------|------------------------|
| integer              | integer            | N (equivalent types)                | N (Both signed)        |
| byte                 | reg [7:0]          | Y                                   | Y                      |
| shortint             | reg [15:0]         | Y                                   | Y                      |
| int                  | integer            | Y                                   | N (Both signed)        |
| longint              | reg [63:0]         | Y                                   | Y                      |
| logic [m:n]          | reg [abs(m-n)+1:0] | N (equivalent types)                | N (Both unsigned)      |
| bit [m:n]            | reg [abs(m-n)+1:0] | Y                                   | N (Both unsigned)      |
| time                 | reg [63:0]         | Y                                   | N (Both unsigned)      |

### Note:

If a value or sign conversion is needed between the actual and formal arguments of a task or function, then the argument cannot be passed by reference.

## Arrays

Arrays can be passed as arguments to tasks and functions from SystemVerilog to OpenVera and vice versa. The formal and actual array arguments must have equivalent element types, the same number of dimensions with corresponding dimensions of the same length. These rules follow the SystemVerilog LRM.

- A SystemVerilog fixed array dimension of the form `[m:n]` is directly mapped to `[abs(m-n)+1]` in OpenVera.
- An OpenVera fixed array dimension of the form `[m]` is directly mapped to `[m]` in SystemVerilog.

Rules for equivalency of other (non-fixed) types of arrays are as follows:

- A dynamic array (or Smart queue) in OpenVera is directly mapped to a SystemVerilog dynamic array if their element types are equivalent (can be directly mapped).
- An OpenVera associative array with unspecified key type (for example, `integer a[]`) is equivalent to a SystemVerilog associative array with key type `reg [63:0]` provided the element types are equivalent.
- An OpenVera associative array with the `string` key type is equivalent to a SystemVerilog associative array with `string` key type provided the element types are equivalent.

Other types of SystemVerilog associative arrays have no equivalent in OpenVera and hence, they cannot be passed across the language boundary.

Some examples of compatibility are described in the following table:

| OpenVera                     | SystemVerilog                      | Compatibility |
|------------------------------|------------------------------------|---------------|
| <code>integer a[10]</code>   | <code>integer b[11:2]</code>       | Yes           |
| <code>integer a[10]</code>   | <code>int b[11:2]</code>           | No            |
| <code>reg [11:0] a[5]</code> | <code>logic [3:0][2:0] b[5]</code> | Yes           |

A 2-valued array type in SystemVerilog cannot be directly mapped to a 4-valued array in OpenVera. However, a cast may be performed as follows:

```
// OpenVera
class Foo {
 .
 .
 .
 task vera_method(integer array[5]) {
 .
 .
 .
 .
 .
 }
// SystemVerilog
 int array[5];
 typedef integer array_t[5];
 import OpenVera::Foo;
 Foo f;
 .
 .
 .
```

```
f.vera_method(array); // Error: type mismatch
f.vera_method(array_t'(array)); // OK
.
.
```

---

## Structs and Unions

Unpacked structs/unions cannot be passed as arguments to OpenVera methods. Packed structs/unions can be passed as arguments to OpenVera: they are implicitly converted to bit vectors of the same width.

packed struct {...} s in SystemVerilog is mapped to reg [m:0] r in OpenVera,  
where m == \$bits(s).

Analogous mapping applies to unions.

---

## Connecting to the Design

This section consists of the following subsections:

- [Mapping Modports to Virtual Ports](#)
  - [Semantic Issues With Samples, Drives, and Expects](#)
- 

## Mapping Modports to Virtual Ports

This section relies on the following extensions to SystemVerilog supported in VCS:

- [Virtual Modports](#)
- [Importing Clocking Block Members Into a Modport](#)

## Virtual Modports

VCS supports a *reference* to a modport in an interface to be declared using the following syntax:

```
virtual interface_name.modport_name virtual_modport_name;
```

For example:

```
interface IFC;
 wire a, b;
 modport mp (input a, output b);
endinterface

IFC i();
virtual IFC.mp vmp;
```

```
.
.
.
vmp = i.mp;
```

## Importing Clocking Block Members Into a Modport

VCS allows a reference to a clocking block member to be made by omitting the clocking block name.

For example, in SystemVerilog a clocking block is used in a modport as follows:

```
interface IFC(input clk);
 wire a, b;
 clocking cb @ (posedge clk);
 input a;
 input b;
 endclocking
 modport mp (clocking cb);
endinterface

program mpg(IFC ifc);
.
.
.
.
virtual IFC.mp vmp;
.
.
.
.
vmp = i.mp;
@(vmp.cb.a); // here you need to specify cb explicitly
.
endprogram
module top();
.
.
IFC ifc(clk); // use this to connect to DUT and TB
mpg mpg(ifc);
dut dut(...);
.
.
endmodule
```

VCS supports the following extensions that allow the clocking block name to be omitted from `vmp.cb.a`.

```
// Example-1
interface IFC(input clk);
 wire a, b;
 clocking cb @ (posedge clk);
 input a;
 input b;
```

```

 endclocking
 modport mp (import cb.a, import cb.b);
 endinterface

program mpg(IFC ifc);
.
.
.
virtual IFC.mp vmp;
.
.
.
vmp = i.mp;
@(vmp.a); // cb can be omitted; 'cb.a' is
 // imported into the modport

endprogram
module top();
.
.
.
IFC ifc(clk); // use this to connect to DUT and TB
mpg mpg(ifc);
dut dut(...);

.
.

endmodule

// Example-2
interface IFC(input clk);
 wire a, b;
 bit clk;
 clocking cb @(posedge clk);
 input a;
 input b;
 endclocking
 modport mp (import cb.*); // All members of cb
 // are imported.
 // Equivalent to the
 // modport in
 // Example-1.
endinterface

program mpg(IFC ifc);
.
.
.
IFC i(clk);
.
.
.
virtual IFC.mp vmp;
.
.
.

```

```

 vmp = i.mp;
 @(vmp.a); // cb can be omitted;
 //'cb.a' is imported into the modport
 endprogram

module top();
 .
 .
 IFC ifc(clk); // use this to connect to DUT and TB
 mpg mpg(ifc);
 dut dut(...);
 .
 .
endmodule

```

A SystemVerilog modport can be implicitly converted to an OpenVera virtual port provided the following conditions are satisfied:

- The modport and the virtual port have the same number of members.
- Each member of the modport converted to a virtual port must either be: (1) a clocking block, or (2) imported from a clocking block using the `import` syntax above.
- For different modports to be implicitly converted to the same virtual port, the corresponding members of the modports (in the order in which they appear in the modport declaration) be of bit lengths. If the members of a clocking block are imported into the modport using the `cb.*` syntax, where `cb` is a clocking block, then the order of those members in the modport is determined by their declaration order in `cb`.

### Example

```

// OpenVera
port P {
 clk;
 a;
 b;
}

class Foo {
 P p;
 task new(P p_) {
 p = p_;
 }

 task foo() {
 .
 .
 .
 @(p.$clk);
 .
 variable = p.$b;
 p.$a = variable;
 }
}

```

```

.
.
.

}

// SystemVerilog
interface IFC(input clk);
 wire a;
 wire b;

 clocking clk_cb @(clk);
 input #0 clk;
 endclocking

 clocking cb @ (posedge clk);
 output a;
 input b;
 endclocking

modport mp (import clk_cb.* , import cb.*); // modport
 // can aggregate signals from multiple clocking blocks.

endinterface: IFC

program mpg(IFC ifc);
 import OpenVera:::Foo;
 .

 virtual IFC.mp vmp = ifc.mp;
 Foo f = new(vmp); // clocking event of ifc.cb mapped to
 // $clk in port P
 // ifc.cb.a mapped to $a in port P
 // ifc.cb.b mapped to $b in port P

 f.foo();
 .

 .
 .

endprogram

module top();
 .

 IFC ifc(clk); // use this to connect to DUT and TB
 mpg mpg(ifc);
 dut dut(...);

 .
 .

endmodule

```

### Note:

In the above example, you can also directly pass the `vmp` modport from an interface instance:

```
Foo f = new(ifc.mp);
```

## Semantic Issues With Samples, Drives, and Expects

When OpenVera code wants to sample a DUT signal through a virtual port (or interface), if the current time is not at the relevant clock edge, the current thread is suspended until that clock edge occurs and then the value is sampled. Native Testbench-OpenVera (NTB-OV) implements this behavior by default. On the other hand, in SystemVerilog, sampling never blocks and the value that was sampled at the most recent edge of the clock is used. Analogous differences exist for drives and expects.

## Notes to Remember

This section discusses the following topics:

- [Blocking Functions in OpenVera](#)
- [Constraints and Randomization](#)
- [Functional Coverage](#)

## Blocking Functions in OpenVera

When a SystemVerilog function calls a virtual function that may resolve to a blocking OpenVera function at runtime, the compiler cannot determine with certainty whether the SystemVerilog function will block. VCS issues a warning at compile time and let the SystemVerilog function block at runtime.

Besides killing descendant processes in the same language domain, `terminate` invoked from OpenVera also kills descendant processes in SystemVerilog. Similarly, `disable fork` invoked from SystemVerilog also kills descendant processes in OpenVera. `wait_child` also waits for SystemVerilog descendant processes and `wait fork` also waits for OpenVera descendant processes.

## Constraints and Randomization

- SystemVerilog code can call `randomize()` on objects of an OpenVera class type.
- In SystemVerilog code, SystemVerilog syntax must be used to turn off/on constraint blocks or randomization of specific `rand` variables (even for OpenVera classes).

- Random stability is maintained across the language domain.

```
//OV
class OVclass{
 rand integer ri;
 constraint cnst{...}
}

//SV
OVclass obj=new();
SVclass Svobj=new();
Svobj.randomize();
obj.randomize() with
{obj.ri==Svobj.var;};
```

## Functional Coverage

There are some differences in functional coverage semantics between OpenVera and SystemVerilog. These differences are currently being eliminated by changing OpenVera semantics to conform to SystemVerilog. In the interoperability mode, `coverage_group` in OpenVera and `covergroup` in SystemVerilog have the same (SystemVerilog) semantics. Non-embedded coverage group can be imported from Vera to SystemVerilog using the package `import` syntax (similar to classes).

Coverage reports are unified and keywords such as `coverpoint`, `bins` are used from SystemVerilog instead of OpenVera keywords.

Here is an example of usage of coverage groups across the language boundary:

```
// OpenVera
class A
{
 B b;
 coverage_group cg {
 sample x(b.c);
 sample y(b.d);
 cross cc1(x, y);
 sample_event = @ (posedge CLOCK);
 }
 task new() {
 b = new;
 }
}
// SystemVerilog

import OpenVera::A;

initial begin
 A obj = new;
 obj.cg.option.at_least = 2;
 obj.cg.option.comment = "this should work";
```

```
@(posedge CLOCK);
$display("coverage=%f", obj.cg.get_coverage());
end
```

---

## Usage Model

Any `define from the OpenVera code is visible in SystemVerilog once they are explicitly included.

**Note:**

OpenVera #define must be rewritten as `define for ease of migration to SystemVerilog.

---

## For Two-Step Flow:

### Compilation

```
% vcs [compile_options] -sverilog -ntb_opts interop
[other_NTB_options] file4.sv file5.vr file2.v file1.v
```

### Simulation

```
% simv [simv_options]
```

---

## Three-Step Flow

### Analysis

```
% vlogan -sverilog -ntb_opts interop [other_NTB_options] \
[vlogan_options] file4.sv file5.vr file2.v file1.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

**Note:**

Specify the VHDL bottommost entity first, and then move up in order.

### Elaboration

```
% vcs [elab_options] top_cfg/entity/module
```

### Simulation

```
% simv [simv_options]
```

**Note:**

- - If RVM class libs are used in the OpenVera code, use -ntb\_opts rvm with the vlogan command.
- - Using -ntb\_opts interop -ntb\_opts rvm with vcs or vlogan, automatically translates rvm\_ macros in the OpenVera package to vmm\_ equivalents.

---

## Limitations

- Classes extended/defined in SystemVerilog cannot be instantiated by OpenVera. OpenVera verification IP needs to be compiled with the Native Testbench syntax and semantic restrictions. These restrictions are detailed in the *Native Testbench Coding Guide*, included in the VCS release.
- SystemVerilog contains several data types that are not supported in OpenVera including real, unpacked-structures, and unpacked-unions. OpenVera cannot access any variables or class data members of these types. A compiler error occurs if the OpenVera code attempts to access the undefined SystemVerilog data member. This does not prevent SystemVerilog passing an object to OpenVera, and then receiving it back again with the unsupported data items unchanged.
- When using VMM RVM Interoperability, you should only register VMM or RVM scenarios with a generator in the same language. You can instantiate an OpenVera scenario in a SystemVerilog scenario, but only a SystemVerilog scenario can be registered with a SystemVerilog generator. You cannot register OpenVera multi-stream scenarios on a SystemVerilog Multi-Stream Scenario Generator (MSSG).

# 22

## Using SystemVerilog Assertions

---

Using SystemVerilog Assertions (SVA) you can specify how you expect a design to behave and have VCS display messages when the design does not behave as specified.

```
assert property (@(posedge clk) req |> ##2 ack)
 else $display ("ACK failed to follow the request);
```

The above example displays "ACK failed to follow the request", if ACK is not high two clock cycles after req is high. This example is a very simple assertion. For more information on how to write assertions, refer to Chapter 17 of the *SystemVerilog Language Reference Manual*.

VCS allows you to:

- Control the SVAs
- Enable or Disable SVAs
- Control the simulation based on the assertion results

This chapter describes the following:

- [Using SVAs in the HDL Design](#)
- [Controlling SystemVerilog Assertions](#)
- [Viewing Results](#)
- [Enhanced Reporting for SystemVerilog Assertions in Functions](#)
- [Controlling Assertion Failure Messages](#)
- [Reporting Values of Variables in the Assertion Failure Messages](#)
- [Reporting Messages When \\$uniq\\_prior\\_checkon/\\$uniq\\_prior\\_checkoff System Tasks are Called](#)
- [Assertion and Unique/Priority Re-Trigger Feature](#)
- [Enabling Lint Messages for Assertions](#)
- [Fail-Only Assertion Evaluation Mode](#)
- [Treating x as true on an Assertion Precondition](#)

- Using SystemVerilog Constructs Inside vunits
- Calling \$error Task When Else Block is Not Present
- Disabling Default Assertion Success Dumping in -debug\_access Option
- Support for Success Count With -assert summary Option by Default
- Disabling Success Callbacks
- Enhancement to Assertion Diagnostics
- Support for Automatic Variables in Sampling Functions
- List of supported IEEE Std. 1800-2012 Compliant SVA Features
- Support for Strong Operators in Assertions
- Suppressing the Run-time out of Bound Access Messages
- Reporting Runtime Violations for Unique/Unique0/Priority For and Foreach Statements
- Using -assert errmsg Runtime Option
- Typed Formal Support for Sequence and Property Using -assert typed\_formal Option
- Disabling Sequence Debugging Using -assert no\_seqdebug Option
- SystemVerilog Assertions Limitations

---

## Using SVAs in the HDL Design

You can instantiate SVAs in your HDL design in the following ways:

- Using VCS Checker Library
- Binding SVA to a Design
- Binding SVA to a Design in VCS
- Inlining SVAs in the Verilog Design
- Inlining SVA in the VHDL design
- Number of SystemVerilog Assertions Supported in a Module

## Using VCS Checker Library

VCS provides you with SVA checkers, which can be directly instantiated in your Verilog/VHDL source files. You can find these SVA checkers files in `$VCS_HOME/packages/sva_cg` directory.

This section describes the use model to analyze, elaborate/compile and simulate the design with SVA checkers. For more information on SVA checker libraries and list of available checkers, see the *SystemVerilog Assertions Checker Library Reference Manual*.

### Instantiating SVA Checkers in Verilog

You can instantiate SVA checkers in your Verilog source just like instantiating any other Verilog module. For example, to instantiate the checker `assert_always`, specify the following:

```
module my_verilog();
 ...
 assert_always always_inst (.clk(clk), .reset(rst),
 .test_expr(test_expr));
 ...
endmodule
```

The use model to simulate the design with SVA checkers is as follows:

#### For two-step flow:

##### Compilation

```
% vcs [vcs_options] -sverilog +define+ASSERT_ON \
+includer+$VCS_HOME/packages/sva -y $VCS_HOME/packages/sva +libext+.v \
Verilog_source_files
```

##### Simulation

```
% simv [simv_options]
```

For more information on SVA checker libraries and a list of available checkers, see the *SystemVerilog Assertions Checker Library Reference Manual*.

#### For three-step flow:

##### Analysis

```
% vlogan -sverilog [vlogan_options] +define+ASSERT_ON \
+includer+$VCS_HOME/packages/sva -y $VCS_HOME/packages/sva +libext+.v \
Verilog_source_files
```

### Note:

It is necessary to use `+define+ASSERT_ON` to turn on the assertions in all checker instances.

### Elaboration

```
% vcs [vcs_options] top_cfg/entity/module
```

### Simulation

```
% simv [simv_options]
```

For more information on SVA checker libraries and a list of available checkers, see the *SystemVerilog Assertions Checker Library Reference Manual*.

## Instantiating SVA Checkers in VHDL

To instantiate SVA checkers in the VHDL source file, you need to do the following:

- Analyze the required SVA checker files using `vlogan`. For example, the command line to analyze the checker files in the default `WORK` library is as follows:

```
% vlogan $VCS_HOME/packages/sva/*.v \
+incdir+$VCS_HOME/packages/sva -y
$VCS_HOME/packages/sva +libext+.v \
+define+ASSERT_ON -sverilog
```

- Analyze the SVA component package file.

You can find SVA checkers in `$VCS_HOME/packages/sva` directory. In the same directory, you can also find the `sva_lib` VHDL package, containing the component definitions for all the checkers in the library. The name of this file is `component.sva_v.vhd`.

For example, suppose you analyze the package file in the default `WORK` library, then the `vhdlan` command line is as follows:

```
% vhdlan $VCS_HOME/packages/sva/component.sva_v.vhd
```

- To use the compiled checkers, you must include the `sva_lib` package in your VHDL file. For example, the following lines include the `sva_lib` package analyzed into the default `WORK` library:

```
library WORK;
use WORK.sva_lib.all;
```

For more information on SVA checker libraries and list of available checkers, see the *SystemVerilog Assertions Checker Library Reference Manual*.

You can now instantiate SVA checkers in your VHDL file, like any other VHDL entity. For example, to instantiate the checker `assert_always`, perform the following:

```
library IEEE;
use IEEE.STD_LOGIC_1164.all;
library WORK;
use WORK.sva_lib.all;

entity my_ent(
 ...
);
end my_ent;

architecture my_arch of my_ent is
...
begin
 ...
 checker_inst : assert_always port map(.clk(clk),
 .reset(rst), a(1));
 ...
end my_arch;
```

The use model to simulate the design with SVA checkers is as follows:

### Analysis

Always analyze Verilog before VHDL.

```
% vlogan [vlogan_options] Verilog_source_files
% vhdlan [vhdlan_options] VHDL_source_files
```

### Elaboration

```
% vcs [vcs_options] top_cfg/entity/module
```

### Simulation

```
% simv [simv_options]
```

## Binding SVA to a Design

Using `bind` statements to bind SVAs to your Verilog design. This is another way to use SVAs. The advantage is that the `bind` statements allow you to bind SVAs to the Verilog designs without modifying or editing your design files.

The syntax for `bind` statement is as follows:

```
bind inst_name/module SVA_module #[SVA_parameters] SVA_inst_name
 [SVA_ports]
```

The `bind` statement for Verilog targets can be used anywhere within your Verilog source file. For example:

```
//Verilog file
module dev (...);
...
endmodule

bind dev dev_checker dcl (.clk(clk), .a(a), .b(b));
```

As shown in the above example, the `bind` statement is specified in the same Verilog file.

The use model to simulate the design is as follows:

## Compilation

```
% vcs -sverilog [compile_options] Verilog_files
```

## Simulation

```
% simv [run_options]
```

## Binding SVA to a Design in VCS

You can use `bind` statements to merge SVAs to your Verilog or VHDL design without modifying or editing the design files..

This is a sample of `bind` statement syntax:

```
bind inst_name/module/entity SVA_module
 #[SVA_parameters] SVA_inst_name [SVA_ports]
```

These examples show how to bind a SVA module with a Verilog or VHDL design unit:

- To Bind with a Verilog module:

```
bind dev dev_checker dcl (.clk(clk), .a(a), .b(b));
```

- To Bind with a VHDL entity

```
bind vh_ent assert_always I1 (.clk(clk), .reset(rst),
 .in(in1[1]));
```

- To Bind with a specific VHDL entity and architecture

```
bind \vh_ent(rtl) assert_always I1 (.clk(clk), .reset(rst),
 .in(in1[1]));
```

- To Bind a VHDL entity and an architecture pair from a specific library

```
bind \L1.vh_ent(rtl) assert_always I1 (.clk(clk),
 .reset(rst), .in(in1[1]));
```

- To Bind with a specific VHDL instance

```
bind tb.top.vh_inst assert always I1 (.clk(clk),
 .reset(rst), .in(in1[1]));
```

You can use `bind` statements in these ways:

1. Writing bind statements within your Verilog design file.
2. Encapsulating bind statements within Verilog `module-endmodule` syntax.

## bind Statements In Your Verilog Source Files

The `bind` statement for Verilog targets can be used anywhere within the Verilog source file. For example:

```
//Verilog file
module dev (...);
...
endmodule

bind dev dev_checker dcl (.clk(clk), .a(a), .b(b));
```

As shown in the above example, the bind statement can be specified in the same Verilog file.

This is a sample use model to simulate the design:

### Analysis

Always analyze Verilog before VHDL.

```
% vlogan -sverilog [vlogan_options] Verilog_files
```

### Elaboration

```
% vcs [elab_options] design_unit
```

### Simulation

```
% simv [run_options]
```

## Encapsulating Bind Statements

You can bind SVAs to both Verilog or VHDL design units by specifying bind statements within Verilog `module-endmodule`. You can also analyze all the bind statements using `vlogan`, and you can specify the required set of bind statements for the simulation during elaboration.

The use model is as follows:

1. Encapsulate your bind statements within the Verilog module-endmodule:

```
module bind_a;
 bind top.vh1.ia child C0(a[0],b[0],cin,sum[0],cout);
 bind top.vh1.ia child C1(a[1],b[1],cin,sum[1],cout);
endmodule

module bind_b;
 bind top.vh1.ib child C2(a[2],b[2],cin,sum[2],cout);
 bind top.vh1.ib child C3(a[3],b[3],cin,sum[3],cout);
endmodule
```

You can write any number of bind statements within a module and any number of such modules containing bind statements.

2. Analyze all Verilog files and bind files using vlogan, and VHDL files using vhdlan.

```
% vlogan [vlogan_options] Verilog_files
% vlogan -sverilog [vlogan_options] Bind_files
% vhdlan [vhdlan_options] VHDL_files
```

3. Specify the module containing the required bind statements during elaboration along with your design top module/entity/configuration:

```
% vcs -sva_bind_enable design_top bind_a [elab_options]
```

The above command line elaborates and links the design, and the specified bind module, thereby enabling only those SVA modules specified within the module bind\_a. The option `-sva_bind_enable` tells VCS to bind the bind statements in bind\_a with the design.

4. Simulate the design

```
% simv [runtime_options]
```

## Supported Data Types

These VHDL data types are supported in bind statements:

- Bit
- Boolean
- Integer and Integer subtype
- std\_logic and std\_ulogic
- std\_logic\_vector and std\_ulogic\_vector

This table lists the supported SystemVerilog datatypes with their matching VHDL datatypes.

*Table 46 SystemVerilog datatypes with their matching VHDL datatypes*

| SystemVerilog Data Types | Integer | Integer Subtype | Bit vector | std_logic vector | std_ulogic vector |
|--------------------------|---------|-----------------|------------|------------------|-------------------|
| Shortint                 | No      | No              | Yes        | Yes              | Yes               |
| Int                      | Yes     | Yes             | Yes        | Yes              | Yes               |
| Longint                  | No      | No              | Yes        | Yes              | Yes               |
| Bit array                | Yes     | Yes             | Yes        | Yes              | Yes               |
| Logic array              | Yes     | Yes             | Yes        | Yes              | Yes               |
| Integer                  | Yes     | Yes             | Yes        | Yes              | Yes               |

#### Note:

- Vectors on either side should match in length, vectors, and datatypes.
- Integer subtype of VHDL can only be connected to a 32-bit vector or integer type in SV.
- One dimensional VHDL array is considered as a packed array in SV.

## Extensions

These are the extensions that are specific to using SVAs in mixed HDL designs:

- You can refer to both Verilog and VHDL inputs, output, and inout ports in the bind statement.

#### Note:

VHDL 93 LRM does not allow reading of VHDL outputs. However, VCS relaxes this limitation, and allows you to read VHDL outports.

- You can refer to internal signals within the architecture in the bind statement.
- ENUMs are also supported.

## Salient Features

- You can write an expression for a port in the bind statement.
- Integer, real and string types of generics are supported.

- You can specify an absolute or a relative XMR or path to any VHDL record in the bind statements. However, the following limitations apply:
  - XMRs can be a primary expression, bit select or part select. The width of the target should match the port width of the SVA module.
  - Expressions cannot involve generics or ports.
  - XMRs should not have extended or escaped names.

## Limitations

These are the limitations of binding SVA to a design:

- XMRs cannot be used to override parameters.
- XMRs are not allowed through the SVA modules.
- SVA modules cannot have VHDL instances below it.

## Inlining SVAs in the Verilog Design

VCS allows you to write inlined SVAs for both VHDL and Verilog design. For Verilog designs, you can write SVAs as part of the code or within pragmas as shown in the following example:

### *Example 182 Writing Assertions as a part of the code*

```
module dut(...);

 ...
 sequence s1;
 @(posedge clk) sig1 ##[1:3] sig2;
 endsequence
 ...
endmodule
```

### *Example 183 Writing Assertions using SVA pragmas (//sv\_pragma)*

```
module dut(...);

 ...
 //sv_pragma sequence s1;
 //sv_pragma @(posedge clk) sig1 ##[1:3] sig2;
 //sv_pragma endsequence
 /*sv_pragma
 sequence s2;
```

```

 @ (posedge clk) sig3 ##[1:3] sig4;
endsequence
*/
.....
endmodule

```

As shown in Example 2, you can use SVA pragmas as `//sv pragma` at the beginning of all SVA lines, or you can use the following to mark a block of code as SVA code:

```

/* sv pragma
sequence s2;
 @ (posedge clk) sig3 ##[1:3] sig4;
endsequence
*/

```

## Use Model

The use model to analyze, elaborate/compile and simulate the designs having inlined assertions is as follows:

### Two-Step Flow

#### Compilation

```
% vcs -sv pragma [compile_options] Verilog_files
```

#### Simulation

```
% simv [run_options]
```

### Three-Step Flow

#### Analysis

```
% vlogan -sv pragma [vlogan_options] file1.v file2.v
```

#### Note:

If you have your assertions inlined using `//sv pragma`, use the analysis option `-sv pragma` as shown above.

#### Elaboration

```
% vcs -sv pragma [elab_options] design_unit
```

#### Simulation

```
% simv [run_options]
```

## Inlining SVA in the VHDL design

Inlining SVAs in VHDL design is possible only by using the SVA pragmas. *The location of the SVA implicitly specifies to which entity-architecture the SVA code is bound to.* You can embed the SVA code in the concurrent portion on your VHDL code using the `--sva_begin` and `--sva_end` pragmas. These pragmas should be written within an `architecture -end architecture` definition block as shown in the following example:

```
architecture RTL of cntrl is
begin
 ...
 --sva_begin
 -- property p1;
 -- @ (posedge clk) a && b #>1 !c ;
 -- endproperty : p1

 -- a_p1: assert property (p1) else $display ($time, " :
 Assertion a_p1 failed");
 --sva_end
end architecture RTL;
```

As soon as VCS encounters `--sva_begin`, it implicitly understands that the following lines until `--sva_end` are SVA constructs.

Within the inlined SVA code, you can:

- use VHDL signals, generics, and constants.
- write Verilog comments, compiler directives, and SVA pragmas.

However, you cannot use a VHDL variable within the inlined SVA code.

## Use Model

### Analysis

Always analyze Verilog before VHDL.

```
% vlogan -sverilog [vlogan_options] file1.v file2.v file3.v
% vhdlan -sva [vhdlan_options] file2.vhd file1.vhd
```

### Note:

- Use the `-sva` option, if you have SVA code inlined in your VHDL.
- For analysis, analyze the VHDL bottom-most entity first, then move up the order.

### Elaboration

```
% vcs [vcs_options] top_cfg/entity/module
```

## Simulation

```
% simv [simv_options]
```

You can also use the option `-sv_opts "vlog_opts_to_SVAs"` with `vhdlan` to specify Verilog options, such as `+define+macro -timescale=timeunit/precision` to the inlined SVA code as shown in the following example:

```
% vhdlan -sva -sv_opts "-timescale=1ns/1ns" myDut.vhd
```

The following example shows the usage of '`ifdef`' within the inlined SVA code:

```
architecture RTL of cntrl is
begin
 ...
 --sva_begin
 -- 'ifdef P1
 -- property p1;
 -- @ (posedge clk) a && b ##1 !c ;
 -- endproperty : p1
 -- 'else
 -- property p1;
 -- @ (posedge clk) a || b ##1 !c ;
 -- endproperty : p1
 -- 'endif
 -- a_p: assert property (p1)
 -- else $display ($time, " : Assertion a_p failed");
 --sva_end
end architecture RTL;
```

In this example, to select the first property `P1`, you need to specify `+define+P1` as an argument to `-sv_opts` option as follows:

```
% vhdlan -sva -sv_opts "+define+P1" myDut.vhd
```

## Number of SystemVerilog Assertions Supported in a Module

VCS supports up to 1,048,000 SystemVerilog assertions per module.

### Note:

Large number of assertions in a module can cause performance issues. If the performance degrades, it is recommended to sub-divide the module into multiple modules and distribute assertions over those modules.

---

## Controlling SystemVerilog Assertions

SVAs can be controlled or monitored using:

- [Compilation/Elaboration and Runtime Options](#)
  - [Concatenating Assertion Options](#)
  - [Assertion Monitoring System Tasks](#)
  - [Using Assertion Categories](#)
- 

### Compilation/Elaboration and Runtime Options

VCS provides various compilation/elaboration options to perform the following tasks:

- The following is the list of assertion options that are enabled when the `-assert enable_diag` compilation/elaboration option is used:
  - `-assert success`
  - `-assert summary`
  - `-assert maxcover=N`
  - `-assert maxsuccess=N`

The following is the list of assertion options that are enabled when `-assert enable_hier` compilation/elaboration option is used:

- `-assert hier`
- `-assert maxfail=N`
- `-assert finish_maxfail=N`

The following is the list of assertion options that do not require the `-assert enable_diag` or `-assert enable_hier` option:

- `-assert dumpoff`
- `-assert nocovdb`
- `-assert nopostproc`
- `-assert quiet`
- `-assert quiet1`
- `-assert no_fatal_action`

- -assert report
- -assert vacuous
- -assert global\_finish\_maxfail=N
- To disable all SVAs in the design, use the `-assert disable` compile-time option. To disable only the SVAs specified in a file, use the `-assert hier=<file_name>` compile-time option.
- To disable assertion coverage, use the `-assert disable_cover` compile-time option. By default, when you use the `-cm assert` option, VCS enables you to monitor your assertions for coverage, and write an assertion coverage database during simulation.
- To disable property checks (that is, `assert` and `assume` directives) and retain assertion coverage (that is, `cover` directives), use the `-assert disable_assert` option at compile time.
- Disable dumping of SVA information in the VPD file

You can use the `-assert dumpoff` option to disable the dumping of SVA information to the VPD file during simulation (for additional information, see [Options for SystemVerilog Assertions](#)).

VCS allows you to do the following tasks during runtime:

- Terminate simulation after certain number of assertion failures.

You can use either the `-assert finish_maxfail=N` or `-assert global_finish_maxfail=N` runtime option to terminate the simulation if the number of failures for any assertion reaches `N` or if the total number of failures from all SVAs reaches `N`, respectively.

- Show both passing and failing assertions

By default, VCS reports only failures. However, you can use the `-assert success` option to enable reporting of successful matches, and successes on `cover` statements in addition to failures.

- Limit the maximum number of successes reported

You can use the `-assert maxsuccess=N` option to limit the total number of reported successes to `N`.

- Disable the display of messages when assertions fail

You can use the `-assert quiet` option to disable the display of messages when assertions fail.

- Enable or disable assertions during runtime

You can use the `-assert hier=file_name` option to enable or disable the list of assertions in the specified file.

- Generate a report file

You can use the `-assert report=file_name` option to generate a report file with the specified name. For additional information, see [Options for SystemVerilog Assertions](#).

- Enable assertion control using wildcard characters.

You can use the `-assert enable_wildcard` option to enable the support of assertion control through wildcard characters. For example, `$assertoff(0, "Top.AL.*.A1")`.

- You can enter more than one keyword, using the plus + separator. For example:

```
% vcs -assert maxfail=10+maxsuccess=20+success
```

- To enable dumping of the SystemVerilog assertion success messages to the VPD file, use the `-assert dumpsuccess` option.

## Concatenating Assertion Options

VCS allows you to concatenate the compile-time and runtime assertion options on the compile command-line using the + separator. VCS identifies and appropriately passes these options to compile and runtime, wherever applicable. Concatenating all options simplifies the use model.

For example, you can concatenate compile-time option `-assert enable_diag` with the runtime option `-assert success`, as follows:

```
% vcs -assert success+enable_diag -R
```

When an option has the same name at both compile time and runtime, and are used on the `vcs` compile command line with the `-R` option, then it the option is applied only at compile time.

## Assertion Monitoring System Tasks

To monitor SystemVerilog assertions, use the following new system tasks:

```
$assert_monitor
$assert_monitor_off
$assert_monitor_on
```

### Note:

Enter these system tasks in an `initial` block. Do not enter these system tasks in an `always` block.

The `$assert_monitor` system task is equivalent to the standard `$monitor` system task. It continually monitors the specified assertions and displays what is happening with them (you can have it displayed only on the next clock of the assertion). The syntax is as follows:

```
$assert_monitor([0|1,] assertion_identifier...);
```

Where:

0

Specifies reporting on the assertion if it is active (VCS checks for its properties). For the remaining assertions, it specifies reporting, whenever they start.

1

Specifies reporting on the assertions only once, the next time they start.

If you specify neither 0 or 1, the default is 0.

`assertion_identifier...`

A comma separated list of assertions. If one of these assertions is not declared in the module definition containing this system task, specify it by its hierarchical name.

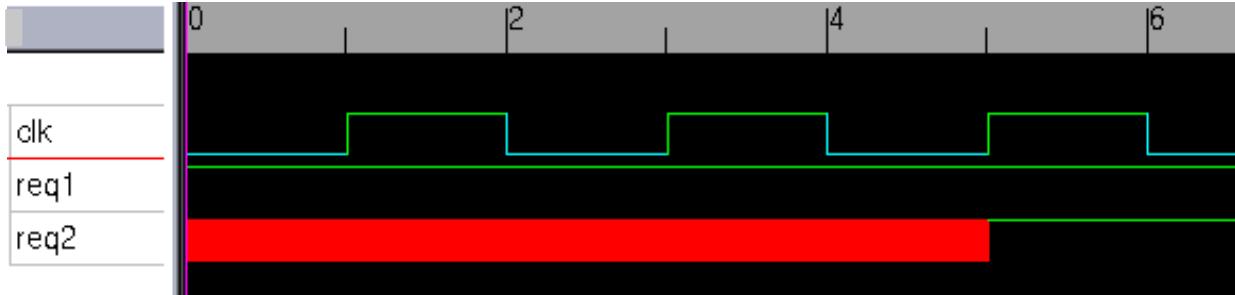
Consider the following assertion:

```
property p1;
 @ (posedge clk) (req1 ##[1:5] req2);
endproperty

a1: assert property(p1);
```

For property `p1` in assertion `a1`, a clock tick is a rising edge on signal `clk`. When there is a clock tick VCS checks to see if signal `req1` is true, and then to see if signal `req2` is true at any of the next five clock ticks.

In this example simulation, signal `clk` initializes to 0 and toggles every 1 ns, so the clock ticks at 1 ns, 3 ns, 5 ns and so on.



A typical display of this system task is as follows:

```
Assertion test.a1 ['design.v'27]:

5ns: tracing "test.a1" started at 5ns:

 attempt starting found: req1 looking for: req2 or any

5ns: tracing "test.a1" started at 3ns:

 trace: req1 ##1 any looking for: req2 or any

 failed: req1 ##1 req2

5ns: tracing "test.a1" started at 1ns:

 trace: req1 ##1 any[* 2] looking for: req2 or any

 failed: req1 ##1 any ##1 req2
```

Breaking this display into smaller chunks:

```
Assertion test.a1 ['design.v'27]:
```

The display is about the assertion with the hierarchical name `test.a1`. It is in the source file named `design.v` and declared on line 27.

```
5ns: tracing "test.a1" started at 5ns:

 attempt starting found: req1 looking for: req2 or any
```

At simulation time, 5 ns VCS is tracing `test.a1`. An attempt at the assertion started at 5 ns. At this time, VCS found `req1` to be true and is looking to see if `req2` is true one to five clock ticks after 5 ns. Signal `req2` doesn't have to be true on the next clock tick, so `req2` not being true is okay on the next clock tick; that's what looking for "or any" means, anything else than `req2` being true.

```
5ns: tracing "test.a1" started at 3ns:

 trace: req1 ##1 any looking for: req2 or any

 failed: req1 ##1 req2
```

The attempt at the assertion also started at 3 ns. At that time, VCS found `req1` to be true at 3 ns and it is looking for `req2` to be true some time later. The assertion "failed" in that `req2` was not true one clock tick later. This is not a true failure of the assertion at 3 ns, it can still succeed in two more clock ticks, but it didn't succeed at 5 ns.

```
5ns: tracing "test.a1" started at 1ns:

 trace: req1 ##1 any[* 2] looking for: req2 or any

 failed: req1 ##1 any ##1 req2
```

The attempt at the assertion also started at 1 ns. `[*` is the repeat operator. `##1 any[* 2 ]` means that after one clock tick, anything can happen, repeated twice. So the second line here says that `req1` was true at 1 ns, anything happened after a clock tick after 1 ns (3 ns) and again after another clock tick (5 ns) and VCS is now looking for `req2` to be true or anything else could happen. The third line here says the assertion "failed" two clock ticks (5 ns) after `req1` was found to be true at 1 ns.

The `$assert_monitor_off` and `$assert_monitor_on` system tasks turn off and on the display from the `$assert_monitor` system task, just like the `$monitoroff` and `$monitoron` system turn off and on the display from the `$monitor` system task.

---

## Using Assertion Categories

You can categorize assertions and then enable and disable them by category. There are two ways to categorize assertions:

- [Using System Tasks](#)
  - [Using Assertion System Tasks](#)
- [Using Attributes](#)
- [Starting and Stopping Assertions Using Assertion System Tasks](#)
  - [Stopping Assertions by Category or Severity](#)

After you categorize assertions you can use these categories to stop and restart assertions.

## Using System Tasks

VCS has a number of system tasks and functions for assertions. These system tasks do the following:

- Set a category for an assertion
- Return the category of an assertion

### Using Assertion System Tasks

You can use the following assertion system tasks to set the category and severity attributes of assertions:

```
$assert_set_severity("assertion_full_hier_name", severity)
```

Sets the severity level attributes of an assertion. The severity level is an unsigned integer from 0 to 255.

```
$assert_set_category("assertion_full_hier_name", category)
```

Sets the category level attributes of an assertion. The category level is an unsigned integer from 0 to 224 - 1.

You can use the following system tasks to retrieve the category and severity attributes of assertions:

```
$assert_get_severity("assertion_full_hier_name")
```

Returns the severity of action for an assertion failure.

```
$assert_get_category("assertion_full_hier_name")
```

Returns an unsigned integer for the category.

After specifying these system tasks and functions, you can start or stop the monitoring of assertions based upon their specified category or severity. For details on starting and stopping assertions, see [Starting and Stopping Assertions Using Assertion System Tasks](#).

## Using Attributes

You can prefix an attribute in front of an `assert` statement to specify the category of the assertion. The attribute must begin with the `category` name and specify an integer value, for example:

```
(* category=1 *) a1: assert property (p1);
(* category=2 *) a2: assert property (s1);
```

The value you specify can be an unsigned integer from 0 to 224 - 1, or a constant expression that evaluates to 0 to 224 - 1.

You can use a `parameter`, `localparam`, or `genvar` in these attributes. For example:

```
parameter p=1;
localparam l=2;
.

.

(* category=p+1 *) a1: assert property (p1);
(* category=l *) a2: assert property (s1);

genvar g;
generate
for (g=0; g<1; g=g+1)
begin:loop
(* category=g *) a3: assert property (s2);
end
endgenerate
```

### Note:

In a `generate` statement the category value cannot be an expression, the attribute in the following example is invalid:

```
genvar g;
generate
for (g=0; g<1; g=g+1)
begin:loop
(* category=g+1 *) a3: assert property (s2);
end
endgenerate
```

If you use a `parameter` for a category value, the parameter value can be overwritten in a module instantiation statement.

You can use these attributes to assign categories to both named and unnamed assertions.  
 For example:

```
(* category=p+1 *) a1: assert property (p1);
(* category=l *) assert property (s1);
```

The attribute is retained in a `tokens.v` file when you use the `-Xman=4` compile-time option and the keyword argument.

## Starting and Stopping Assertions Using Assertion System Tasks

There are assertions system tasks for starting and stopping assertions. These system tasks are as follows:

### Stopping Assertions by Category or Severity

You can stop assertions by category using the following option:

```
$assert_category_stop(categoryValue, [maskValue[,globalDirective]]);
```

The option stops all assertions associated with the specified category.

You can stop assertions for the specified severity level using the following option:

```
$assert_severity_stop(severityValue, [maskValue[,globalDirective]]);
```

The option stops all assertions associated with the specified severity level.

where,

*categoryValue*

Since there is a `maskValue` argument, it is now the result of an `and`ing operation between the assertion categories and the `maskValue` argument. If the result matches this value, these categories stop. Without the `maskValue` argument, this argument is the value you specify in `$assert_set_category` system tasks or `category` attributes.

*maskValue*

A value that is logically anded with the category of the assertion. If the result of this `and` operation matches the `categoryValue`, VCS stops monitoring the assertion.

*globalDirective*

The value can be either of the following values:

0

Enables an `$assert_category_start` system task that does not have a `globalDirective` argument, to restart the assertions stopped with this system task.

1

Prevents an `$assert_category_start` system task that does not have a `globalDirective` argument from restarting the assertions stopped with this system task.

### Starting Assertions by Category or Severity

You can start assertions by category using the following option:

```
$assert_category_start(categoryValue,
[maskValue[,globalDirective]]);
```

The option starts all assertions associated with the specified category.

You can start assertions for the specified severity level using the following option:

```
$assert_severity_start(severityValue, [maskValue[,globalDirective]]);
```

The option starts all assertions associated with the specified severity level. The severity level is an unsigned integer from 0 to 255.

where:

*categoryValue*

Since there is a `maskValue` argument, this argument is the result of an `and`ing operation between the assertion categories and the `maskValue` argument. If the result matches this value, these categories start. Without the `maskValue` argument, this argument is the value you specify in `$assert_set_category` system tasks or `category` attributes.

*maskValue*

A value that is logically anded with the category of the assertion. If the result of this `and` operation matches the `categoryValue`, VCS starts monitoring the assertion.

*globalDirective*

Can be either of the following values:

0

Enables an `$assert_category_stop` system task (that does not have a `globalDirective` argument) to stop the assertions started with this system task.

1

Prevents an `$assert_category_stop` system task that does not have a `globalDirective` argument from stopping the assertions started with this system task.

### Example Showing How to Use MaskValue

[Example 184](#) stops the odd numbered categories

**Example 184 MaskValue Numbering:**

```
$assert_set_category(top.d1.a1,1);
$assert_set_category(top.d1.a2,2);
$assert_set_category(top.d1.a3,3);
$assert_set_category(top.d1.a4,4);

.
.
.
.

$assert_category_stop(1,'h1);
```

The categories are masked with the `maskValue` argument and compared with the `categoryValue` argument as shown in the following table

|            | bits | categoryValue |          |
|------------|------|---------------|----------|
| category 1 | 001  |               |          |
| maskValue  | 1    |               |          |
| result     | 1 1  |               | match    |
| category 2 | 010  |               |          |
| maskValue  | 1    |               |          |
| result     | 0 1  |               | no match |
| category 3 | 011  |               |          |
| maskValue  | 1    |               |          |
| result     | 1 1  |               | match    |
| category 4 | 100  |               |          |
| maskValue  | 1    |               |          |
| result     | 0 1  |               | no match |
| .          | .    |               |          |

1. VCS logically `ands` the category value to the `maskValue` argument, which is 1.
2. The result of the `and` operation is true for categories 1 and 3 as per the calculation shown above. The result is false for categories 2 and 4.
3. VCS stops all the assertions which result in a true match with the `and` operation.

[Example 185](#) uses the `globalDirective` argument.

#### *Example 185 Mask Value with Global Directive*

```
$assert_set_category(top.d1.a1,1);
$assert_set_category(top.d1.a2,2);
$assert_set_category(top.d1.a3,3);
$assert_set_category(top.d1.a4,4);
.
.
$assert_category_stop(1,'h1,1);
$assert_category_start(0,'h1);
```

The assertions that are stopped or started with `globalDirective` value 1, cannot be restarted or stopped with a call to `$assert_category_start`, without using the `globalDirective` argument. The above code cannot restart assertions.

The assertions can only be restarted with a call to `$assert_category_start` with `globalDirective`, as follows:

```
$assert_category_start(1,'h1,1);
or
$assert_category_start(1,'h1,0);
```

## Viewing Results

By default, VCS reports only assertion of the failures. However, you can use the `-assert success` runtime option to report both pass and failures.

Assertion results can be viewed:

- Using a Report File
- Using Verdi

For more information on viewing assertions using Verdi, refer to the “*Assertion Debug*” chapter in the Verdi and Silioti Command Reference Guide.

## Using a Report File

Using the `-assert report=file_name` option, you can create an assertion report file. VCS writes all the SVA messages to the specified file.

Assertion generates messages with the following format:

| File and line with<br>the assertion                | Full hierarchical name<br>of the assertion       | Start time        | Status (succeeded at ...,<br>failed at ...,<br>not finished)   |
|----------------------------------------------------|--------------------------------------------------|-------------------|----------------------------------------------------------------|
|                                                    |                                                  |                   |                                                                |
| "design.v", 157: top.cnt_in.a2: started at 22100ns | Offending '(busData == mem[\$past(busAddr, 3)])' | failed at 22700ns |                                                                |
|                                                    |                                                  |                   |                                                                |
|                                                    |                                                  |                   | Expression that failed (only with failure of check assertions) |

## Enhanced Reporting for SystemVerilog Assertions in Functions

This section describes an efficient reporting convention for functions containing assertions in the following topics:

- [Introduction](#)
- [Use Model](#)
- [Name Conflict Resolution](#)
- [Checker and Generate Blocks](#)

## Introduction

In earlier releases, when assertions were present inside functions, assertion path names were reported based on the position of the function call in the source file. For example, consider the following code:

```
module top;
bit b, a1, a2, a3, a4, a5;
function bit myfunc(input bit k);
 $display("FUNC name: %m");
 AF: assert #0(k && !k);
 return !k;
endfunction
```

```

always_comb a1=myfunc(b);
always_comb begin: A
 begin: B
 a2=myfunc(b);
 begin a3=myfunc(!b); end
 end
end
always_comb begin
 a4=myfunc(b);
 a5=myfunc(!b);
end
endmodule

```

If you run this code, it generates the following output:

```

"top.v", 5: top.\top.v_18_myfunc.AF : started
"top.v", 5: top.\top.v_17_myfunc.AF : started
"top.v", 5: top.\top.v_13_myfunc.AF : started
"top.v", 5: top.\top.v_12_myfunc.AF : started
"top.v", 5: top.\top.v_9_myfunc.AF : started

```

But the problem with this type of naming convention is that when code changes, the output of the simulation also changes. To overcome this limitation, a new naming convention is implemented under the `-assert funchier` compile-time option. This new naming convention is implemented as follows:

- Function names are generated based on the named blocks under which the functions are called. Each function name is appended with an index (index=0, 1, 2, 3...), where index 0 is assigned to the first function call, index 1 is assigned to the second function call, and so on.
- For unnamed blocks, the function name is based on the closest named block.
- If there is no named scope around the function call, then a module scope is used as a named block with an empty name.
- Each assertion status reporting the message contains the file name and the line number of the function caller.

## Use Model

Use the `-assert funchier` option to enable the new function naming convention, as shown in the following command:

```
% vcs -sverilog -assert funchier+svaext top.v
% simv
```

If you run the above code using this command, it generates the following output:

```
"top.v", 5: top.myfunc_2.AF ("top.v", 18): started
"top.v", 5: top.myfunc_1.AF ("top.v", 17): started
```

```
"top.v", 5: top.\A.B.myfunc_1.AF ("top.v", 13): started ...
"top.v", 5: top.\A.B.myfunc_0.AF ("top.v", 12): started
"top.v", 5: top.myfunc_0.AF ("top.v", 9): started
```

## Name Conflict Resolution

When a function name generated with the new naming convention conflicts with an existing block or identifier name in that scope, then the suffix index is incremented until the conflict is resolved.

## Checker and Generate Blocks

When a function is present inside a checker, the generated name of that function contains the checker name appended to all named blocks and identifiers in that checker.

Similarly, when a function is present inside a `generate` block, the generated name of that function contains the generated block name appended to all named blocks and identifiers in that `generate` block.

## Controlling Assertion Failure Messages

This section describes the mechanism to control failure messages for SystemVerilog Assertions (SVA), OpenVera Assertions (OVA), Property Specification Language (PSL) assertions, and OVA case checks.

This section contains the following topics:

- [Introduction](#)
- [Options for Controlling Default Assertion Failure Messages](#)
- [Options to Control Termination of Simulation](#)
- [Option to Enable Compilation of OVA Case Pragmas](#)

## Introduction

Earlier releases did not provide the flexibility to control the display of default messages for assertion (SVA, OVA, or PSL) failures based on the presence of an action block (for SVA) or a user message (for OVA and PSL). Also, there was no control over whether these assertion failures contributed to the failure counts for `-assert [global_]finish_maxfail, or affected simulation if $ova_[severity|category]_action(<severity_or_category>, "finish") was specified.`

You can now use the options described in the following topics to enable additional controls on failure messages, and to terminate the simulation and compilation of OVA case pragmas.

## Options for Controlling Default Assertion Failure Messages

There are two ways to disable these default messages during runtime. You can use the following runtime options to control the default assertion failure messages:

- `-assert no_default_msg[=SVA|OVA|PSL]`
- `-assert quiet and -assert quiet1`

### Using `-assert no_default_msg[=SVA|OVA|PSL]` Option

The `no_default_msg[=SVA|OVA|PSL]` option disables the display of default failure messages for SVA assertions that contain a fail action block, and OVA and PSL assertions that contain user messages.

The default failure messages are displayed for:

- SVA assertions without fail action blocks
- PSL and OVA assertions that do not contain user messages

When used without arguments, this option affects SVA, OVA, and PSL assertions. You can use an optional argument with this option to specify the class of assertions that should be affected.

#### Note:

The `-assert quiet and -assert report` options override the `-assert no_default_msg` option. That is, if you use either of these options along with `-assert no_default_msg`, then the latter has no effect.

The `-assert no_default_msg=SVA` option affects only SVA.

The `-assert no_default_msg=OVA` and `-assert no_default_msg=PSL` options affect both OVA and PSL assertions, but not SVA.

In addition to the default message, an extra message is displayed by default, for PSL assertions that have a severity (info, warning, error, or fatal) associated with them. This message is considered as a user message, and no default message is displayed, if you use the `-assert no_default_msg[=PSL]` option.

#### Example

Consider the following assertion:

```
As1: assert property (@(posedge clk) P1) else $info("As1 fails");
```

By default, VCS displays the following information for each assertion failure:

```
"sva_test.v", 15: top.As1: started at 5s failed at 5s
Offending 'a'
Info: "sva_test.v", 15: top.As1: at time 5
As1 fails
```

If you use the `-assert no_default_msg` option at runtime, it disables the default message, and displays only the user message, as shown below:

```
Info: "sva_test.v", 15: top.As1: at time 5
As1 fails
```

## Using `-assert quiet` and `-assert quiet1` Options

The `-assert quiet` option disables default messages when assertion fails. The `-assert quiet1` option disables messages and prints the summary at the end. The `-assert quiet` and `-assert quiet1` options are applicable to SVA only.

For example, `-assert quiet1` option disables the default messages, and prints the summary statistics at the end, as follows:

```
Info: "test.sv", 21: top.A1: at time 35 ns
A1 fails
Info: "test.sv", 21: top.A1: at time 45 ns
A1 fails
Summary: 1 assertions, 1 with attempts, 1 with failures
V C S S i m u l a t i o n R e p o r t
Time: 128 ns
CPU Time: 0.380 seconds; Data structure size: 0.0Mb
```

## Options to Control Termination of Simulation

You can use the following runtime options to control the termination of simulation:

`-assert no_fatal_action`

Excludes failures on SVA assertions with fail action blocks for computation of failure count in the `-assert [global_]finish_maxfail=N` runtime option. This option also excludes failures of these assertions for termination of simulation, if you use the following command:

```
$ova_[severity|category]_action(<severity_or_category>, "finish")
```

This option does not affect OVA case violations and OVA or PSL assertions with or without user messages.

Specifying `$fatal()` system task in the fail action block of an SVA assertion or in a fatal severity associated with a PSL assertion, results in termination of simulation irrespective of whether this option is used or not.

This option is useful when you want to exclude failures of assertions having fail action blocks from adding up to the global failure count, for the `-assert [global]_finish_maxfail=N` option.

### Example

Consider the following assertion:

```
As1: assert property (@(posedge clk) P1) else $info("As1 fails");
```

If you use the `-assert global_finish_maxfail=1` option at runtime, then the simulation terminates at the first As1 assertion failure. Now, if you use `-assert global_finish_maxfail=1 -assert no_fatal_action` at runtime, then the failure of assertion As1 does not cause the simulation to terminate.

`-ova_enable_case_maxfail`

Includes OVA case violations in computation of global failure count for the `-assert global_finish_maxfail=N` option.

The `-assert finish_maxfail=N` option does not include OVA case violations. This option maintains a per-assertion failure count for termination of simulation.

### Example

Consider an OVA case pragma, as shown in the following code, to check the case statements for full case violations:

```
reg [2:0] mda[31:0][31:0];
//ova full_case on;
initial begin
 for(i = 31; i >= 0; i = i - 1) begin
 for(j = 0; j <= 31; j = j + 1) begin
 case(mda[i][j])
 1: begin
 testdetect[i][j] = 1'b1;
 end
 endcase
 #1;
 end
 end
end
```

The above code violates full case check. Therefore, case violations are displayed as follows:

```
Select expression value when violation happened for last iteration :
3'b000
```

```
Ova [0]: "ova_case_full.v", 20: Full case violation at time 9 in a
Failed in iteration: [31] [9]
```

By default, these violations are not considered in the failure count for the `-assert global_finish_maxfail=N` option. But if you use the `-ova_enable_case_maxfail` option at runtime, then the case violations are added in the failure count.

## Option to Enable Compilation of OVA Case Pragmas

You can use the following compile-time option to enable compilation of OVA case pragmas:

`-ova_enable_case`

Enables the compilation of OVA case pragmas only when used without the `-Xova` or `-ova_inline` option. All the inlined OVA assertion pragmas are ignored.

`-Xova` or `-ova_inline` is the superset of the `-ova_enable_case` option. They are used to compile both the case pragmas and assertions.

### Example

Consider the following code:

```
//ova parallel_case on;
//ova full_case on; /* case pragma*/
always @(negedge clock)
 case (opcode)
 //ova check_bool (alu_out>10, "ddd", negedge clock); /* assertion pragma
 */
 3'h0: alu_out = accum;
 3'h1: alu_out = accum;
 3'h2: alu_out = accum + data;
 3'h3: alu_out = accum & data;
 3'h4: alu_out = accum ^ data;
 3'h5: alu_out = data;
 3'h6: alu_out = accum;
 endcase
```

The above code contains both OVA case pragmas and assertions. This option ignores the OVA assertion pragmas and compiles only the case pragmas.

## Reporting Values of Variables in the Assertion Failure Messages

You can use the `-assert offending_values` compile-time option to enable reporting of the values of all variables used in the assertion failure messages, as shown:

```
"test.sv", 12: test.a1: started at 5s failed at 5s
 Offending 'ack'
```

```
$rose(sh) = 0
$fell(rst) = 1
ack = 'b0
```

The values of the variables contained in the failing portion of a property are generated using the following formats:

*Table 47 Reporting Formats*

| Variable Type                        | Format                                                                                                                                                                                                                           |
|--------------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Simple scalar bit or logic variables | <var_name> = %b                                                                                                                                                                                                                  |
| Bitvector variables                  | <var_name> = %h                                                                                                                                                                                                                  |
| Int and integer variables            | <var_name> = %0d                                                                                                                                                                                                                 |
| Real, realtime variables             | <var_name> = %g                                                                                                                                                                                                                  |
| Sampled value function calls         | <\$sampled_function_call(argument list)> = %b <b>Note:</b> The values of the variables in the argument list are not reported. Also, if sampled value functions are nested, only the value of the top-level function is reported. |
| Unknown Expressions                  | <expr> = %h                                                                                                                                                                                                                      |
| User-defined function calls          | <function_name>(list of var_name - value pairs)                                                                                                                                                                                  |
| Part select and bit select items     | VCS reports with the sampled value of the selector, if it is a variable.                                                                                                                                                         |

---

## Limitations

The reporting of failing assertion variables is not supported for the following:

- When an assertion is used in the action block of another assertion.
- For other directives, such as `assume`, `restrict`. This feature is supported only for `assert` directive.
- Sequence method
- PSL
- Randomize with

- When the type is one of the following:
  - Parameter/constant
  - Local variable
  - Clocking block variables
  - Dynamic types including class variables

## Using \$uniq\_prior\_checkon and \$uniq\_prior\_checkoff System Tasks

Following is the following syntax for \$uniq\_prior\_checkon and \$uniq\_prior\_checkoff system tasks.

```
$uniq_prior_checkon (level [,list_of_modules_or_instances]);
$uniq_prior_checkoff (level [,list_of_modules_or_instances]);
```

Where,

level is either '0' or '1'.

- '0' applies to the entire hierarchy, that is, the current module and all module instances and child instances.
- '1' applies to the current module/instance only. 'list\_of\_modules\_or\_instances' can be a simple module/instance name or hierarchical path to module/instance.

## Reporting Messages When \$uniq\_prior\_checkon/\$uniq\_prior\_checkoff System Tasks are Called

VCS reports messages when the \$uniq\_prior\_checkon/\$uniq\_prior\_checkoff system tasks are called in the source code or from UCLI. This feature allows you to control assertion failures.

Consider [Example 186](#),

### *Example 186 test.v*

```
module m;
 bit [1:0]a;
 bit b,c;
 initial begin
 repeat (8)
 begin
```

```
#1;
c=1'b1;
a=2'b10;
#1;
a=2'b11;
$monitor($time, "a %0d \n", a);
end
end
always_comb
unique case(a)
2'b10: b=1'b0;
2'b10: b=1'b1;
default: b=1'b0;
endcase
initial
begin
#2 $uniq_prior_checkon();
#8 $uniq_prior_checkoff();
end
endmodule
```

The following output is generated when the  
\$uniq\_prior\_checkon/\$uniq\_prior\_checkoff system tasks are called in the source  
code:

```
Starting Unique/Priority checks at time 2s : Level = 0 arg = * (Source -
test.v,23)
Stopping Unique/Priority checks at time 10s : Level = 0 arg = * (Source -
test.v,24)
```

*Example 187 test.ucli*

```
run 5
call {$uniq_prior_checkoff}
run 2
call {$uniq_prior_checkon}
run
exit
```

Consider [Example 186](#) and [Example 187](#). The following output is generated when the  
\$uniq\_prior\_checkon/\$uniq\_prior\_checkoff system tasks are called from UCLI:

```
Stopping Unique/Priority checks at time 5s : Level = 0 arg = * (from inst
m (UCLI))
Starting Unique/Priority checks at time 7s : Level = 0 arg = * (from inst
m (UCLI))
```

## Assertion and Unique/Priority Re-Trigger Feature

When `$uniq_prior_checkon` system task is called, unique/priority case reports the previous violations. Also, it flushes out all the pending violations recording in the `always_comb` block during the off period. The re-trigger feature reports previous violations only and does not re-execute any process.

Consider [Example 188](#),

*Example 188 example.v*

```
module Top;
 function foo (input a);
 U1: unique case (a)
 1'b1:;
 endcase
 assert #0 (a);
 endfunction
 reg a;
 always_comb begin:AC
 $display($time, "AC is called");
 foo(a);
 end
 always@(*) begin:AS
 $display($time, "AS is called");
 foo(a);
 end
 initial begin
 a = 1'b0;
 end
 initial begin
 $uniq_prior_checkoff(0, Top); $asserton(0,Top);
 #1 $uniq_prior_checkon(0, Top); $asserton(0,Top);
 end
endmodule
```

In this example, `foo` function is called in `always_comb` and `always @` blocks. With this new feature, when `$uniq_prior_checkon` system task is turned on at time 1 ns, then unique or priority messages are triggered from all process blocks.

**Note:**

Other statements inside the `always` block are not executed.

You can compile the example using the following command:

```
% vcs -sverilog -assert svaext example.v
-xlrm uniq_prior_final

% simv
```

VCS generates the following output:

```

Stopping Unique/Priority checks at time 0s : Level = 0 arg = Top (Source
- example.v,21)
Starting assertion attempts at time 0s: level = 0 arg = Top (from inst
Top (example.v:22))
 0AS is called
 0AC is called
 0AC is called
"example.v", 6: Top.foo.unnamed: started at 0s failed at 0s
 Offending 'a'
#0 in foo.unnamed at Example.v:6
#1 in Top at Example.v:15

"example.v", 6: Top.foo.unnamed: started at 0s failed at 0s
 Offending 'a'
#0 in foo.unnamed at example.v:6
#1 in Top at example.v:11

Starting Unique/Priority checks at time 1s : Level = 0 arg = Top (Source
- example.v,23)
Starting assertion attempts at time 1s: level = 0 arg = Top (from inst
Top (example.v:24))

Warning-[RT-NCMUCS] No condition matches in statement
example.v, 3
No condition matches in 'unique case' statement. 'default' specification
is missing, inside Top.foo.U1, at time 1s.

#0 in foo.U1 at Example.v:3
#1 in Top at Example.v:15

Warning-[RT-NCMUCS] No condition matches in statement
example.v, 3
No condition matches in 'unique case' statement. 'default' specification
is missing, inside Top.foo.U1, at time 1s.

#0 in foo.U1 at example.v:3
#1 in Top at example.v:11

```

## Flushing Off the Assertion Re-Trigger Feature

To flush off all these assertion re-trigger feature, you can use the `-assert disable_flush` compile-time or runtime option.

For the example provided above, when you pass the `-assert disable_flush` option at runtime, VCS flushes off all the assertions that are re-triggered.

For example, you can compile and simulate the example using the following commands:

```
% vcs -sverilog example.v -xlrn uniq_prior_final
% simv -assert disable_flush
```

VCS generates the following output:

```
Stopping Unique/Priority checks at time 0s : Level = 0 arg = Top (Source
- example.v,20)
 OAS is called
 OAC is called
 OAC is called
Starting Unique/Priority checks at time 1s : Level = 0 arg = Top (Source
- example.v,21)
```

## Enabling Lint Messages for Assertions

You can use the `+lint=sva` option at compile time to enable lint messages for SystemVerilog Assertions with the rules listed in [Table 48](#). You can also use the `+lint=<ID>` or `+lint=all` compile-time option to enable this feature.

### Note:

For SVA-LDA, SVA-NCRT, and SVA-PWLNT lint IDs, you can use the `+lint=<ID>:<N>:<M>` option to control the display of the lint messages. Here, `N` denotes the number of times the lint message shall appear and `M` denotes the threshold limit to be set. For the remaining lint IDs specified in the [Table 48](#), you can use the `+lint=<ID>:<N>` option to control the display of messages.

*Table 48 List of New LINT IDs*

| Assertion Rule Description                                            | Lint Message                                               |
|-----------------------------------------------------------------------|------------------------------------------------------------|
| Assertion with large delays                                           | Lint-[SVA-LDA] Large delay assertion                       |
| Consequent contains throughout operator on non-consecutive repetition | Lint-[SVA-NCRT] Non-consecutive repetition in 'throughout' |
| Assertions using \$past with >5 clock cycles                          | Lint-[PWLNT] PAST with large number of ticks               |
| Non-singular edge found in assertion clock                            | Lint-[SVA-NSEF] Non-singular edge found                    |
| Complex clock expression is used with an assertion                    | Lint-[SVA-CE] Complex expression found                     |
| Non-sampled variable used in the action block of an assertion         | Lint-[SVA-NSVU] Non-sampled variable used                  |

*Table 48 List of New LINT IDs (Continued)*

| Assertion Rule Description                                            | Lint Message                                             |
|-----------------------------------------------------------------------|----------------------------------------------------------|
| Assertions using local variables                                      | Lint-[SVA-LVU] Local variable used                       |
| Assertion declared outside a module                                   | Lint-[SVA-ADOM] Assertion declared outside module        |
| Disable iff expression used inside an assertion                       | Lint-[SVA-DIU] 'disable iff' used in assertion statement |
| Assertions used inside generate "for" loop                            | Lint-[SVA-AGFL] Assertions in generate for loop          |
| Unnamed assertion                                                     | Lint-[SVA-UA] Unnamed Assertion                          |
| Cover on a sequence                                                   | Lint-[SVA-SCU] Sequences inside 'Cover' statement        |
| Deferred assertion                                                    | Lint-[SVA-DAU] Deferred assertion used                   |
| Assertions using antecedent expression that results in an empty match | Lint-[SVA-AEM] Antecedent empty match                    |
| Consequent expression of an assert property is always true            | Lint-[SVA-CAT] Consequent always true                    |
| Implication in a cover property                                       | Lint-[SVA-ICP] Implication in cover property             |
| Using (clk iff gate_expr) inside an assert property                   | Lint-[SVA-LCE] Logical 'AND' in clock expression         |
| Pass action block in assert property                                  | Lint-[SVA-PAB] Pass action block                         |
| \$info/\$display statement in a cover property                        | Lint-[SVA-IUC] Info messages used in cover               |
| Assertion with empty "begin --end" action block                       | Lint-[SVA-EFAB] Empty fail action block                  |
| Assertion in a loop without using the loop index                      | Lint-[SVA-FINUA] For-loop index not used in assertion    |
| Assertion with a severity task in the pass action block               | Lint-[SVA-STPAB] Severity task in pass action block      |

## Fail-Only Assertion Evaluation Mode

Fail-only is an assertion evaluation mode by which VCS provides an optional optimization controlled by the `-assert failonly` compile-time option. This option enables fail-only mode for concurrent assertions.

Immediate/deferred assertions and concurrent assertions without pass action blocks, local variables, match operators, or multiple clocks tend to benefit from this evaluation mode.

**Note:**

VCS ignores the fail-only assertion evaluation mode, if you use any of the following options:

```
-assert enable_diag, -debug_access, -cm assert
```

By default, VCS reports the start and end times of assertion evaluation attempts. Therefore, each attempt needs to be stored in memory until it matures. This can cause slowdown if there are multiple attempts pending till the simulation is finished.

If you use the `-assert failonly` option at compile time, VCS reports only assertion failures with their end times. This reduces the memory footprint and speeds up the simulation.

Consider [Example 189](#).

*Example 189 test.v*

```
`timescale 1ns/1ns
module top();
 reg a,b,c;
 reg clk = 0;

 A1: assert property(@(posedge clk) a ##1 !c[*1:$] |-> b);

 always
 #1 clk = ~clk;
 initial begin
 a = 1;
 b = 0; c = 0;

 #100 b = 1;
 #2 $finish;
 end
endmodule
```

The following output is generated with the `-assert failonly` option:

```
"test.v", 6: top.A1: failed at 101ns
```

In the above output, VCS reports only end times.

The following output is generated without `-assert failonly` (default mode) option:

```
"test.v", 6: top.A1: started at 99ns failed at 101ns
 Offending 'c'
"test.v", 6: top.A1: started at 97ns failed at 101ns
 Offending 'c'
.....
"test.v", 6: top.A1: started at 1ns failed at 101ns
 Offending 'c'
```

In the default mode, VCS reports start and end times for each failing attempt.

## Key Points to Note

- VCS reports only assertion failures with their end times.
- VCS reports only a single failure for multiple failures of a single assertion maturing at the same time.
- Behavior with coverage:
  - Reports unique matches only
  - The `-cm assert` option is not supported with the `-assert failonly` option.

Consider [Example 190](#).

### *Example 190 cov.v*

```
module top;
 reg a,b,c;
 reg clk=0;
 C2: cover property(@(posedge clk) b[=1:$] ##1 c);
 initial begin
 a=1;b=0;c=1;
 #25 b=1;
 #110 $finish;
 end
 always #5 clk=~clk;
endmodule
```

The following output is generated with the `-assert failonly` option:

```
"cov.v", 7: top.C2, 9 match
```

[4 succeeds at same time 45] hence counted only once hence count is 9 vs 12 in default mode

The following output is generated without the `-assert failonly` (default mode) option:

```
"cov.v", 7: top.C2, 13 attempts, 12 match
```

## Limitations

The feature has the following limitations:

- Reporting of offending expressions upon failure is not supported.
- Success reporting (including vacuous success) is not supported.
- VPI callback on success (including vacuous success) is not supported.
- Attempt start time reporting is not supported.

## Treating x as true on an Assertion Precondition

VCS treats x values as FALSE while evaluating expressions. Therefore, it is not possible to see an assertion failure when the antecedent sequence expression is not satisfied. For example, consider the following assertion:

```
known_a : assert property (@(posedge clk) value_p |-> value_o);
```

In the above expression, `value_o` (consequent property expression) is evaluated only when `value_p` (antecedent sequence expression) is 1. VCS does not evaluate consequent property expression when `value_p` is x. This behavior is in compliance with *IEEE System Verilog LRM 1800-2012*.

However, VCS enables the feature to treat x as TRUE while evaluating the antecedent of assertions using the `-assert allow_antec_x` option. Therefore, for the following assertion:

```
known_a : assert property (@(posedge clk) value_p |-> value_o);
```

When `-assert allow_antec_x` option is used, VCS evaluates consequent property expression when `value_p` is x. Therefore, `value_p` (antecedent sequence expression) is considered TRUE and `value_o` (consequent property expression) is evaluated.

## Use Model

To enable this feature, use the `-assert allow_antec_x` option at compile time. Following is the use model:

```
% vcs <filename> -sverilog -assert allow_antec_x
```

## Usage Example

The following example illustrates a case where evaluation of antecedent of assertion is considered TRUE.

*Example 191 test.v*

```
module top;

bit clk;

logic a = 1'bx;
logic b = 1'bx;
logic c = 1'bx;

always clk = #2 ~clk;

always@(*)
 c = a & b;

initial begin
 a = 1'b0; b = 1'b0;
 #2 a = 1'b1; b = 1'b0; $display($time,", a = %h, b = %h, c = %h",
 a,b,c);
 #2 a = 1'bx; b = 1'b1; $display($time,", a = %h, b = %h, c = %h",
 a,b,c);
 #2 a = 1'b1; b = 1'b1; $display($time,", a = %h, b = %h, c = %h",
 a,b,c);
 #2 a = 1'bx; b = 1'b0; $display($time,", a = %h, b = %h, c = %h",
 a,b,c);
 #2 a = 1'bx; b = 1'b1; $display($time,", a = %h, b = %h, c = %h",
 a,b,c);
 #2 a = 1'bx; b = 1'bx; $display($time,", a = %h, b = %h, c = %h",
 a,b,c);
 #2 a = 1'b0; b = 1'b1; $display($time,", a = %h, b = %h, c = %h",
 a,b,c);
 #20 $finish;
end

A1 : assert property (@(edge clk) (a & b) |-> c);

endmodule
```

The compile command is as follows:

```
% vcs test.v -sverilog -assert allow_antec_x
```

Following is the simulation command:

```
% simv
```

The output is as follows:

```
2, a = 1, b = 0, c = 0
4, a = x, b = 1, c = 0
6, a = 1, b = 1, c = x
"test.v", 26: top.A1: started at 6s failed at 6s
Offending 'c'
8, a = x, b = 0, c = 1
10, a = x, b = 1, c = 0
12, a = x, b = x, c = x
"test.v", 26: top.A1: started at 12s failed at 12s
Offending 'c'
14, a = 0, b = 1, c = x
"test.v", 26: top.A1: started at 14s failed at 14s
Offending 'c'
```

## Using SystemVerilog Constructs Inside vunits

VCS supports using SystemVerilog and SystemVerilog Assertions inside a Property Specification Language (PSL) verification unit (vunit). This feature enables the following:

- Allows SV or SVA code inside a vunit.
- Allows you to easily bind the checkers containing assertions and model a vunit code to a design.
- Allows module instantiation inside a vunit.

Use the `-assert svvunit` compile-time option, as shown in the following example to enable this feature. You can specify this option with the `vcs` or `vlogan` command as follows:

```
% vcs -assert svvunit <filename.v>
<design_filename.v> \ <vunit_filename.psl>
```

or

```
% vlogan -assert svvunit <filename.v> <design_filename.v> \
<vunit_filename.psl>
```

where,

<vunit\_filename.psl> is the PSL vunit that contains the SV or the SVA code. For example:

```
% vcs -assert svvunit test.v design.v vunit_checker.psl
```

## Limitations

The feature has the following limitations:

- Inheritance of vunits is not supported.
- vunit binding is supported only for modules.
- SV checker (checker/endchecker construct) instantiation in vunit is not supported.
- VHDL entity instantiation in vunit is not supported.
- In the above use model, you cannot specify PSL constructs in any vunit specified in the same vlogan command. You must separate the PSL vunits from SV vunits and use them in two separate compilations, as shown in the following examples:

```
% vlogan vunit_psl.psl design.v
% vlogan -assert svvunit vunit_sv.psl test.v
```

where,

- vunit\_psl.psl is a vunit that contains PSL code
- vunit\_sv.psl is a vunit that contains SV or SVA code.

## Calling \$error Task When Else Block is Not Present

You can use the following runtime option to enable the calling of the \$error task for the assert statements as per the IEEE 1800-2012 SystemVerilog LRM:

```
-assert error_default_action_block
```

If you use this option, VCS calls the \$error task if an assert statement fails and else clause is not specified. This feature is supported for immediate, deferred, and concurrent assertions.

Consider the following test case (test.v):

```
module assertit;
 logic clk = 0; initial repeat (10) #1 clk++;
 logic [2:0] a = 0; always @(posedge clk) a <= a + 1;
 al: assert property (@(posedge clk) !a[2]);
```

```
a2: assert property (@(posedge clk) !a[2]) else $error("failed");
endmodule
```

The output for both asserts (`a1` and `a2`) are identical for the preceding test case as follows:

```
"test.v", 6: assertit.a1: started at 9s failed at 9s
 Offending '(!a[2])'
Error: "test.v", 6: assertit.a1: at time 9
"test.v", 7: assertit.a2: started at 9s failed at 9s
 Offending '(!a[2])'
Error: "test.v", 7: assertit.a2: at time 9
failed
```

## Disabling Default Assertion Success Dumping in -debug\_access Option

By default, VCS does not dump the SystemVerilog assertion successes to the VPD file when the `-debug_access` option is enabled. This optimization can improve the VCS performance.

You can use the `-assert dumpsuccess` option at runtime to enable dumping of the SystemVerilog assertion successes to the VPD file.

## Support for Success Count With -assert summary Option by Default

VCS allows you to view the assertion success count with the `-assert summary` option even if you do not specify the `-assert success` option.

### Use Model

The following is the use model of this feature:

- `% vcs <filename> -assert enable_diag`
- `simv -assert summary`

### Usage Example

*Example 192 test.v*

```
module top;
logic clk;
bit a;
```

```
always #1 clk = ~clk;
A1: assert property (@(posedge clk) a);

initial begin
 clk = 0;
 #10 a = 1;
 #20 $finish;
end
endmodule
```

Run the `test.v` file with the following commands:

- % vcs test.v -assert enable\_diag
- % simv -assert summary

The following output is generated:

```
"test.v", 6: top.A1: started at 1s failed at 1s
Offending 'a'
"test.v", 6: top.A1: started at 3s failed at 3s
Offending 'a'
"test.v", 6: top.A1: started at 5s failed at 5s
Offending 'a'
"test.v", 6: top.A1: started at 7s failed at 7s
Offending 'a'
"test.v", 6: top.A1: started at 9s failed at 9s
Offending 'a'
$finish called from file "test.v", line 11.
$finish at simulation time 30

===== Generating Assertion Summary =====

"top.A1", 15 attempts, 10 successes, 5 failures, 0 incompletes

===== End Assertion Summary =====
```

Therefore, the `-assert summary` option, reports the number of assertions successes by default.

## Disabling Success Callbacks

You can disable the success and attempts callbacks on assertions using the `-assert skip_success_cb` runtime option. This option improves the runtime performance by generating only the failure callbacks.

## Use Model

The following is the use model of this feature:

```
% vcs -sverilog -debug_access <filename1.tab, filename2.c, filename.v>
```

Where, `filename1.tab` and `filename2.c` files are applicable only for VPI callbacks.

```
% simv -assert skip_success_cb
```

**Note:**

If you use `-debug_access+all` option during compile time, then the `-assert skip_success_cb` run time option is ignored and all the callbacks are generated.

## Enhancement to Assertion Diagnostics

In some designs, assertion can never get triggered or no attempt can happen in the following scenarios:

- when assertions are disabled using `$assertoff/assertkill`
- when there is no clock activity happening
- when antecedent of the implication never satisfies the condition

The `-assert verbose` runtime option gives information about assertions with no attempt in the simulation log. Specifically, the `-assert verbose` option displays the following information about the assertions where no attempt started:

- the reason for “No attempt started” for each assertion where no attempt started due to `$assertoff` or `$assertkill` system task.
- the number of assertions where no attempt started due to no clock activity in the simulation log and assert report.
- the count of assertions based on the reason the attempts were not started in the simulation log and assert report.

## Use Model

Use the following option at runtime:

```
% simv -assert verbose=noattempt -assert report=<filename>
```

**Note:**

When assertion is not triggered because the antecedent of the implication never satisfies the condition, use the `-assert enable_diag` option at compile time to get additional information that can impact the runtime performance. For example,

```
% vcs -assert enable_diag <other_options>
```

## Usage Example

Consider the following testcase:

*Example 193 test.v*

```
module top;
 logic clk1, clk2, rst0, cnd0, sig0;
 initial
 begin
 clk1 = 0;
 rst0 = 1'b0;
 rst0=1;
 sig0 = 1'b0;
 cnd0 = 1'b0;
 end
 always #6 clk1 = ~clk1;
 m m1(clk1); //clk present
 m m2(clk2); //No attempt present due to no clk period present
 initial $assertoff(0,top.m1.A1); //top.m1.A1 assertions turned off
 initial #10 $finish;

 ma0: assert property (@(posedge clk1) disable iff(rst0) cnd0 |->
 sig0); // when antecedent of the implication never satisfies the
 condition

endmodule

module m(input logic clk);
 A1 : assert property(@(posedge clk) 1);
endmodule
```

Run the example using the following commands:

- % vcs -sverilog test.v -full64 -assert enable\_diag
- % simv -assert verbose=noattempt -l new.log

Or,

- % simv -assert verbose=noattempt -assert report=report1

Following is the output:

```
"test.v", 17: top.ma0: Antecedent of the implication never satisfied.
"test.v", 21: top.m1.A1: No attempt started, Disabled via assertcntrl
"test.v", 21: top.m2.A1: No attempt started, No clk toggle
Summary: 3 assertions, 1 with attempts, 0 with failures,
Antecedent of the implication never satisfied : 1, No clk toggle: 1,
Disabled via assertcntrl: 1
```

In this example, for every “No attempt started,” the reason for the no attempt is displayed and also the number of assertions where no attempt started due to clock activity. Also, you can see that the count of assertions based on the reasons the attempts were not started. You can also get the same information from the `report1` file.

## Support of String Variable to Assertion Control System Tasks

VCS supports string data type variable for specifying hierarchy in the following assertion control system tasks:

- `$assertoff`
- `$asseron`
- `$assertkill`
- `$assertcontrol`

## Use Model

Use the following use model to enable this feature:

```
$vcs -sverilog -full64 example.v -R
```

## Example

Following is the example of this feature:

*Example 194 example.v*

```
module top();
 bit clk,a;
 always #3 a = ~a;
 always #1 clk =~ clk;
 initial #30 $finish;
```

```

mid m1(clk);

string s1 = "A1";

initial begin
 A1: assert #0 (a == 1'b1);
end

initial begin
 #5; $assertoff(0,s1);
 #5; $asserton(0,s1);
 #5; $assertkill(0,top.m1.s2);
 #5; $assertcontrol(4,1,1,1,s1);
end

endmodule

module mid(input bit clk);

bit a,b;

always #3 b = ~b;

always #6 a = ~a;

string s2 = "top.m1.A2";

A2: assert property (@(posedge clk) a |-> ##2 b);

endmodule

```

## Simulation Log:

Execute the following command to run the above example:

```
$finish called from file "example.v", line 9.

$finish at simulation time 30
```

## Limitation

This feature is not supported for `control_type` values (1,2,6,7,8,9,10,11) for the `$assertcontrol` system task.

## Support for Automatic Variables in Sampling Functions

VCS supports automatic variables in the sampled system function calls like `$sampled`, `$past`, `$rose`, `$fell`, and `$changed` in procedural context.

## Use Model

The use model is as follows in both VCS two-step and three-step flows:

```
% vcs -sverilog -assert auto_var_in_sampled <other_options> <file_name>
% simv
```

## Usage Examples

### *Example 195 test.v*

```
module top;
 bit clk;
 reg q;
 always #5 clk=~clk;
 logic [1:0] arr[1:0];
 initial begin
 arr = {2'b10,2'b10};
 #10 arr = {2'b01,2'b01};
 #15 $finish();
 end
 always @(posedge clk) begin
 for (int i=0; i<2; i++) begin
 for (int j=0; j<2; j++) begin
 q <= $past(arr[i][j]);
 $display("value of $past(arr[%0d] [%0d])

at time %0t = %0b",i,j,$time,$past(arr[i][j]));
 end
 end
 end
endmodule
```

Use the following option at compile time:

```
% vcs -sverilog -assert auto_var_in_sampled
% simv
```

The output generated is as follows:

```
value of $past(arr[0][0]) at time 5 = x
value of $past(arr[0][1]) at time 5 = x
value of $past(arr[1][0]) at time 5 = x
value of $past(arr[1][1]) at time 5 = x
value of $past(arr[0][0]) at time 15 = 0
value of $past(arr[0][1]) at time 15 = 1
value of $past(arr[1][0]) at time 15 = 0
value of $past(arr[1][1]) at time 15 = 1
```

---

## Limitation

The limitation of this feature is as follows:

- VCS does not support automatic variables in sampling functions if the select index is in between a cross-module reference (XMR).

## List of supported IEEE Std. 1800-2012 Compliant SVA Features

The following features are supported in compliance with the *IEEE SystemVerilog LRM Std 1800TM-2012*. For more details, see IEEE 1800TM-2012 Standard for SystemVerilog—Unified Hardware Design, Specification, and Verification Language Manual.

- Overlapping operators in multiclock environment
- Immediate and Deferred Assertions
- `weak` and `strong` sequence operators

For limitations, see the [Limitations](#) section.

- Implication and equivalence operators (`->` and `<->`)
- `until` operator in four variants:
  - `until`
  - `s_until`
  - `until_with`
  - `s_until_with`

- **nexttime operator in four variants:**
  - nexttime property\_expr
  - nexttime [N] property\_expr
  - s\_nexttime property\_expr
  - s\_nexttime [N] property\_expr
- **always operator in three variants:**
  - always property\_expr
  - always [cycle\_delay\_const\_range\_expression] property\_expr
  - s\_always [constant\_range] property\_expr strong
- **eventually operator in three variants**
  - eventually [constant\_range] property\_expr
  - s\_eventually property\_expr
  - s\_eventually
- **followed-by operator (#-, #=)**
- **accept\_on and reject\_on abort conditions**
- **Inferred value functions:**
  - \$inferred\_clock
  - \$inferred\_disable
  - \$inferred\_enable

`$inferred_enable` is a VCS extension to the Inferred functions and not a standard LRM feature.

- Local variable initialization and input
- Global clocking
- Global clocking past value functions:
  - \$past\_gclk(expression)
  - \$rose\_gclk(expression)
  - \$fell\_gclk(expression)

- \$stable\_gclk(expression)
- \$changed\_gclk(expression)
- Global clocking future value functions:
  - \$future\_gclk(expression)
  - \$rising\_gclk(expression)
  - \$falling\_gclk(expression)
  - \$steady\_gclk(expression)
  - \$changing\_gclk(expression)
- let construct
- Checker endchecker construct:
  - Checker declarations
  - Checker instantiation
  - Procedural and Static checkers
  - Default clocking in checkers
  - Default disable in checkers
  - Checker instantiation in a procedural for loop
  - Random variable support in checker

To enable the checker endchecker construct, you must use the -assert checker option at compile time.

- Immediate and deferred assertions
- edge operator
- Bit/part/field select support in the let operator
- Elaboration system tasks
- VPI support

## Support for \$countbits System Function

VCS supports the IEEE 1800-2012 SystemVerilog LRM system function `$countbits`. The following `$countbits` function counts the number of bits that have a specific set of values (for example, 0, 1, X, Z) in a bit vector:

```
$countbits (expression, control_bit { , control_bit })
```

This function returns an integer value equal to the number of bits in `expression` whose values match one of the `control_bit` entries. For example:

- `$countbits (expression, '1)` returns the number of bits in `expression` having value 1.
- `$countbits (expression, '1, '0)` returns the number of bits in `expression` having values 1 or 0.
- `$countbits (expression, 'x, 'z)` returns the number of bits in `expression` having values X or Z.

This support allows efficient implementation of basic non-temporal assertions in the presence of unknown values.

## Support for Real Data Type Variables

VCS supports the following as per the IEEE 1800-2012 SystemVerilog LRM:

- Real data type variables in the sub-expressions of a concurrent assertion.
- Sampling of real data type variables similar to the sampling of any integral variable.

## Support for \$assertcontrol Assertion Control System Task

VCS supports the IEEE 1800-2012 SystemVerilog LRM assertion control system task `$assertcontrol`. The `$assertcontrol` system task provides the capability to enable, disable, or kill the assertions based on the assertion type or directive type. Similarly, this task also provides the capability to enable or disable action block execution of assertions and expect statements based on the assertion type or directive type.

Syntax:

```
$assertcontrol (control_type [, [assertion_type]
[, [directive_type] [, [levels] [,
list_of_scopes_or_assertions]]]]) ;
```

The `$assertcontrol` system task provides finer assertion selection controls than `$asserton`, `$assertoff`, and `$assertkill` system tasks.

## Limitations

The feature has the following limitations:

- The `Unique0` assertion type is not supported.
  - The assertion type 16 (`Expect` statement) is not supported.
  - The control type values from 6 to 11 (`PassOn`, `PassOff`, `FailOn`, `FailOff`, `NonvacuousOn`, `VacuousOff`) are not supported.
  - The control type `Kill` is not supported for `Unique` and `Priority` assertion types.
- 

## Enabling IEEE Std. 1800-2012 Compliant Features

You must use the `-assert svaext` compile-time option to enable the IEEE Std. 1800-2012 compliant SVA features.

## Limitations

The feature has the following limitations:

- In VCS, strong and weak properties are not distinguished in terms of their reporting at the end of simulation. In all cases, if a property evaluation attempt did not complete evaluation, it is reported as unfinished evaluation attempt, and allows you to decide whether it is a failure or a success.
- Checker declaration are allowed in unit scope only.
- Bind construct with checkers is not supported.

The limitations on debug support are as follows:

- UCLI support for new assertions is not supported.
- 

## Support for Strong Operators in Assertions

VCS supports the `s_eventually` operator in the following four contexts in Assertions.

- The `s_eventually` operator is used as a top-level operator in a property expression.

For example,

```
L1: assert property (@(posedge clk) disable iff (rst) s_eventually (a
##[1:$] b));
```

- The `s_eventually` operator is used as a top-level operator of the consequent expression.

For example,

```
(antecedent|-> s_eventually (consequent))

L2: assert property (@(posedge clk) disable iff(rst) req |->
s_eventually (a ##[1:40] b));
```

- The `s_eventually` operator is applied to the last consequent expression in the implication chain.

For example,

```
antecedent |=> consequent1 |=> s_eventually(consequent2);

L3: assert property (@(posedge clk) req |-> ack [*1:3] |=>
s_eventually (a ##[1:$] b));
```

- Property operator with one operand using the `s_eventually` operator and all other operands are non-temporal.

For example,

```
property pl;
 req |-> s_eventually {##[1:$] ack}; // strong operator
end property;

property p2;
 ack && data; // boolean expression
end property;

property p3;
 grant |-> data; // only overlapping implication
end property

L4: assert property (@(posedge clk) disable iff(rst) pl and p2 and
p3);
```

#### Note:

VCS does not support the usage of the following strong operators in Assertions and issues the [SVA-SONS] error message.

- Usage of `s_eventually` other than the four contexts mentioned above
- Usage of `s_eventually` with other unsupported strong operators
- `s_nexetime`

- s\_always
- s\_until
- s\_until\_with

## Suppressing the Run-time out of Bound Access Messages

VCS provides `-suspend_boundscheck` run-time option for suppressing the run-time out of bound access messages using the `-boundscheck` compile option.

The following system tasks can be used to resume the bounds check during simulation:

- `$xprop_assert_on("xindex-rd");`
- `$xprop_assert_on("xindex-wr");`

## Use Model

The following is the use model of this feature:

```
% vcs -sverilog -full64 top.v -boundscheck -debug_acc
% simv -suspend_boundscheck
```

## Example

Consider following testcase or example for this feature:

### *Example 196 top.v*

```
module test(input wire index);
 logic [3:0] mem[1:0];
 reg [7:0] i;

 initial begin
 i = 4;
 mem[i] = 0;
 $display("Done");
 end

```

```
endmodule
```

---

## Output

Execute the following commands:

- Execute the following compilation command:

```
% vcs -sverilog -full64 top.v -boundscheck -debug_acc
```

- Execute the following runtime command without `-suspend_boundscheck` option:

```
% simv
```

Simulation output:

```
Warning-[AOOBWW] Array out of bounds write
top.v, 7
test
 Array write "mem[4]" out of bounds.
 Inside instance 'test', at simulation time 0
 Please make sure index is within range.
```

**Done**

- Execute the following runtime command with the `-suspend_boundscheck` option:

```
% simv -suspend_boundscheck
```

Simulation output:

**Done**

---

## Reporting Runtime Violations for Unique/Unique0/Priority For and Foreach Statements

VCS reports runtime violations for `unique-foreach`, `unique0-foreach`, `unique-for`, `unique0-for`, `priority-foreach`, and `priority-for` statements in the following cases:

- In `unique` statement:
  - VCS reports a violation when the `if ()` statement inside the loop is true for more than one iteration.
  - VCS reports a violation when the `if ()` statement inside the loop is not true for any iteration.

- In **unique0** statement:
  - VCS reports a violation when the `if ()` statement inside the loop is true for more than one iteration.
- In **priority** statement:
  - VCS reports a violation when the `if ()` statement inside the loop is not true for any iteration.

**Note:**

- Unique/unique0/priority is only applicable for the `for` and `foreach` loops.
- The statement of `for/foreach` can only be an `if ()` statement without `else`.
- There should not be any break inside the `if ()` statement for the Unique/unique0/priority loop.

**Example***Example 197 test.sv*

```
module top;
bit b, select[1:2];
int my_selected_vector, my_vectors[1:2] = '{24,48};
always@(b) begin
 unique for (int entry = 1; entry < 3;)
 if (select[entry])
 my_selected_vector = my_vectors[entry];
end
initial begin
 select = '{1,1};
#5 b = 1;
#1 $display ("my_selected_vector: %p", my_selected_vector);
end
endmodule
```

**Command line:**

```
% vcs -sverilog test.sv -R
```

**The generated output is as follows:**

```
Warning-[RT-MTOCMULP] If condition is true for more than one iteration in
loop
test.sv, 5
If condition is true for multiple iteration of 'unique' for/foreach loop
statement inside top.unnamed$$_0, at time 5s.
```

```
my_selected_vector: 24
```

## Limitation

The following is the limitation of this feature:

- You cannot suppress the violations for priority-for, priority-foreach, unique-for, unique-foreach, unique0-for, or unique0-foreach for zero-delay glitches. The -xlmr uniq\_prior\_final option does not affect these constructs.

## Using -assert errmsg Runtime Option

Use the `-assert errmsg` runtime option to report the error message along with the assertion failure message.

For example,

If the assertion contains any severity (error, info, or warning) in the fail action block, VCS reports the string as *ERROR* as follows:

- With `-assert errmsg assert_ab: assert property (ab) $info("PASS"); else $error("FAIL");`  
*ERROR: "test.sv", 38: test.assert\_ab: started at 90ns failed at 90ns Offending 'a'*

If the assertion does not contain any severity in the fail action block, VCS reports the string as *Error* as follows:

- With `-assert errmsg assert_ab: assert property (ab); Error: "test.sv", 38: test.assert_ab: started at 90ns failed at 90ns Offending 'a'`

### Note:

If you do not use the `-assert errmsg` runtime option, the string (*ERROR* or *Error*) is not displayed along with the assertion failure message though you specify any severity in the fail action block.

## Typed Formal Support for Sequence and Property Using -assert typed\_formal Option

The formal argument of a sequence and property can be typed and untyped. If type of actual and formal are not same then actual type needs to be type casted to formal type.

When datatype of actual expression and formal expression is not same, VCS adds the support for all the integral datatype (such as int, integer, packed struct, packed

vector) except enumeration type and anonymous packed struct, using the -assert typed\_formal option:

### Use Model

Following is the use model of this feature:

```
vcs -sverilog <file_name> -assert typed_formal
```

### Examples

#### Packed Struct as Typed Formal

##### *Example 198 test1.v*

```
module sva;
 logic clk = 0;
 typedef struct packed {
 logic [4:0] reqid;
 logic snoop;
 } payload;
 payload fe_slreq;
 payload fe_slreq1 = '1;
 typedef struct packed {
 logic valid;
 payload payload;
 } pkt;

 property p(pkt f2);
 @ (clk) (f2.valid == 0);
 endproperty : p
 pkt p2;
 C1 : assert property (p(fe_slreq));
 C2 : assert property (p(fe_slreq1));

 always #1 clk = ~clk;
 initial #500 $finish();
 initial repeat (20) begin
 #5 fe_slreq1 = '1;
 fe_slreq = '0;
 end
endmodule
```

Execute the following command to run the above example:

```
vcs -sverilog test1.v -R -assert typed_formal
```

Following is the output:

```
"test1.v", 25: sva.C1: started at 0s failed at 0s
 Offending '(fe_slreq.valid == 0)'
"test1.v", 25: sva.C1: started at 1s failed at 1s
 Offending '(fe_slreq.valid == 0)'
```

```
"test1.v", 25: sva.C1: started at 2s failed at 2s
 Offending '(fe_slreq.valid == 0)'
"test1.v", 25: sva.C1: started at 3s failed at 3s
 Offending '(fe_slreq.valid == 0)'
"test1.v", 25: sva.C1: started at 4s failed at 4s
 Offending '(fe_slreq.valid == 0)'
"test1.v", 25: sva.C1: started at 5s failed at 5s
 Offending '(fe_slreq.valid == 0)'
$finish called from file "test1.v", line 30.
$finish at simulation time 500
 V C S S i m u l a t i o n R e p o r t
```

## Packed Vector as Typed Formal

### *Example 199 test2.v*

```
module top;
bit clk;
bit [31:0] reg2=32'b011;
sequence prop1 (reg [2:0] p1);
@(posedge clk) (5'b10010 == {2'b10,p1});
endsequence
A1: assert property(prop1(reg2));
initial #5 clk = ~clk;
endmodule
```

Execute the following command to run the above example:

```
vcs -sverilog test2.v -R -assert typed_formal
```

Following is the output:

```
"test2.v", 7: top.A1: started at 5s failed at 5s
 Offending '(5'b10010 == {2'b10, reg2})'
 V C S S i m u l a t i o n R e p o r t
```

## Limitations

The following are the limitations of this feature:

- Does not support if formal datatype is enumeration.
- Actual expression with function call or local variable s not supported.
- Anonymous packed struct or packed dimension as typed formal.
- Vector of packed struct where packed dimension is declared in typed formal.

## Disabling Sequence Debugging Using -assert no\_seqdebug Option

VCS stops the sequence debugging even with +vpi using the -assert no\_seqdebug compile time option except for the following options:

- -assert vpiSeqFail
- -assert vpiSeqBeginTime

### Use Model

The following is the use model for this feature:

```
vcs -sverilog <file_name> -assert no_seqdebug +vpi
```

If -assert no\_seqdebug option is used with any of the above options, the following downgradable CT warning message is displayed.

Warning-[SVA-SEQDEBUG] Switch -assert no\_seqdebug will be ignored  
 It cannot be used with switch : '-assert vpiSeqFail'.

Warning-[SVA-SEQDEBUG] Switch -assert no\_seqdebug will be ignored  
 It cannot be used with switch : '-assert vpiSeqBeginTime'.

If \$assert\_monitor task is not compatible with -assert no\_seqdebug option, the following compile error message is displayed.

### *Example 200 test.v*

```
module test;
reg clk = 0;
reg clk1 = 0;
int PTA;
byte PBA;

always clk = #1 ~clk;

initial
begin
PTA = 0;
PBA = 0;
#20 $finish();
end

always@ (posedge clk)
begin
PTA = PTA +1;
end

always@ (negedge clk)
```

```

begin
PBA = PBA +1;
end

sequence seq; @ (posedge clk)
PTA == PBA;
endsequence

PTABA : assert property (@(posedge clk) seq);

initial $assert_monitor(PTABA);

endmodule

Error-[SVA-NOSEQDEBUG] Illegal use of $assert_monitor task
test.v, 33
test, "$assert_monitor(test.PTABA);"
'$assert_monitor' task seen in presence of -assert no_seqdebug switch.
'-assert no_seqdebug' cannot be used if '$assert_monitor' tasks are
used in
design.

```

**Sequence callbacks does not work with -assert no\_seqdebug option, the following runtime warning message is displayed.**

```

Warning-[SVA-NOSEQCB] VPI based sequence marker callback will be ignored
Switch -assert no_seqdebug incompatible with

'cbAssertionAttemptMarker/cbAssertionSeqInstMatched/cbAssertionSeqInstEn
dMatched'
callback. To enable sequence debugging with VPI callbacks please
recompile
the design without the switch.
This switch is used for better performance.

```

## Limitations

Following are the limitations of this feature:

- -assert no\_seqdebug option cannot be used with the following +vpi options, if used warning messages are displayed as described above.
  - -assert vpiSeqFail
  - -assert vpiSeqBeginTime
- The following sequence callbacks are not supported with -assert no\_seqdebug option.
  - cbAssertionAttemptMarker
  - cbAssertionSeqInstMatched

- cbAssertionAttemptEndMarker
- cbAssertionAttemptBeginMarker
- cbAssertionSeqInstEndMatched
- cbAssertionSeqInstBeginMatched
- cbAssertionSeqInstFailed

## Assertion Summary Output With -assert global\_finish\_maxfail=<N> and -error=SVART-NAFRLTS Options

VCS displays assertion failures in the assertion summary through the `-assert summary` option. This summary also includes the failure messages that were not reported earlier because some of the warning message types were upgraded to error with the `-error=SVART-NAFRLTS` runtime option. When the `-assert global_finish_maxfail=<N>` option is used along with the `-error=SVART-NAFRLTS` option, the number of assertion failures displayed during simulation becomes consistent with the number of failures reported in the assertion summary report.

### Note:

VCS executes the already evaluated assertions even after the failure message count is more than <N>. You may see more than <N> number of assertion failures with the `-assert global_finish_maxfail=<N>` option.

### Use Model

The following is the use model for this feature:

```
%> vcs -sverilog test.sv -assert enable_diag
%> simv -assert global_finish_maxfail=3 -error=SVART-NAFRLTS -assert
summary
```

#### Example: test.sv

```
module top;

reg clk= 0, reset = 1, err = 0;

assert_mod_1 inst1 (.clk(clk), .reset(reset), .err(err));
assert_mod_1 inst2 (.clk(clk), .reset(reset), .err(err));
assert_mod_1 inst3 (.clk(clk), .reset(reset), .err(err));
assert_mod_1 inst4 (.clk(clk), .reset(reset), .err(err));
assert_mod_1 inst5 (.clk(clk), .reset(reset), .err(err));
assert_mod_2 inst6 (.clk(clk), .reset(reset), .err(err));

initial begin
```

```

#4; reset = 0; err = 1;
#4; reset = 1;
#4 $finish();
end
initial forever #2 clk = ~clk;
endmodule

module assert_mod_1 (input wire clk, input wire reset, input wire err);
reg wait_for_reset_falling;
initial
begin
wait_for_reset_falling = 1'b1;
wait (reset === 0);
wait_for_reset_falling = 1'b0;
end
assert_1: assert property (@(posedge clk) disable iff (reset == 1'b0 ||
 wait_for_reset_falling == 1'b1) (!err));
endmodule

module assert_mod_2 (input wire clk, input wire reset, input wire err);
assert_2 : assert property (@(posedge clk) disable iff (reset == 1'b0)
 (!err))
else
$display("Fail");
endmodule

```

The following is the output:

```

"test.sv", 28: top.inst1.assert_1: started at 10s failed at 10s
Offending '(!err)'
"test.sv", 28: top.inst2.assert_1: started at 10s failed at 10s
Offending '(!err)'
"test.sv", 28: top.inst3.assert_1: started at 10s failed at 10s
Offending '(!err)'
"test.sv", 28: top.inst4.assert_1: started at 10s failed at 10s
Offending '(!err)'
"test.sv", 28: top.inst5.assert_1: started at 10s failed at 10s
Offending '(!err)'
"test.sv", 32: top.inst6.assert_2: started at 10s failed at 10s
Offending '(!err)'
Fail

===== Generating Assertion Summary =====

"top.inst1.assert_1", 3 attempts, 2 successes, 1 failures, 0 incompletes
"top.inst2.assert_1", 3 attempts, 2 successes, 1 failures, 0 incompletes
"top.inst3.assert_1", 3 attempts, 2 successes, 1 failures, 0 incompletes
"top.inst4.assert_1", 3 attempts, 2 successes, 1 failures, 0 incompletes
"top.inst5.assert_1", 3 attempts, 2 successes, 1 failures, 0 incompletes
"top.inst6.assert_2", 3 attempts, 2 successes, 1 failures, 0 incompletes
Summary: 6 assertions, 6 with attempts, 6 with failures

```

===== End Assertion Summary =====

Error-[SVART-NAFRLTS] Too many assertion failures.  
The SVA run-time detected that the number of assertion failures has reached  
the maximum allowed limit (3). Terminating simulation.  
Please verify that you are not using the options setting the limit too low

You can use the `-assert dis_summary_fix` runtime switch to revert to the previous behaviour:

```
% vcs -sverilog test.sv -assert enable_diag
% simv -assert global_finish_maxfail=3 -error=SVART-NAFRLTS -assert
summary -assert dis_summary_fix
```

The following is the output:

```
"test.sv", 28: top.inst1.assert_1: started at 10s failed at 10s
Offending '(!err)'

"test.sv", 28: top.inst2.assert_1: started at 10s failed at 10s
Offending '(!err)'

"test.sv", 28: top.inst3.assert_1: started at 10s failed at 10s
Offending '(!err)'

Error-[SVART-NAFRLTS] Too many assertion failures.

The SVA run-time detected that the number of assertion failures has reached
the maximum allowed limit (3). Terminating simulation.

Please verify that you are not using the options setting the limit too low
```

===== Generating Assertion Summary =====

```
"top.inst1.assert_1", 3 attempts, 2 successes, 1 failures, 0 incompletes
"top.inst2.assert_1", 3 attempts, 2 successes, 1 failures, 0 incompletes
"top.inst3.assert_1", 3 attempts, 2 successes, 1 failures, 0 incompletes
```

```
"top.inst4.assert_1", 3 attempts, 3 successes, 0 failures, 0 incompletes
"top.inst5.assert_1", 3 attempts, 3 successes, 0 failures, 0 incompletes
"top.inst6.assert_2", 3 attempts, 2 successes, 1 failures, 0 incompletes
Summary: 6 assertions, 6 with attempts, 4 with failures
=====
End Assertion Summary
=====
```

---

## SystemVerilog Assertions Limitations

This section describes the limitations that apply to SystemVerilog assertions.

---

### Debug Support for New Constructs

UCLI support for new assertions (LTL operators, checker block) supported under `-assert svaext` is not fully qualified.

---

### Note on Cross Features

Some of the new features in assertions (LTL operators, checker block) under `-assert svaext` have known limitations with cross-feature support, such as Debug and Coverage. Check with the Synopsys support for any unexpected results with cross-feature behavior for these new constructs.

Some known issues:

- `-cm property_path` is not available for the new constructs.
- New sequence operators when used as sampling event for covergroups may not function well.

# 23

## Using Property Specification Language

---

VCS supports the Simple Subset of the IEEE 1850 Property Specification Language (PSL) standard. Refer to Section 4.4.4 of the *IEEE 1850 PSL LRM* for the subset definition.

You can use PSL in Verilog, VHDL, or mixed designs along with SystemVerilog Assertions (SVA), SVA options, SVA system tasks, and OpenVera (OV) classes.

This chapter discusses the following topics:

- [Including PSL in the Design](#)
  - [Use Model](#)
  - [PSL Assertions Inside VHDL Block Statements in Vunit](#)
  - [PSL Macro Support in VHDL](#)
  - [Using SVA Options, SVA System Tasks, and OV Classes](#)
  - [Limitations](#)
- 

### Including PSL in the Design

You can include PSL in your design in any of the following ways:

- Inlining the PSL using the `//psl` or `/*psl */` pragmas in Verilog and SystemVerilog, and `--psl` pragma in VHDL.
  - Specifying the PSL in an external file using a verification unit (`vunit`).
- 

### Examples

The following examples shows how to inline PSL in Verilog using the `//psl` and `/*psl */` pragmas, and in VHDL using the `--psl` pragma.

#### In Verilog

```
module mod;
 ...
 // psl a1: assert always {r1; r2; r3} @ (posedge clk);
```

```
/* psl
 A2: assert always {a;b} @ (posedge clk);
 ...
 */
endmodule
```

### In VHDL

```
LIBRARY ieee;
USE ieee.std_logic_1164.all;

entity vh_ent is
...
end vh_ent;

architecture arch_vh_ent of vh_ent is
...
-- psl default clock is (clk'event and clk = '1');
-- psl sequence seq1 is {in1;[*2];test_sig};

-- psl property p1 is
-- (never seq1);

-- psl A1: assert p1 report " : Assertion failed P1";
end arch_vh_ent;
```

The following examples show how to use `vunit` to include PSL in the design.

### In Verilog

```
vunit vunit1 (verilog_mod)
{
 a1: assert always {r1; r2; r3} @ (posedge clk);
}
```

### In VHDL

```
vunit test(vh_entity)
{
 default clock is (clk'event and clk = '1');

 property foo is
 always ({ a = '0'} |=> {(b = prev(b) and c = prev(c))});
 assume foo;
}
```

## Use Model

If you inline the PSL code, you must compile/analyze it with the `-psl` option.

If you use `vunit`, you must compile/analyze the file that contains the `vunit` with the `-pslfile` option. You do not need to use this option if the file has the `.psl` extension.

### Compilation

```
% vcs -psl [vcs_options] Verilog_files
```

### Simulation

```
% simv
```

---

## Examples

To simulate the PSL code that is inlined in a mixed design (`test.v` and `dut.vhd`), execute the following commands:

```
% vlogan -psl test.v
% vhdlan -psl dut.vhd
% vcs -psl top
% simv
```

To simulate both the PSL code inlined in a VHDL file (`test.vhd`), and the `vunit` specified in an external file (`checker.psl` or `checker.txt`), execute the following commands:

```
% vhdlan -psl test.vhd checker.psl
% vcs -psl top
% simv
```

or

```
% vhdlan -psl test.vhd -pslfile checker.txt
% vcs -psl top
% simv
```

### Analysis

```
% vlogan -psl [vlogan_options] Verilog_files
% vhdlan -psl [vhdlan_options] VHDL_files
```

### Note:

Specify the VHDL bottommost entity first, then move up in order.

### Elaboration

```
% vcs -psl top_cfg/entity/config
```

### Note:

Ensure that you specify the `-psl` option while elaborating the design.

## Simulation

```
% simv
```

---

## Examples

To simulate the PSL code that is inlined in a mixed design (`test.v` and `dut.vhd`), execute the following commands:

```
% vlogan -psl test.v
% vhdlan -psl dut.vhd
% vcs -psl top
% simv
```

To simulate both the PSL code inlined in a VHDL file (`test.vhd`), and the `vunit` specified in an external file (`checker.psl` or `checker.txt`), execute the following commands:

```
% vhdlan -psl test.vhd checker.psl
% vcs -psl top
% simv
```

or

```
% vhdlan -psl test.vhd -pslfile checker.txt
% vcs -psl top
% simv
```

---

## PSL Assertions Inside VHDL Block Statements in Vunit

This section describes support for Property Specification Language (PSL) assertions inside VHDL block statements in a `vunit`.

This section contains the following topics:

- [Introduction](#)
  - [Use Model](#)
  - [Limitations](#)
- 

## Introduction

VCS supports the usage of PSL assertions inside VHDL block statements in `vunit`. This feature extends the capability of VHDL block statements in a `vunit` by allowing PSL assertions inside VHDL block statements.

## Example

```

library IEEE;
use IEEE.STD_LOGIC_1164.all;
use ieee.numeric_std.all
USE ieee.std_logic_unsigned.all

vunit ins2 (half_dsp_block(rtl)) {

 default clock is clock'event and clock = '0';

 signal sig1: integer;
 sequence S1 is {a0(3 downto 0) = "1000" [->1]} @ (clock'event and
clock = '0');

 blk1: block is
 begin
 property P1 is always {b2 ; S1} @ (clock'event and clock =
'1');

 A1: assert always P1 @ (clock'event and clock = '1');
 sig1 <= conv_integer(a0);

 end block blk1;

 A2: assert always {sig1 = 8};
}

```

## Use Model

Use the `-assert psl_in_block` analysis option to enable the usage of PSL assertions inside VHDL block statements in vunit, as shown in the following command:

```
$> vhdlan test.vhd -psl -assert psl_in_block test.psl
```

The following PSL constructs are allowed in the VHDL block statements. You must specify these constructs in `block_statement_part`, and not in `block_declarative_part`.

- Property and sequence declarations
- Assert statement
- Assume statement
- Restrict statement
- Cover statement

## Limitations

The following are the limitations of using PSL assertions inside VHDL block statements in a vunit:

- This enhancement does not support the following non-PSL VHDL constructs, which are currently not allowed inside a block statement in a vunit:
  - Guard expressions
  - Port declarations and port maps
- The following PSL constructs are not supported in the VHDL block statements:
  - Generic declarations and generic maps
  - Default clocks
  - Vunit inheritance

## PSL Macro Support in VHDL

VCS supports the `%if` and `%for` PSL macros in VHDL as described in IEEE-1850-2010 PSL. You can use these macros to conditionally or iteratively generate PSL statements. This removes the need for to rewrite the entire PSL code (a time-consuming task).

The following sections explain how to use these constructs:

- [Using the %for Construct](#)
- [Using the %if Construct](#)
- [Using Expressions with %if and %for Constructs](#)
- [PSL Macro Support Limitations](#)

## Using the %for Construct

The `%for` construct replicates a piece of code.

The syntax for `%for` iteration range is:

```
%for /var/ in /expr1/ ... /expr2/ do
...
%end
```

The syntax for %for iteration list is:

```
%for /var/ in { /item/ [, /item/] * } do
...
%end
```

Following are the arguments:

- var — Variable name
- expr — Expression on which macro substitution is performed. This argument should be a numeric decimal value.
- item — Value to be substituted for instances of the variable name on each iteration of the %for macro. This value can only contain alphanumeric characters ('a' to 'z', 'A' to 'Z', and '0' to '9') and underscores.

If an item contains only digits, it is treated as a number during expression evaluation.

Bareword macro substitution is not done on items in the %for iteration list. However, %{} style macro substitutions are done on these items. This provides the flexibility to control the strings in the list. For example, consider the following code:

```
%for xx in { aa, bb, cc } do
%for yy in { xx, %{xx}, zz } do
...
```

The loop iterator yy takes the following values:

- xx, aa, zz in the first iteration of loop xx
- xx, bb, zz in the second iteration of loop xx
- xx, cc, zz in the third iteration of loop xx

When a macro substitution of a list item iterator occurs, it is only done on one level of substitution. That is, if the list item value itself is a name that matches the name of a macro iterator, then the value of that iterator is not substituted. The value substituted is the string defined in the item list. Consider the code in [Example 201](#).

#### *Example 201 Macro Substitution*

```
%for xx in 1...2 do
 %for yy in { xx, zz } do
 Lbl_%{yy}_%{xx} : assert ...
 %end
%end
```

In [Example 201](#), when the `yy` iterator value is substituted, the resulting value is `xx`, and not the current value of the `xx` iterator (1 or 2):

```
Lbl_xx_1: assert ...
Lbl_zz_1: assert ...
Lbl_xx_2: assert ...
Lbl_zz_2: assert ...
```

The `%{ } macro substitution` within a quote ("") delimited string is supported. Bareword string substitution is not allowed within a quoted string. For example, the following code:

```
%for xx in 1 ... 2 do
 report "xx = %{xx}";
%end
```

Expands to:

```
Report "xx = 1";
Report "xx = 2";
```

You can use the `%` character as a string delimiter. No macro substitution is performed within `%` delimited strings.

For macro expansion, any occurrence of macro keywords that include the `%` character (`%for`, `%if`, `%then`, `%else`, `%end`, and the `%{...}` substitution macro) takes priority over string initiation. For example:

```
report %xx = %{xx}%;
```

The above example results in a syntax error at `{` and an unterminated string (starting delimiter is the last `%` on the line).

## Using the `%if` Construct

The `expr` argument of the `%if` macro must evaluate to an integer.

- If the expression resolves to an integer other than zero, then the expression is true and the `%then` clause is processed.
- If the expression resolves to zero, then the expression is false and the `%else` statement, if present, is processed.

The syntax for `%if` is as follows:

```
%if /expr/ %then
...
%end
```

or

```
%if /expr/ %then
...
%else
%end
```

## Using Expressions with %if and %for Constructs

You can use the following in the expressions with `%if` and `%for` constructs:

- Decimal literals
- Alphanumeric strings
- Operators:  
`=, -, *, /, %, =, !=, <, <=, >, >=`
- Parentheses:  
`('(' and ')')`

All arithmetic operations are integral.

VCS generates an error message if:

- An operand of an arithmetic operation is non-numeric
- Either operand evaluates to a non-alphanumeric string

Comparison operations are integral if both operands are integral. If either operand is alphanumeric (after substitution), then lexical comparison is performed.

For example, consider the following expression:

```
%if (foo(1) == 0)
```

This expression is an error, because the left operand of the equality does not evaluate to an alphanumeric string.

## PSL Macro Support Limitations

- The `%for` and `%if` macros are not supported in inline PSL pragmas
- The `%{ }%` macro substitution cannot span lines. However, `%for` and `%if` header constructs can span lines
- The `%for` and `%if` macros are not supported within encrypted blocks. Macro text can contain encrypted blocks
- VHDL-style extended identifiers are not supported as `%for` macro iterator names

- The `%{ } style replacement macros within other %{ } style replacement macros are not allowed`
- Example: `%{ ii + %{jj + 1} }`
- Octal and hexadecimal literal numeric values are not supported
  - A nested `%for` iterator name cannot use the same name as an outer `%for` macro's iterator name. For example:
- ```
%for xx in 1...2 do
    %for xx in 3...4 do
    ...

```
- C preprocessor directives (for example, `#define`, `#ifdef`, `#else`, `#include`, and `#undef`) are not supported.

Using SVA Options, SVA System Tasks, and OV Classes

VCS enables you to use all assertion options with SVA, PSL, and OVA. For example, to enable PSL coverage and debug assertions while elaborating/compiling the PSL code, execute the following commands:

For two-step flow:

```
% vcs -psl -cm assert -debug_access -assert enable_diag test.v
% simv -cm assert -assert success
```

For three-step flow:

```
% vhdlan -psl dut.vhd checkers.psl
% vhdlan test.vhd
% vcs top -psl -cm assert -debug_access -assert enable_diag
% simv -cm assert -assert success
```

For information on all assertion options, see Appendix - Compilation/Elaboration Options.

You can control PSL assertions in any of the following ways:

- Using the `$asserton`, `$assertoff`, or `$assertkill` SVA system tasks.
- Using NTB-OpenVera assert classes.

Note that VCS treats the `assume` PSL directive as the `assert` PSL directive.

Limitations

The VCS implementation of PSL has the following limitations:

- VCS does not support binding `vunit` to an instance of a module or entity
- VCS does not support generic declarations and generic maps in VHDL block statements in a `vunit`
- VCS does not support the following data types in your PSL code -- `shortreal`, `real`, `realtime`, associative arrays, and dynamic arrays
- VCS does not support the `union` operator and union expressions in your PSL code
- Clock expressions have the following limitations:
 - You must not include the `rose()` and `fell()` built-in functions
 - You must not include endpoint instances
- Endpoint declarations must have a clocked SERE with either a clock expression or default clock declaration
- VCS does not support the `%for` and `%if` macros
- VCS supports only the `always` and `never` FL invariance operators in top-level properties. Ensure that you do not instantiate top-level properties in other properties
- VCS supports all LTL operators, except `sync_abort` and `async_abort`. You can apply the abort operator only to the top property
- VCS does not support the `assume_guarantee`, `restrict`, and `restrict_guarantee` PSL directives

24

Using SystemC

The VCS SystemC Co-simulation Interface enables VCS and the SystemC modeling environment to work together when simulating a system coded in the Verilog, VHDL, and SystemC languages.

With this interface, you can use the most appropriate modeling language for each part of the system and verify the correctness of the design. For example, the VCS SystemC Co-simulation Interface allows you to:

- Use a SystemC module as a reference model for the VHDL or Verilog RTL design under test in your testbench
- Verify a Verilog or VHDL netlist after synthesis with the original SystemC testbench
- Write test benches in SystemC to check the correctness of Verilog and VHDL designs
- Import legacy VHDL or Verilog IP into a SystemC description
- Import third-party VHDL or Verilog IP into a SystemC description
- Export SystemC IP into a Verilog or VHDL environment when only a few of the design blocks are implemented in SystemC
- Use SystemC to provide stimulus to your design

The VCS/SystemC Co-simulation Interface creates the necessary infrastructure to co-simulate SystemC models with Verilog or VHDL models. The infrastructure consists of the required build files and any generated wrapper or stimulus code. VCS writes these files in subdirectories in the `./csrc` directory. To use the interface, it is not required to update or write to these files.

During co-simulation, the VCS/SystemC Co-simulation Interface is responsible for:

- Synchronizing the SystemC kernel and VCS
- Exchanging data between the two environments

Note:

- The unified profiler can report CPU time profile information about the SystemC part or parts of a design. For more information, see chapter "*The Unified Simulation Profiler*" in the *VCS User Guide*.
- Examples of Verilog/VHDL instantiated in SystemC and SystemC instantiated in Verilog/VHDL are included in the `$VCS_HOME/doc/examples/systemc` directory.

For more information about SystemC co-simulation interface and how you can use SystemC with VCS for your design, see *VCS SystemC User Guide* available in the SolvNetPlus online support site.

25

Dynamic Test Loading

This document describes the Dynamic Test Loading (DTL) feature in the following sections:

- [Introduction](#)
 - [Advantages of DTL](#)
 - [How DTL Works](#)
 - [Use Model](#)
 - [Dynamic Test Loading Modes](#)
 - [Guidelines for Using DTL](#)
 - [Debug Support for Dynamic Test Loading](#)
 - [Limitations](#)
-

Introduction

As system-on-chip (SoC) designs are becoming more complex, verifying them has become a necessary part for the product to succeed. The resources available and the time-to-market requirements for development impact verification. SoC verification includes implementing a verification environment, creating a testbench, performing logic simulations, analyzing the results to detect and isolate problems, and so on. Therefore, verifying the high-end SoC can consume significant time of the total development effort from initial concept to final implementation.

DTL is an approach to load and run any test dynamically at any time during simulation. The test loaded at runtime can include different UVM tests or sequences and the advantages of DTL are listed in the below section.

Advantages of DTL

VCS Dynamic Test Loading technology improves the overall verification efficiency and avoids repeated steps during compilation and simulation and helps to reduce TAT during verification.

Dynamic Test Loading helps to achieve the following:

- Faster compile and simulation TAT.
- Improved regression throughput.
- Improved verification productivity.
- DTL can be used in debug, development, or regression phases of verification. For more information, see the [Dynamic Test Loading Modes](#) section.

How DTL Works

- UVM tests must be encapsulated inside SystemVerilog packages. The UVM base test derived from the UVM test is called static test. The static tests are encapsulated inside the static package. Subsequent tests derived from the base test of the static package are called dynamic tests. Dynamic tests are encapsulated inside the dynamic packages. You can add more than one test inside the static or dynamic packages.
- DUT and the test environment are compiled only once and they are not recompiled later, therefore ensuring faster compilation TAT.
- If changes are made only to the dynamic package encapsulating the new or changing tests, simulator executable (`simv`) is not regenerated.
- In general, many tests can have common starting sequences in a verification setup. The common part is run only for the first test and subsequent tests reuse the initialization or common operations among tests. This reduces the runtime for the dynamic tests.
- The saved snapshot generated from the static test package does not have to be regenerated if changes are made only to the dynamic package encapsulating the new tests.
- All the dynamic tests can be compiled and simulated in parallel.

Use Model

Following are the steps to use DTL:

1. Encapsulate all test classes derived from UVM classes in SystemVerilog packages.
2. Compile the test packages using DTL Two-Step Flow:

- Compile the static package using the `-enable_dynamic_tb` compile-time option and other partition compile options.

```
% vcs -partcomp DUT base_test_pkg -dir=IP_BASE_TEST
      -enable_dynamic_tb
```

where,

`-partcomp`

Enables Partition Compile flow.

`-dir`

Specify the directory to record the partition database.

`-enable_dynamic_tb`

Enables DTL flow.

This creates a partition database in the `IP_BASE_TEST` directory along with the simulator executable.

- Enclose other tests and sequences derived from the base test classes in separate dynamic packages. Compile the dynamic packages using the `-dynamic_tb` option.

```
% vcs -partcomp test1_pkg -dir= IP_TEST1 -sharedlib=IP_BASE_TEST
      -dynamic_tb
```

```
% vcs -partcomp test2_pkg -dir= IP_TEST2 -sharedlib=IP_BASE_TEST
      -dynamic_tb
```

where,

`-partcomp/-sharedlib/-dir`

Enables Partition Compile flow.

`-dynamic_tb`

Enables dynamic test compile for DTL.

When multiple top packages are provided with `-dynamic_tb` option, it shows the following error message. Only one top package is allowed.

```
Error-[DTL-MULTITOP-ERR] Dynamic Test Load Error
Multiple top packages are not supported with -dynamic_tb. Top
packages found: test2_pkg test1_pkg
```

Note:

The names of the top packages found in the above error message varies based on the testcase.

Note:

This step reuses the partition information as per the `-sharedlib` option and `simv` executable is not generated. You can observe a very fast compilation for dynamic tests.

3. Simulate base test. Save the Simulation state after the common initial sequence (until `reset_phase` in the following example) of the test is executed.

```
% simv +UVM_TEST_NAME=base_test -ucli -I save.cmds
```

where,

`save.cmds`

UCLI command file.

```
stop -in uvm_component::post_reset_phase -once -continue -command {run
0; save saveddesign; ; run}
```

Note:

- You can use VCS save restore feature using UCLI commands or from the `$save`/`$restore` calls from Verilog code.
- When the simulation state is restored, the setup and restore portions are executed in one continuous simulation.
- In case of only one UVM run phase in the test and there are no sub-phases:

The test is saved in the middle of the UVM `run_phase` of `base_test`, then at restore it resumes execution and completes the `run_phase` of `base_test`. Once the current phase execution is completed, UVM switches to the next phase of `dyn_test`.

- For DTL `$save` and `$restore`, the UVM run phase must have at least two sub-phases so that you can save the simulation in one sub-phase, and upon restoration, the subsequent sub-phase can be triggered for the dynamic test. Hence the test must have at least two sub-phases to successfully change to the `dyn_test` `run_phase` during the restore operation.

4. Run other tests by dynamically loading the test packages. Restore simulation state after the common part is over and save on simulation TAT.

```
% simv +UVM_TEST_NAME=test1 -ucli -I restore.cmds
+dtl_add_pkg=IP_TEST1:test1_pkg
```

where,

```
+dtl_add_pkg
```

Name of the test to be loaded dynamically at Runtime.

The package name information must be in the following format:

```
<dir_name>:<pkg_name>
```

If there are multiple logical libraries, then the package name information must be in the following format:

```
<dir_name>:<lib_name>.<pkg_name>
restore.cmds
```

UCLI command file

[Table 49](#) describes the commands in the UCLI file.

Table 49 UCLI File Commands

Commands	Description
restore saved design	Restore simulation after common part of base_test is executed.
call \\$dtl_load	Load package specified on simv using the +dtl_add_pkg.
call top.refresh	Refresh UVM test after \$dtl_load option is called. This is needed only when VCS save restore is used.
Run	Run rest of test <N>.

The following code snippet is used for refresh task:

```
task refresh ;
  string test_name;
  uvm_root root = uvm_root::get();
  if($test$plusargs("UVM_TESTNAME")) begin
    if($value$plusargs("UVM_TESTNAME=%s",test_name)) begin
      `uvm_info("restore",$sformatf("UVM_TESTNAME name passed from
the command line=%0s",test_name),UVM_NONE);
    end
  end
```

```
$display("[SNPS_DEBUG_RESTORE] Refreshed test_name to %0s at
    time=%0t",test_name,$time);
$display("[SNPS_DEBUG_RESTORE] Running test=%0s at
    time=%0t",test_name,$time);
root.refresh_test(test_name);
endtask
```

Note:

In a typical UVM verification environment, dynamic test contents can perform the following:

- Define and launch new sequences.
- Extend the post `run()` built-in phase methods: `extract()`, `check()`, `finalize()`.

Dynamic Test Loading Modes

DTL is suitable for all kinds of verification requirements and can be used in debug, development, or regression phases of verification.

Regression Model

In this model:

- All tests are available and known.
- Tests do not change during the execution.
- Tests having common initialization pattern, link training and configuration phases.

How DTL Works in Regression Model

1. Encapsulate all tests in one package.
2. Generates `simv` executable with all the regression tests.
3. Run the first base test and save the simulation run after the common routine like initialization or `reset_phase()`.
4. Dynamically loads the new test and continues the simulation from the saved state. It improves simulation TAT for regression.

Development Model

In this model:

- All tests are available, but they are not frozen and changing.
- Tests can be modified during development or debugging failures.
- New tests are added with the new functionality.

How DTL Works in Development Model

For the incremental change to the existing test:

1. Static tests are compiled separately inside the static test package to generate `simv`.
2. Dynamic tests are encapsulated into independent, self-contained packages and compiled without regenerating `simv`.
3. Saves the simulation run with first test after the common pattern. For example, `reset_phase()`.
4. Make incremental changes in the test and compile the modified dynamic test package only. You do not have to regenerate the `simv`. It saves compile TAT.
5. Restore simulation from saved state. Dynamically load the required test and continue the simulation from the saved simulation state. You do not need to update the saved simulation state for the new test. It saves simulation TAT.

For the new test addition:

1. Compile static tests separately inside static test package to generate `simv`.
2. During simulation, save the simulation run with first test after the common pattern. For example, the `reset_phase()`.
3. A new test can be written and compiled now to be used post-restore. New tests can be encapsulated into independent, self-contained packages and compiled without regenerating the `simv`. It saves compile TAT.
4. Once the restore is done, dynamically choose one of the tests and complete the simulation from the saved simulation state. It is not required to regenerate the saved simulation state again for the new test as tests are being loaded and restored dynamically. It saves simulation TAT.

Guidelines for Using DTL

1. Test changes:
 - If the source code of a dynamic test is changed, only that test package must be recompiled.
 - If the source code of the static test is changed, the static test package must be recompiled and simv must be regenerated. If the new test is dependent on the static test, then the new test must be recompiled.
2. Dynamically loaded test packages must configure the UVM tests in the main phase and not in the build phase. This is required as the build phase is not executed for the dynamically loaded test. The dynamic test is restored from the saved state of the base test.

 For example, if threads are forked from the fork-join process invocation in the build phase of the base test, and the save point is created before the completion of the process, during the dynamic loading of a new test, these threads from the base test do not exist, and a new test starts from the main phase.
3. Compile time options must be the same for static and dynamic test compilation.
4. Runtime options can be different for dynamic tests if these arguments are executed post restore.
5. Do not use multiple tops at `-dynamic_tb` steps.
6. Only SystemVerilog package files should be passed to `-dynamic_tb` steps.
7. Dynamic test compilation can have only packages, not modules.
8. Usage of cross-module references (XMR) in the base test or dynamic test package:
 - As per the SystemVerilog LRM, packages cannot have XMR access outside of the package.
 - Dynamic test cannot have XMRs to the base test package.
 - To support XMR usage inside dynamic test packages, include XMRs as part of the top-level testbench or module to avoid XMR errors.
 - XMRs present in `$unit` scope of the base test or dynamic test packages should be moved to testbench or module to avoid XMR errors.
9. Dynamic loading of multiple tests is not supported in one simulation. You can load only one test.
10. To use Dynamic Performance Optimizer (DPO) along with DTL, disable the `save` `restore` during the DPO learn phase and enable it during the DPO apply phase.

Debug Support for Dynamic Test Loading

You can debug the dynamically loaded tests using *Verdi Interactive Debug*. When Verdi is invoked using DTL, you can use Verdi features to debug and trace new tests in post-process or interactive mode.

Use the `-ucli2Proc` debug option with `$save` and `$restore` UCLI commands to allow interactive debug for only a specific test.

Note:

Debug operation is supported on an object in the existing tests and equivalent object in the dynamically loaded test. The new dynamic test must also be compiled with the same debug switches as used with the existing design and tests. The new packages are accessible after they are loaded by the UCLI restore. Therefore, the debug operations performed in the original saved simulation do not automatically apply to the new packages. For example, if you enable full dumping in the original simulation, the new packages are not automatically dumped.

Limitations

- DTL supports only VCS UUM flow. It does not support VCS two-step flow.
- DTL dynamic package does not support the code coverage metrics.
- DTL does not support SVA inside the DTL dynamic package.
- DTL ignores the Low Power options that are passed at the dynamic test compilation stage.
- DTL ignores the X-Propagation options that are enabled at the dynamic test compilation stage.
- DTL does not support the following debug options:
 - `-debug_acc+reverse`
 - `-kdb` : instead use `-kdb=common_elab`
- DTL does not support multiple top packages with `-dynamic_tb` option.

For more details, see the Partition Compile Limitations section and Saving and Restarting the Simulation section in *VCS User Guide*.

26

C Language Interface

Generally, C and C++ code is added with both Verilog and VHDL. There are many different mechanisms to add C and C++ code and how you add these code designs depends on your objective as well as the performance and restrictions of each mechanism. VCS supports the following ways to use C and C++ in your design:

- [Using PLI](#)
- [Using VPI Routines](#)
- [Using VHPI Routines](#)

VHPI enables you to use foreign architecture-based models written in C language in the VCS VHDL using DirectC.

- [Using DirectC](#)
- [Using SystemC](#)
- Using SystemVerilog DPI routines - See the SystemVerilog LRM.

For the description of PLI 1.0, PLI 2.0, and VHPI routines, see the *C Language Interface Reference Manual*.

Note:

PLI1.0 refers to TF and ACC routines, and PLI2.0 refers to VPI.

- [Using DPI Open Array](#)

Using PLI

PLI is the programming language interface (PLI) between C/C++ functions and VCS. It helps link applications containing C/C++ functions with VCS, so that they execute concurrently. The C/C++ functions in the application use the PLI to read and write delay and simulation values in the VCS executable. Later during simulation VCS can call these functions.

VCS supports PLI 1.0 and PLI 2.0 routines for the PLI. Therefore, you can use VPI, ACC, or TF routines to write the PLI application. See Appendix , [PLI Access Routines](#).

This chapter covers the following topics:

- [Writing a PLI Application](#)
 - [Functions in a PLI Application](#)
 - [Header Files for PLI Applications](#)
 - [PLI Table File](#)
 - [Enabling ACC Capabilities](#)
-

Writing a PLI Application

When writing a PLI application, you need to perform the following tasks:

1. Write the C/C++ functions of the application calling the VPI, ACC, or TF routines to access data inside VCS.
2. Associate user-defined system tasks and system functions with the C/C++ functions in your application. VCS calls these functions when it compiles or executes these system tasks or system functions in the Verilog source code. In VCS, associate the user-defined system tasks and system functions with the C/C++ functions in your application using a PLI table file (see [PLI Table File](#)). In this file, you can also limit the scope and operations of the ACC routines for faster performance.
3. Enter the user-defined system tasks and functions in the Verilog source code.
4. Compile/Analyze, elaborate, and simulate your design, specifying the table file and including the C/C++ source files (or compiled object files or libraries) so that the application is linked with VCS in the `simv` executable. If you include object files, use the `-cc` and `-ld` options to specify the compiler and linker that generated them. Linker errors occur if you include a C/C++ function in the PLI table file, however, omit the source code for this function at compile time.

To use the debugging features, perform the following tasks:

1. Write a PLI table file, limiting the scope and operations of the ACC routines used by the debugging features.
2. Compile/Analyze, elaborate, and simulate your design, specifying the table file.

These procedures are not mutually exclusive. It is possible that you have a PLI application that you write and use during the debugging phase of your design. If so, you can write a PLI table file that both:

- Associates user-defined system tasks or system functions with the functions in your application and limits the scope and operations called by your functions for faster performance.
- Limits scope and operations of the functions called by the debugging features in VCS.

Functions in a PLI Application

When you write a PLI application, you typically write a number of functions. The following are the PLI functions that VCS expects with a user-defined system task or system function:

- The function that VCS calls when it executes the user-defined system task. Other functions are not necessary, however, this call function must be present. It is not unusual for there to be more than one call function. You will need a separate user-defined system task for each call function. If the function returns a value, then you must write a user-defined system function for it instead of a user-defined system task.
- The function that VCS calls during compilation to check if the user-defined system task has the correct syntax. You can omit this check function.
- The function that VCS calls for miscellaneous reasons, such as the execution of \$stop, \$finish, or other reasons, such as a value change. When VCS calls this function, it passes a reason argument to it that explains why VCS is calling it. You can omit this miscellaneous function.

These are the functions you instruct VCS about in the PLI table file; apart from these PLI applications can have several more functions that are called by other functions.

Note:

You do not specify a function to determine the return value size of a user-defined system function; instead you specify the size directly in the PLI table file.

Header Files for PLI Applications

For PLI applications, you need to include one or more of the following header files:

`vpi_user.h`

For PLI Applications whose functions call IEEE Standard VPI routines as documented in the *IEEE Verilog Language Reference Manual*.

`acc_user.h`

For PLI Applications whose functions call IEEE Standard ACC routines as documented in the *IEEE Verilog Language Reference Manual*.

`vcsuser.h`

For PLI applications whose functions call IEEE Standard TF routines as documented in the *IEEE Verilog Language Reference Manual*.

`vcs_acc_user.h`

For PLI applications whose functions call the special ACC routines implemented exclusively for VCS.

You can find these header files in the `$VCS_HOME/your_platform/lib` directory.

PLI Table File

The PLI table file (also referred to as the `pli.tab` file) is used to:

- Associate user-defined system tasks and system functions with functions in a PLI application. This enables VCS to call these functions when it compiles or executes the system task or function.
- Limit the scope and operation of the PLI functions called by the debugging features. See [Specifying Access Capabilities for PLI Functions](#) and [Specifying Access Capabilities for VCS Debugging Features](#).

Syntax

The following is the syntax of the PLI table file:

`$name PLI_specifications [access_capabilities]`

Where,

`$name`

Specify the name of the user-defined system task or function.

`PLI_specifications`

Specify one or more specifications, such as the name of the C function (mandatory), size of the return value (mandatory only for user-defined system functions), and so on. For a complete list of PLI specifications, see [PLI Specifications](#).

`access_capabilities`

Specify the access capabilities of the functions defined in the PLI application. Use this to control the PLI 1.0 or PLI 2.0 functions' ability to access the design hierarchy. For more information, see [Access Capabilities](#).

Synopsys recommends you to enable this feature while using PLIs to improve the runtime performance.

PLI Specifications

The PLI specifications are as follows:

`call=function`

Specifies the name of the function defined in the PLI application. This is mandatory.

`check=function`

Specifies the name of the check function.

`misc=function`

Specifies the name of the misc function.

`data=integer`

Specifies the value passed as the first argument to the call, check, and miscellaneous functions. The default value is 0.

Use this argument if you want more than one user-defined system task or function to use the same call, check, or misc function. In such a case, specify a different integer for each user-defined system task or function that uses the same call, check, or misc function.

`size=number`

Specifies the size of the returned value in bits. While this is mandatory for user-defined system functions, you can ignore or specify 0 for user-defined system tasks. For user-defined system functions, specify a decimal value for the number of bits. For example, `size=64`. If the user-defined system function returns a real value, specify `r`. For example, `size=r`

`args=number`

Specifies the number of arguments passed to the user-defined system task or function.

`minargs=number`

Specifies the minimum number of arguments.

`maxargs=number`

Specifies the maximum number of arguments.

`nocelldefinepli`

This option is functionally equivalent to the `+nocelldefinepli+1` compile option. This option disables debugging and dumping of FSDB/VPD for objects in modules defined under the `'celldefine` compiler directive.

`persistent`

Marks the specified function as being called in the PLI application, even if the corresponding system task or function is not called in the design. If the function is not found or defined in the PLI application, VCS exits with an undefined reference error message.

Example 1

```
$val_proc call=val_proc check=check_proc misc=misc_proc
```

In this line, VCS calls the function named `val_proc` when it executes the associated user-defined system task named `$val_proc`. It calls the `check_proc` function at compile time to see if the user-defined system task has the correct syntax, and calls the `misc_proc` function in special circumstances, such as interrupts.

Example 2

```
$set_true size=16 call=set_true
```

In this line, there is an associated user-defined system function that returns a 15-bit return value. VCS calls the function named `set_true` when it executes this system function.

Note:

Do not enter blank space inside a PLI specification. The following example of PLI specifications does not work:

```
$set_true size = 16 call = set_true
```

Access Capabilities

You can specify access capabilities in a PLI table file for the following reasons:

- PLI functions associated with your user-defined system task or system function. To do this, specify the access capabilities on a line in a PLI table file after the name of the user-defined system task or system function and its PLI specifications. For more details, see [Specifying Access Capabilities for PLI Functions](#).
- For the debugging features VCS can use. To do this, specify access capabilities alone on a line in a PLI table file, without an associated user-defined system task or system function. For more details, see [Specifying Access Capabilities for VCS Debugging Features](#).

In many ways, specifying access capabilities for your PLI functions, and specifying them for VCS debugging features is the same. However, the capabilities that you enable, and the parts of the design to which you can apply them are different.

Specifying Access Capabilities for PLI Functions

The format for specifying access capabilities is as follows:

```
acc=|+=|-=| :=capabilities: module_names|
interface_names|sig_name|inst_name [+]|%CELL|%TASK|*
```

Where,

acc

Keyword that begins a line for specifying access capabilities.

= | += | -= | :=

Operators for adding, removing, or changing access capabilities. The operators in this syntax are as follows:

=

A shorthand for +=.

+=

Specifies adding the access capabilities to the parts of the design that follow, as specified by module name, interface name, %CELL, %TASK, or * wildcard character.

-=

Specifies removing the access capabilities from the parts of the design that follow, as specified by module name, interface name, %CELL, %TASK, or * wildcard character.

:=

Specifies changing the access capabilities of the parts of the design that follow, as specified by module name, interface name, %CELL, %TASK, or * wildcard character, to only those in the list of capabilities on this specification. A specification with this operator can change the capabilities specified in a previous specification.

capabilities

Comma-separated list of access capabilities. The capabilities that you can specify for the functions in your PLI specifications are as follows:

r or read

Reads the values of nets and registers in your design.

rw or read_write

Both reads from and writes to the values of registers or variables (but not nets) in your design.

`wn`

Enables writing values to nets.

`cbk or callback`

To be called when named objects (nets registers, ports) change value.

`cbka or callback_all`

To be called when named and unnamed objects (such as primitive terminals) change value.

`frc or force`

Enables force values on nets and registers.

`dr`

Enables driver debugging capability.

`frcr`

Enables force values on registers.

`frcn`

Enables force values on nets.

`prx or pulserx_backannotation`

Sets pulse error and pulse rejection percentages for module path delays.

`s or static_info`

Enables access to static information, such as instance or signal names and connectivity information. Signal values are not static information.

`tchk or timing_check_backannotation`

Back-annotates timing check delay values.

`gate or gate_backannotation`

Back-annotates delay values on gates.

`mp or module_path_backannotation`

Back-annotates module path delays.

`mip or module_input_port_backannotation`

Back-annotates delays on module input ports.

`mipb or module_input_port_bit_backannotation`

Back-annotates delays on individual bits of module input ports.

`module_names`

Comma-separated list of module identifiers.

Specifying modules enables, disables, or changes (depending on the operator) the ability of the PLI function to use the access capability in all instances of the specified module.

`interface_names`

Comma-separated list of interface identifiers or names.

Specifying interfaces enables, disables, or changes (depending on the operator) the ability of the PLI function to use the access capability in all instances of the specified interface.

`sig_name`

Signal name on which PLI capabilities are applied. `sig_name` is the full hierarchical path.

For example,

```
<mod_name>.<sig_name> or <mod_name>.<inst_name>.<sig_name>
<interface_name>.<sig_name>
```

Signal names defined inside the generate block are not supported.

The `pli.tab` file does not support debug capabilities in the VHDL part of the design.

`inst_name`

Full hierarchical path of the instance name on which PLI capabilities are applied. You can use the “*” wildcard character at the end of an instance path.

`+`

Specifies adding, removing, or changing the access capabilities for not only the instances of the specified modules/interfaces, but also the instances hierarchically under the instances of the specified modules/interfaces.

The ‘[+]’ feature is not supported for the debug capabilities specified at the signal level (`sig_name`).

`%CELL`

Enables, disables, or changes (depending on the operator) the ability of the PLI function to use the access capability in all instances of module definitions compiled under the `'celldefine` compiler directive and all module definitions in Verilog library directories and library files (as specified with the `-y` and `-v` analysis options).

`%TASK`

Enables, disables, or changes (depending on the operator) the ability of the PLI function to use the access capability in all instances of module definitions that contain the user-defined system task or system function associated with the PLI function.

`%TFARGS`

Same as `%TASK`, but also enables debug capability on the arguments to the TF call.

`*`

Enables, disables, or changes (depending on the operator) the ability of the PLI function to use the access capability throughout the entire design. Using wildcard characters impact the performance of VCS.

Note:

Do not use blank spaces when specifying access capabilities.

The following examples are the PLI specification examples from the previous section with access capabilities added to them. The examples wrap to more than one line, however, when you edit your PLI table file, ensure that there are no line breaks in these lines.

Example 1

```
$val_proc call=val_proc check=check_proc misc=misc_proc acc+=rw,tchk:top,bot acc-=tchk:top
```

This example adds the access capabilities for reading and writing to nets and registers, and for back-annotating timing check delays, to these PLI functions, and enables them to do these things in all instances of modules `top` and `bot`. It then removes the access capability for back-annotating timing check delay values from these PLI functions in all instances of module `top`.

Example 2

```
$value_passer size=0 args=2 call=value_passer persistent acc+=rw:%TASK acc-=rw:%CELL
```

This example adds the access capability to read from and write to the values of nets and registers to these PLI functions. It enables them to do these things in all instances of modules declared in module definitions that contain the `$value_passer` user-defined system task. The example then removes the access capability to read from and write to the values of nets and registers from these PLI functions, in module definitions compiled under the `'celldefine` compiler directive and all module definitions in the Verilog library directories and library files.

Example 3

```
$set_true size=16 call=set_true acc+=rw:*
```

This example adds the access capability to read from and write to the values of nets and registers to the PLI functions. It enables them to do this throughout the entire design.

Example 4

```
acc=rw:top.I1,top.I2
```

This example adds the access capabilities on instances top.I1, top.I2.

Example 5

```
acc=rw:mod1.s1,mod1.s2
```

This example adds the access capabilities on signals s1, s2 of mod1.

Example 7

```
acc=rw:top.I1.s1,top.I1.s2,top.I2.s1,top.I2.s2
```

This example adds the access capabilities on signals s1, s2 of instances top.I1, top.I2.

Example 8

```
acc=rw:top.I1+
```

This example adds the access capabilities on instance top.I1 and its sub-instances.

Example 9

```
acc=rw:top.I1.*
```

This example adds read and write capabilities on instance top.I1 and all its sub-instances.

Usage Example

Consider the following files, test.v, driver.c, and pli.tab:

Example 202 Testcase test.v

```
module top;
integer myInt;
dut d1();
initial
begin
    force myInt = 20;
    $display(" After Force d1.g myInt at %0t is %0d %0d", $time,
    d1.g,myInt);
    #5;
    $myFrc(myInt);
    $display(" After PLI call d1.g myInt at %0t is %0d %0d", $time,
    d1.g,myInt);
    #5
    release myInt;
```

```
$myDrv(d1.g);
$display(" After PLI call d1.g myInt at %0t is %0d %0d", $time,
d1.g,myInt);
end
endmodule

module dut;
integer g;
endmodule
```

Example 203 driver.c

```
#include "acc_user.h"

myDrv()
{
    handle reg = acc_handle_object("top.d1.g");
    static s_setval_delay delay_s = { { 0, 1, 0, 0.0 }, accNoDelay };
    static s_setval_value value_s = {accIntVal};
    value_s.value.integer=0;
    acc_set_value(reg, &value_s, &delay_s);
}

myFrc()
{
    handle reg = acc_handle_object("top.myInt");
    static s_setval_delay delay_s = { { 0, 0, 0, 0.0 }, accForceFlag };
    static s_setval_value value_s = {accIntVal};
    value_s.value.integer =2;
    acc_set_value(reg, &value_s, &delay_s);

}
```

Example 204 pli.tab

```
$myDrv call=myDrv acc=rw:top.d1
$myFrc call=myFrc acc=frc:top.myInt
```

Compile the test.v code as follows:

```
% vcs driver.c -P pli.tab test.v -R -sverilog
```

Perform simulation:

```
% simv
```

VCS displays the following output:

```
After Force d1.g myInt at 0 is x 20
After PLI call d1.g myInt at 5 is x 2
After PLI call d1.g myInt at 10 is 0 2
```

Wildcard Support in PLI Tab File

The format for specifying wildcard characters in the PLI tab file is as follows:

```
acc+=capabilities:module_name1::sig_name1,module_name2::sig_name2,...
```

Where,

acc

Keyword that begins a line for specifying access capabilities.

+=

Specifies adding the access capabilities to the parts of the design that follow as specified by module name, interface name, %CELL, %TASK, or * wildcard character.

capabilities

Comma-separated list of access capabilities.

module_name

Comma-separated list of module identifiers.

sig_name

Signal name on which PLI capabilities are applied. sig_name is the full hierarchical path.

For example,

```
<mod_name>.<sig_name>
```

Note:

- Wildcard characters are supported only for module_name and sig_name.

Examples:

- acc+= frc:top.i1.i2::sig*

Adds force capability to the signals/variables in instance top.i1.i2 whose name starts with sig.

- acc+=frc:Mod::a*

Adds force capability to the signals/variables in module Mod whose name starts with a.

- acc+=frc:Mod::a*,Mod::*b

Adds force capability to the signals/variables in module Mod whose name either starts with a or ends with b.

- acc+=frc:Mod*::c

Adds force capability to the signals/variables with name `c` in modules with name starting with `Mod`.

- `acc+=frc:Mod*::a*`

Adds force capability to signals/variables with name starting with `a` and in the modules with name starting with `Mod`.

Note:

- For aggregate types (like struct and unions), only name of the struct node must be specified. This applies the debug capability to all the members of struct. You cannot apply debug capability to individual members of a struct.
- For signals inside a generate block or named block, specifying the name of generate/named block is not required. VCS automatically searches for signals/variables inside all the generate/named blocks of a module. For example, consider the following files:

Example 205 test.v

```
module mod;

    reg var1, abc1;

    generate for (genvar i=0; i<2; i=i+1)
begin: GEN
    reg var2, abc2;
end
endgenerate

initial
begin: INIT
    reg var3, abc3;
end

always @ (posedge clk)
begin: ALWAYS
    begin: BLOCK
        reg var4, abc4;
    end
end
endmodule
```

Example 206 PLI Tab File

```
acc+=frc:mod::var*
```

Applies force capability to all nodes with name starting with `var`:

```
var1, GEN[0].var2, GEN[1].var2, INIT.var3, ALWAYS.BLOCK.var4
```

Specifying Access Capabilities for VCS Debugging Features

The format for specifying access capabilities for VCS debugging features is as follows:

```
acc=|+=|-=|:=capabilities:module_names|interface_names[+]|%CELL|*
```

Here:

acc

Keyword that begins a line for specifying access capabilities.

=|+=|-=|:=

Operators for adding, removing, or changing access capabilities.

The removal of capability is done after all additions/changes are performed.

capabilities

Comma separated list of access capabilities.

module_names

Comma separated list of module identifiers. The specified access capabilities are added, removed, or changed for all instances of these modules.

interface_names

Comma separated list of interface identifiers. The specified access capabilities are added, removed, or changed for all instances of these interfaces.

+

Specifies adding, removing, or changing the access capabilities for not only the instances of the specified modules/interfaces, but also the instances hierarchically under the instances of the specified modules/interfaces.

%CELL

Specifies all modules compiled under the `'celldefine` compiler directive and all modules in the Verilog library directories and library files (as specified with the `-y` and `-v` options.)

*

Specifies that debug capability is applied to all modules in the design.

The access capabilities and the interactive commands are as follows:

ACC Capability	What it enables your PLI functions to do
r or read	For specifying “reads” in your design, it enables commands for performing the following:

ACC Capability	What it enables your PLI functions to do
	<p>Creating an alias for another UCLI command (<code>alias</code>)</p> <p>Displaying UCLI help</p> <p>Specifying the radix of displayed simulation values (<code>oformat</code>)</p> <p>Displaying simulation values</p> <p>Descending and ascending the module hierarchy</p> <p>Displaying the set breakpoints on signals</p> <p>Displaying the port names of the current location, and the current module instance or scope, in the module hierarchy</p> <p>Displaying the names of instances in the current module instance or scope</p> <p>Displaying the nets and registers in the current scope</p> <p>Moving up the module hierarchy</p> <p>Deleting an alias for another UCLI command</p> <p>Ending the simulation</p>
<code>rw</code> or <code>read_write</code>	Both reads from and writes to the values of registers or variables (but not nets) in your design.
<code>cbk</code> or <code>callback</code>	<p>Commands perform the following tasks:</p> <p>Setting a repeating breakpoint. In other words always halting simulation, when a specified signal changes value</p> <p>Setting a one shot breakpoint. In other words, halting simulation the next time the signal changes value, however, not the subsequent time it changes value</p> <p>Removing a breakpoint from a signal</p> <p>Showing the line number or number in the source code of the statement or statements that causes the current value of a netA longer way to specify this capability is with the <code>callback</code> keyword.</p>
<code>frc</code> or <code>force</code>	<p>Commands perform the following tasks: Forcing a net or a register to a specified value so that this value cannot be changed by subsequent simulation events in the design</p> <p>Releasing a net or register from its forced value A longer way to specify this capability is with the <code>force</code> keyword.</p>

Example 1

The following specification enables many interactive commands including those for displaying the values of signals in specified modules and depositing values to the signals that are registers:

```
acc+=r:top,mid,bot
```

Note that there are no blank spaces in this specification. Including blank spaces causes syntax error.

Example 2

The following specifications enable most interactive commands for most of the modules in a design. They then change the ACC capabilities preventing breakpoint and force commands in instances of modules in Verilog libraries and modules designated as cells with the `'celldefine` compiler directive.

```
acc+=rw,cbk,frc:top+ acc:=rw:%CELL
```

In this example, the first specification enables the interactive commands that are enabled by the `rw`, `cbk`, and `frc` capabilities for module `top`, which, in this example, is the top-level module of the design, and all module instances under it. The second specification limits the interactive commands for the specified modules to only those enabled by the `rw` (same as `r`) capability.

Using the PLI Table File

To specify the PLI table file, specify the `-P` compile-time option, followed by the name of the PLI table file (by convention, the PLI table file has a `.tab` extension).

For example,

```
-P pli.tab
```

When you enter this option in the `vcs` command line, you can also enter C source files, compiled `.o` object files, or `.a` libraries in the `vcs` command line, to specify the PLI application that you want to link with VCS.

For example,

```
vcs -P pli.tab pli.c my_design.v
```

When you include `.o` object files and `.a` libraries, you do not have to recompile the PLI application every time you compile your design.

Enabling ACC Capabilities

VCS allows you to enable ACC capabilities throughout your design and also specify ACC capabilities in specific parts of your design (as described in [PLI Table File](#)). It also

enables you to specify selected write capabilities using a configuration file. As enabling ACC capabilities has an impact on performance, VCS allows you to enable only the ACC capabilities you need.

Using the Configuration File

Specify the configuration file with the `+optconfigfile` compile time option.

For example,

```
+optconfigfile+filename
```

The VCS configuration file enables you to enter statements that specify:

- Using the optimizations of Radiant Technology on part of a design
- Enabling PLI ACC write capabilities for all memories in the design, disabling them for the entire design, or enabling them for part or parts of the design hierarchy
- Four-state simulation for part of a design

The entries in the configuration file override the ACC write-enabling entries in the PLI table file.

The syntax of each type of statement in the configuration file to enable ACC write capabilities is as follows:

```
set writeOnMem;
```

OR

```
set noAccWrite;
```

OR

```
module {list_of_module_identifiers} {accWrite};
```

OR

```
instance {list_of_module_instance_hierarchical_names} {accWrite};
```

OR

```
tree [(depth)] {list_of_module_identifiers} {accWrite};
```

OR

```
signal {list_of_signal_hierarchical_names} {accWrite};
```

Where,

```
set
```

Keyword preceding a property that applies to the entire design.

`writeOnMem`

Enables ACC write to memories (any single or multi-dimensional array of the `reg` data type) throughout the entire design.

`noAccWrite`

Disables ACC write capabilities throughout the entire design.

`accWrite`

Enables ACC write capabilities.

`module`

Keyword specifying that the `accWrite` attribute in this statement applies to all instances of the modules in the list, specified by module identifier.

`list_of_module_identifiers`

Comma separated list of module identifiers (also called module names).

`instance`

Keyword specifying that the `accWrite` attribute in this statement applies to all instances in the list.

`list_of_module_instance_hierarchical_names`

Comma separated list of module instance hierarchical names.

Follow the Verilog syntax for signal names and hierarchical names of module instances.

`tree`

Keyword specifying that the `accWrite` attribute in this statement applies to all instances of the modules in the list, specified by module identifier, and also applies to all module instances hierarchically under these module instances.

`depth`

An integer that specifies how far down the module hierarchy from the specified modules you want to apply the `accWrite` attribute. You can specify a negative value. A negative value specifies descending to the leaf level and counting up levels of the hierarchy to apply these attributes. This specification is optional. Enclose this specification in parentheses: `()`

`signal`

Keyword specifying that the `accWrite` attribute in this statement applies to all signals in the list.

list_of_signal_hierarchical_names

Comma separated list of signal hierarchical names.

Selected ACC Capabilities

This section describes the compile time and runtime options that enable VCS and PLI applications to use only the ACC capabilities you need. The procedure to use these options is as follows:

Use the following runtime options to instruct VCS to keep track of, or learn, the ACC capabilities:

Table 50 Runtime Options to Enable PLI Learn

Option	Description
+vcs+learn+pli	Enables module level PLI learn
+vcs+learn+pli+instpli	Enables instance level PLI learn
+vcs+learn+pli+sigpli	Enables signal PLI learn at module level
+vcs+learn+pli+sigpli +vcs+learn+pli+instpli	Enables signal PLI learn at instance level

Note:

PLI learn at signal or instance level is enabled by default, if the PLI table file has any hierarchical path specified.

VCS uses this information to create a secondary PLI table file named `pli_learn.tab`. You can use this table file to recompile your design so that subsequent simulations use only the ACC capabilities that are needed.

1. Instruct VCS to apply what it has learned in the next compilation of your design, and specify the secondary PLI table file, with the `+applylearn+filename` compile-time option (if you omit `+filename` from the `+applylearn` compile-time option, VCS uses the `pli_learn.tab` secondary PLI table file).
2. Simulate again with a `simv` executable in which only the ACC capabilities you need are enabled.

Learning What Access Capabilities are Used

Include the `+vcs+learn+pli` runtime option to instruct VCS to learn the access capabilities that are used by the modules in your design and write them into a secondary PLI table file named, `pli_learn.tab`.

This file is considered as secondary PLI table file because it does not replace the first PLI table file that you used (if you used one). This file does, however, modifies the access capabilities that are specified in a first PLI table file, or other means of specifying access capabilities, so that you enable only the capabilities you need in subsequent simulations.

See the contents of the `pli_learn.tab` file that VCS writes to understand the access capabilities that are used during simulation. The following is an example of this file:

```
////////////////// SYNOPSYS INC ///////////////////////
// PLI LEARN FILE
// AUTOMATICALLY GENERATED BY VCS (TM) LEARN MODE
///////////////////////////////
acc=r:testfixture
//SIGNAL STIM_SRLS:r
acc=rw:SDFFR
//SIGNAL S1:rw
```

The following line in this file specifies that during simulation, the read capability was needed for signals in the module named `testfixture`.

```
acc=r:testfixture
//SIGNAL STIM_SRLS:r
```

The comment lets you know that the only signal for which this capability is needed is the signal named, `STIM_SRLS`. This line is in the form of a comment because the syntax of the PLI table file does not permit specifying access capabilities on a signal-by-signal basis.

The following line in this file specifies that during simulation, the read and write capabilities are needed for signals in the module named, `SDFFR`, specifically for the signal named `S1`.

```
acc=rw:SDFFR
//SIGNAL S1:rw
```

The following are the examples of the `pli_learn.tab` file for `+vcs+learn+pli+instpli` and `+vcs+learn+pli+sigpli` options:

Consider [Example 202](#), [Example 203](#), and [Example 204](#).

Compile the `test.v` code as follows:

```
% vcs driver.c -P pli.tab test.v -R -sverilog
```

Perform simulation using the `+vcs+learn+pli+instpli` option:

```
% simv +vcs+learn+pli+instpli
```

VCS displays the following output:

```
////////////////// SYNOPSYS INC ///////////////////////
// PLI LEARN FILE
// AUTOMATICALLY GENERATED BY VCS (TM) LEARN MODE
///////////////////////////////
```

```
acc=frc:top
//SIGNAL myInt:frc
acc=rw:top.d1
//SIGNAL g:rw
```

Perform the simulation using the `+vcs+learn+pli+sigpli` option:

```
% simv +vcs+learn+pli+sigpli
```

VCS displays the following output:

```
/////////////// SYNOPSYS INC /////////////
// PLI LEARN FILE
// AUTOMATICALLY GENERATED BY VCS(TM) LEARN MODE
///////////////
acc=frc:top.myInt

acc=rw:dut.g
```

Signs of a Potentially Significant Performance Gain

You might see one of following comments in the `pli_learn.tab` file:

- `!!VCS_LEARNED: NO_ACCESS_PERFORMED`

This indicates that none of the enabled access capabilities were used during the simulation.

- `!!VCS_LEARNED: NO_DYNAMIC_ACCESS_PERFORMED`

This indicates that only static information was accessed through access capabilities and there was no value change information during simulation.

These comments indicate that there is a potentially significant performance gain when you apply the access capabilities in the `pli_learn.tab` file.

Compiling to Enable Only the Access Capabilities You Need

After you have run the simulation to learn what access capabilities are actually used by your design, you can then recompile the design with the information you have learned, so the resulting `simv` executable uses only the access capabilities you require.

When you recompile your design, include the `+applylearn` compile time option.

If you have renamed the `pli_learn.tab` file that VCS writes when you include the `+vcs+learn+pli` runtime option, specify the new filename in the compile time option by appending it to the option with the following syntax:

```
+applylearn+filename
```

When you recompile your design with the `+applylearn` compile time option, it is important that you also re-enter all the compile time options that you used for the previous compilation. For example, if in a previous compilation, have specified a PLI table file with

the `-P` compile-time option, specify this PLI table file again, using the `-P` option along with the `+applylearn` option.

Note:

If you change your design after VCS writes the `pli_learn.tab` file, and you want to make sure that you are using only the access capabilities you need, you will need to have VCS write another one by including the `+vcs+learn+pli` runtime option and then compiling your design again with the `+applylearn` option.

Limitations

VCS does not maintain a history of all access capabilities. However, the capabilities that are maintained and specified in the `pli_learned.tab` file are as follows:

- `r` - read
- `rw` - read and write
- `cbk` - callbacks
- `cbka` - callback all including unnamed objects
- `frc` - forcing values on signals
- The `+applylearn` compile-time option does not work if you also use either the `+multisource_int_delays` or `+transport_int_delays` compile time option , because interconnect delays need global access capabilities.

If you enter the `+applylearn` compile-time option more than once on the `vcs` command line, VCS ignores all occurrences except the first.

Note:

The `+applylearn` option is for performance and if you enter this option with options for debugging, such as `-debug_access`, VCS ignores the debugging options.

Using VPI Routines

To enable VPI capabilities in VCS, use the `+vpi` compilation/elaboration option, as shown in the following example:

```
% vcs +vpi top -P test.tab test.c
```

The header file for the VPI routines is `$VCS_HOME/include/vpi_user.h`.

The `+vpi` option is automatically enabled as part of `-debug_access`.

The `+vpi+1` option prevents VPI capability (like line callbacks and breakpoints) on statements inside tasks, functions, initial blocks, and always blocks. This takes precedence over `+vpi`.

The `+vpi+1+assertions` option is same as `+vpi+1`, but allows VPI capability on assertions. This takes precedence over `+vpi+1`.

You can also use the PLI `.tab` file to associate your user-defined system tasks with your VPI routines. For more information, see [PLI Table File for VPI Routines](#).

Support for VPI Callbacks

The `vpi_register_cb()` callback mechanism can be registered for callbacks to occur for simulation events, such as value changes on an expression or terminal, or the execution of a behavioral statement. When the `cb_data_p->reason` field is set to one of the following, the callback occurs as follows:

- `cbForce/cbRelease` — After a force or release is occurred
- `cbAssign/cbDeassign` — After a procedural assign or deassign statement is executed

VPI callbacks `cbForce` and `cbRelease` are supported with the following limitations:

- The force and release commands generates a callback only if `cb_data_p > obj` is a valid handle. If it is set to NULL, it does not generate a callback.
- For `cbForce`, `cbRelease`, `cbAssign`, and `cbDeassign` callbacks, the handle that you supplied while registering the callback is returned and not the corresponding statement handle [NULL handles are not allowed].

For more information about the VPI callbacks, see section *Simulation-event-related callbacks in the Verilog IEEE LRM 1364-2001*.

Integrating a VPI Application With VCS

If you create one or more shared libraries for a VPI application, the application must not contain the `vlog_startup_routines` array.

Instead, enter the `-load` compile-time option to specify the registration routine. The syntax is as follows:

```
-load shared_library:registration_routine
```

It is not required to specify the path name of the shared library, if that path is part of your `LD_LIBRARY_PATH` environment variable.

The following are some examples of using this option:

- -load lib1.so:my_register

The `my_register()` routine is in `lib1.so`. The location of `lib1.so` is in the `LD_LIBRARY_PATH` environment variable.

- -load lib1.so:my_register,new_register

The registration routines `my_register()` and `new_register()` are in `lib1.so`. The location of `lib1.so` is in the `LD_LIBRARY_PATH` environment variable.

- -load lib1.so:my_register -load lib2.so:new_register

The registration routine `my_register()` is in `lib1.so` and the second registration routine `new_register()` is in `lib2.so`. The path to both of these libraries are in the `LD_LIBRARY_PATH` environment variable. You can enter more than one `-load` options to specify multiple shared libraries and their registration routines.

- -load lib1.so:my_register

The registration routine `my_register()` is in `lib1.so`. The location of `lib1.so` is in the `LD_LIBRARY_PATH` environment variable.

- -load /usr/lib/mylib.so:my_register

The registration routine `my_register()` is in `lib1.so`, which is in `/usr/lib/mylib.so`, and not in the `LD_LIBRARY_PATH` environment variable.

PLI Table File for VPI Routines

The PLI table file for VPI routines works similar to and with the same syntax as a PLI table file for user-defined system tasks that execute C functions. The following is an example of such a PLI table file:

```
$set_mipd_delays call=PLIbook_SetMipd_calltf
check=PLIbook_SetMipd_complet tf acc=mip,mp,gate,tchk,rw:test+
```

Note that this entry includes `acc=` even though the C functions in the PLI specification call VPI routines instead of PLI 1.0 routines. The syntax has not changed; you use the same syntax for enabling PLI 1.0 and PLI 2.0 routines.

This PLI table file is used for an example file named `set_mipd_delays_vpi.c`, which is available with *The Verilog PLI Handbook* by Stuart Sutherland, Kluwer Academic Publishers, Boston, Dordrecht, and London.

Virtual Interface Debug Support

A Virtual Interface is a reference object that can either be initially assigned at its declaration or not assigned. You can debug the Virtual Interface object when it is initially assigned or not assigned within a module or a class.

To debug the Virtual Interface objects, the VPI properties defined in the SystemVerilog LRM, such as `vpiVirtual`, `vpiActual`, and `vpiInterfaceDecl`, are supported. For more information about these properties, see the *IEEE SystemVerilog LRM*.

Example

The following example shows the VPI routines usage for Virtual Interface Debug:

`virtual_interface.sv`

```
interface ifc (input logic clk);
    event reset;
    int ifci;
    modport tracker (input clk);
endinterface: ifc

package p;

class C;
    virtual ifc.tracker busmpIF; #VI declared in class scope
    virtual ifc busIF;
    int i;

    function new (virtual ifc inf);
        busIF = inf;
    endfunction // new

    function test(virtual ifc inf);
        busIF = inf;
        $display("hello");
    endfunction: test
endclass: C
endpackage: p

module mod( input logic clk);
    import p::*;
    ifc trkIF(.clk(clk));
    virtual ifc modbusIF = trkIF; #VI declared in Module scope
    virtual ifc.tracker modportIF2;

    C c;

    initial begin
        `ifdef DUMP
            $vcpluson;
```

```

`endif
    c = new(trkIF);
    c.test(modbusIF);
    modbusIF.ifci <= 10;
#1
    $getVar;
    $display("end the first round\n");
#1
    modbusIF.ifci <= 11;
$getVar;
    $display("end the second round.");
end
endmodule: mod

```

pli.c

```

#include <stdio.h>
#include <stdlib.h>
#include "vcs_vpi_user.h"
#include "sv_vpi_user.h"

void traverse(){
    vpiHandle Han, iterHan, scanHan, cls, obj, intfHan, Hactual;

    vpi_configure(vpiDisplayWarnings,"true");

    intfHan = vpi_handle_by_name("mod.vbusIF",NULL);
    vpi_printf("\tVARIABLE `\$s'\n", vpi_get_str(vpiName,intfHan));
    vpi_printf("\t--- DefName `\$s'\n\t--- FullName:`\$s\n\t---\n");
    vpiType:`\$s\n",
              vpi_get_str(vpiDefName,intfHan),
    vpi_get_str(vpiFullName,intfHan),
              vpi_get_str(vpiType,intfHan));
    if(vpi_get(vpiVirtual, intfHan)){
        vpi_printf("\t\$s is Virtual\n");
        vpi_get_str(vpiName,intfHan));
    }
    Hactual = vpi_handle(vpiActual, intfHan);
    if ( Hactual )
    {
        vpi_printf("\n\tActual `\$s'\n", vpi_get_str(vpiName,Hactual));
        vpi_printf("\t--- DefName `\$s'\n\t--- FullName:`\$s\n\t---\n");
        vpiType:`\$s\n",
              vpi_get_str(vpiDefName,Hactual),
    vpi_get_str(vpiFullName,Hactual),
              vpi_get_str(vpiType,Hactual));
        if(vpi_get(vpiVirtual, Hactual)){
            vpi_printf("\tActual Handle is Virtual Interface\n");
        }
    }
}

```

pli.tab

```
$getVar call=traverse acc+=r:* acc+=cbk:*
```

To compile this example code, use the following commands:

```
% vcs -P pli.tab pli.c virtual_interface.sv -debug_access+all -sverilog
% simv -verdi
```

Limitations

The following are the limitations with this functionality:

- Virtual Interface passed as a method port is not displayed in Verdi.
- Virtual Interface as an array is not supported.
- Virtual Interface debugging is not supported in UCLI.
- \$vcdplusmsglog do not dump Virtual Interface.

Unimplemented VPI Routines

VCS does not support all the functionalities specified for VPI routines in the *IEEE Standard Verilog® Hardware Description Language, 1364-2001*, as some routines would be rarely used and some of the data access operations of other routines would be rarely used. The unimplemented routines are as follows:

- vpi_get_data
- vpi_put_data
- vpi_sim_control

Object data model diagrams in the *IEEE Verilog Language Reference Manual* specify that some VPI routines should be able to access data that is rarely needed. These routines and the data they cannot access are as follows:

vpi_get_value

- Cannot retrieve the value of var select objects (see diagram 26.6.8 Variables in *IEEE Standard Verilog® Hardware Description Language, 1364-2001*) and func call objects (diagram 26.6.18 Task, function declaration in *IEEE Standard Verilog® Hardware Description Language, 1364-2001*).
- Cannot retrieve the value of VPI operators (expressions) unless they are arguments to system tasks or system functions.
- Cannot retrieve the value of UDP table entries (`vpiVectorVal` not implemented).

`vpi_put_value`

Cannot set the value of `var` select objects (see diagram 26.6.8 Variables in *IEEE Standard Verilog® Hardware Description Language, 1364-2001*) and primitive objects (see diagram 26.6.13 Primitive, `prim` term in *IEEE Standard Verilog® Hardware Description Language, 1364-2001*).

`vpi_get_delays`

Cannot retrieve the values of continuous assign objects (see diagram 26.6.24 Continuous assignment in *IEEE Standard Verilog® Hardware Description Language, 1364-2001*) or procedurally assigned objects.

`vpi_put_delays`

Cannot put values on continuous assign objects (see diagram 26.6.24 Continuous assignment in *IEEE Standard Verilog® Hardware Description Language, 1364-2001*) or procedurally assigned objects.

`vpi_register_cb`

Cannot register the following types of callbacks that are defined for this routine:

cbError	cbPliError
---------	------------

Modified VPI Features

VCS complies with some of the constants that are standardized to comply with the IEEE LRM 1800-2009.

[Table 51](#) provides the modified constants for the handle type returned by `vpi_get(vpiType, handle)`.

Table 51 Modified Constants for the Handle Type Returned by vpi_get(vpiType, handle)

In G-2012.09 and earlier releases	From H-2013.06 release
<code>vpilmmediateAssertType</code>	<code>vpilmmediateAssert</code>
<code>vpilmmediateAssumeType</code>	<code>vpilmmediateAssume</code>
<code>vpilmmediateCoverType</code>	<code>vpilmmediateCover</code>
<code>vpiAssertType</code>	<code>vpiAssert</code>
<code>vpiAssumeType</code>	<code>vpiAssume</code>

Table 51 Modified Constants for the Handle Type Returned by `vpi_get(vpiType, handle)` (Continued)

vpiCoverType	vpiCover
*vpilmmediateFinalAssertType	vpilmmediateAssert
*vpilmmediateFinalAssumeType	vpilmmediateAssume
*vpilmmediateFinalCoverType	vpilmmediateCover
vpiEndedOp	vpiTriggeredOp
vpiModPortPort	vpiModportPort

* Use `vpi_get(vpiFinal, <assert_handle>)` to determine if they are final type.

[Table 52](#) provides the modified constants used in iterators.

Table 52 Modified Constants Used in Iterators `vpi_iterate(constant)`

In G-2012.09 and earlier releases	From H-2013.06 release
vpiSequence	vpiExpr
vpiSequenceExpr	vpiExpr
vpiModPort	vpiModport
vpildentifier	vpiSeqFormalDecl when ref handle is vpiSequenceDecl
vpildentifier	vpiPropFormalDecl when ref handle is vpiPropertyDecl

Example

pli.c

```
modport_iter = vpi_iterate(vpiModPort, refHandle);
// here refHandle points to object of type interface
```

Error Message

```
< ...: error: 'vpiModPort' undeclared (first use in this function)
```

Solution

You must change the code to comply with the LRM 1800-2009.

[Table 53](#) provides the constants that are updated for the new value.

Table 53 Constants That Are Updated for the New Value

Modified Constants
vpiPortType
vpiInterfacePort
vpiMember
vpiStructUnionMember
vpiAssertion
vpiClockingEvent
vpiDisableCondition
vpiIfOp
vpiElseOp
vpiCompAndOp
vpiCompOrOp
vpiAssignmentOp
vpiAcceptOnOp
vpiRejectOnOp
vpiSyncAcceptOnOp
vpiSyncRejectOnOp
vpiOverlapFollowedByOp
vpiNonOverlapFollowedByOp
vpiNexttimeOp
vpiAlwaysOp

Table 53 Constants That Are Updated for the New Value (Continued)

vpiEventuallyOp
vpiUntilOp
vpiUntilWithOp
vpiImpliesOp
vpiInsideOp

Example

plic.c

```
assertIter = vpi_iterate(700, 0x0);Solution
```

Warning Message

Warning-[VCS-VPI-DEPRECATED] VPI value deprecated

In 'vpi_iterate' call, the VPI value for `vpiAssertion` (700) is deprecated and will not be supported in the next release.
Please check, fix and recompile your PLI program.

Solution

When using the hardcoded values, change the code to use the constants.

Using VHPI Routines

VHPI enables you to use foreign architecture-based models written in C language in VCS VHDL.

Diagnostics for VPI/VHPI PLI Applications

As defined in the LRM, VPI/VHPI remains silent when an error occurs. The application checks for error status to report an error. If error detection mechanisms are not in place, the C code of the application must be modified and recompiled. In addition, you might have to recompile the HDL code, if required.

However, you can use the following runtime diagnostic options to make the PLI application to report errors without code modification:

- -diag vpi
- -diag vhpi

For more information, see [Diagnostics for VPI/VHPI PLI Applications](#).

Using DirectC

DirectC is an extended interface between Verilog and C/C++. It is an alternative to the PLI. Unlike the PLI, DirectC enables you to do the following:

- More efficiently pass values between Verilog module instances and C/C++ functions by calling the functions directly along with actual parameters in your Verilog code.
- Pass more types of data between Verilog and C/C++. With the PLI, you can only pass Verilog information to and from a C/C++ application. With DirectC you do not have this limitation.

With DirectC, for example, you can model a simulation environment for your design in C/C++ in which you can pass pointers from the environment to your design and store them in Verilog signals, and at a later simulation time, pass these pointers to the simulation environment.

Similarly, you can use DirectC to develop applications to run with VCS to which you can pass pointers to the location of simulation values for your design.

DirectC is an alternative to PLI, however, DirectC is not a replacement for PLI. Certain functionalities can only be enabled using PLI. For example, there are PLI TF and ACC routines to implement a callback to start a C/C++ function when a Verilog signal changes value. However, this functionality cannot be enabled with DirectC.

You can use Direct C/C++ function calls for existing and proven C code as well as C/C++ code that you write in the future. You can also use them without much rewriting of, or additions to, your Verilog code. You call them the same way you call (or enable) a Verilog function or Verilog task.

This section describes the DirectC interface in the following sections:

- [Using Direct C/C++ Function Calls](#)
- [Using Direct Access](#)
- [Using Abstract Access](#)

- [Enabling C/C++ Functions](#)
 - [Extended BNF for External Function Declarations](#)
-

Using Direct C/C++ Function Calls

To enable a direct call of a C/C++ function during simulation, perform the following tasks:

1. Declare the function in your Verilog code.
2. Call the function in your Verilog code.
3. Compile/Elaborate your design and C/C++ code using elaboration/compile-time options for DirectC.

However, there are complications to this otherwise straightforward procedure.

DirectC allows the invocation of C++ functions that are declared in C++ using the `extern "C"` linkage directive. The `extern "C"` directive is necessary to protect the name of the C++ function from being mangled by the C++ compiler. Plain C functions do not undergo mangling, and therefore, do not need any special directive.

The declaration of these functions involves specifying a direction for the parameters of the C function, because, in the Verilog environment, they become analogous to Verilog tasks as well as functions. Verilog tasks are similar to void C functions in that they do not return a value. However, Verilog tasks do have input, output, and inout arguments, whereas C function parameters do not have explicitly declared directions. For more information, see [Declaring the C/C++ Function](#).

You can use the following access modes for C/C++ function calls. These modes do not make much difference in your Verilog code; they only pertain to the development of the C/C++ function. They are as follows:

- The slightly more efficient direct access mode - this mode has rules for how values of different types and sizes are passed to and from Verilog and C/C++. For more information about this mode, see [Using Direct Access](#).
- The slightly less efficient, but with better error handling abstract access mode. VCS creates a descriptor for each actual parameter of the C function. You access these descriptors using a specially defined pointer called a handle. All formal arguments are handles. DirectC includes a library of accessory functions for using these handles. For more information, see [Using Abstract Access](#).

The abstract access library of accessory functions contains operations for reading and writing values and for querying about argument types, sizes, and so on. An alternative library, with perhaps different levels of security or efficiency, can be developed and used in abstract access without changing your Verilog or C/C++ code.

If you have an existing C/C++ function that you want to use in a Verilog design, consider using direct access and see if you really need to edit your C/C++ function or write a wrapper so that you can use direct access inside the wrapper. There is a small performance gain by using direct access compared to abstract access.

If you are about to write a C/C++ function to use in a Verilog design, decide how you want to use it in your Verilog code and write the external declaration for it, then decide which access mode you want. You can change the mode later with a small change in your Verilog code.

Using abstract access is “safer” because the library of accessory functions for abstract access issues error messages that help you to debug the interface between C/C++ and Verilog. With direct access, errors result in segmentation faults, memory corruption, and so on.

Abstract access can be generalized more easily for your C/C++ function. For example, with open arrays you can call the function with 8-bit arguments at one point in your Verilog design and call it again some place else with 32-bit arguments. The accessory functions can manage the differences in size. With abstract access you can have the size of a parameter returned to you. With direct access you must know the size.

Functioning of C/C++ Code in a Verilog Environment

Similar to Verilog functions, and unlike Verilog tasks, no simulation time elapses during the execution of a C/C++ function.

C/C++ functions work in two-state and four-state simulation, and in some cases, work better in two-state simulation. Short vector values, 32-bits or less, are passed by value instead of by reference. Using two-state simulation makes a difference in how you declare a C/C++ function in your Verilog code.

The parameters of C/C++ functions, are analogous to the arguments of Verilog tasks. They can be input, output, or inout, similar to the arguments of Verilog tasks. Do not specify them as such in your C code, but you can when you declare them in your Verilog code. Accordingly your Verilog code can pass values to parameters declared to be input or inout, but not output, in the function declaration in your Verilog code, and your C function can only pass values from parameters declared to be inout or output, but not input, in the function declaration in your Verilog code.

If a C/C++ function returns a value to a Verilog register (the C/C++ function is in an expression that is assigned to the register) the return value of the C/C++ function is restricted to the following:

- The value of a scalar `reg` or `bit`

Note:

In two-state simulation, a `reg` has a new name, `bit`.

- The value of the C type `int`
- A pointer
- A short, 32 bits or less, vector `bit`
- The value of a Verilog `real` which is represented by the C type `double`

Therefore, C/C++ functions cannot return the value of a four-state vector `reg`, long (longer than 32 bits) vector `bit`, or Verilog `integer`, `realtime`, or `time` data type. You can pass these type of values out of the C/C++ function using a parameter that you declare to be `inout` or `output` in the declaration of the function in your Verilog code.

Declaring the C/C++ Function

A partial EBNF specification for external function declaration is as follows:

```

source_text      ::= description +
description       ::= module | user_defined_primitive | extern_declaration
extern_declaration ::= extern access_mode ? attribute ? return_type
function_id
    (extern_func_args ? ) ;
access_mode      ::= ( "A" | "C" )
attribute         ::= pure
return_type       ::= void | reg | bit | DirectC_primitive_type
| small_bit_vector
small_bit_vector ::= bit [ (constant_expression : constant_expression) ]
extern_func_args  ::= extern_func_arg ( , extern_func_arg ) *
extern_func_arg   ::= arg_direction ? arg_type arg_id ?
arg_direction     ::= input | output | inout
arg_type          ::= bit_or_reg_type | array_type | DirectC_primitive_type
bit_or_reg_type   ::= ( bit | reg ) optional_vector_range ?
optional_vector_range ::= [ ( constant_expression : constant_expression ) ? ]
array_type        ::= bit_or_reg_type array [ (constant_expression :
constant_expression) ? ]
DirectC_primitive_type ::= int | real | pointer | string

```

Where,

`extern`

Keyword that begins the declaration of the C/C++ function declaration.

access_mode

Specifies the mode of access in the declaration. Enter C for direct access, or A for abstract access. Using this entry enables some functions to use direct access and others to use abstract access.

attribute

An optional attribute for the function. The `pure` attribute enables some optimizations. Enter this attribute if the function has no side effects and is dependent only on the values of its input parameters.

return_type

The valid return types are `int`, `bit`, `reg`, `string`, `pointer`, and `void`. See [Table 54](#) for a description of what these types specify.

small_bit_vector

Specifies a bit-width of a returned vector `bit`. A C/C++ function cannot return a four-state vector `reg`, but it can return a vector `bit` if its bit-width is 32 bits or less.

function_id

The name of the C/C++ function.

direction

One of the following keywords: `input`, `output`, `inout`. In a C/C++ function, these keywords specify the same thing that they specify in a Verilog task; see [Table 55](#).

arg_type

The valid argument types are `real`, `reg`, `bit`, `int`, `pointer`, `string`.

[bit_width]

Specifies the bit-width of a vector `reg` or `bit` that is an argument to the C/C++ function. You can leave the bit-width open by entering `[]`.

array

Specifies that the argument is a Verilog memory.

[index_range]

Specifies a range of elements (words, addresses) in the memory. You can leave the range open by entering `[]`.

arg_id

The Verilog register argument to the C/C++ function that becomes the actual parameter to the function.

Note:

Argument direction (that is,, input, output, inout) applies to all arguments that follow it until the next direction occurs; the default direction is input.

Table 54 C/C++ Function Return Types

Return Type	Specifies
int	The C/C++ function returns a value for type int.
bit	The C/C++ function returns the value of a bit, which is a Verilog reg in two state simulation, if it is 32 bits or less.
reg	The C/C++ function returns the value of a Verilog scalar reg.
string	The C/C++ function returns a pointer to a character string.
pointer	The C/C++ function returns a pointer.
void	The C/C++ function does not return a value.

Table 55 C/C++ Function Argument Directions

keyword	Specifies
input	The C/C++ function can only read the value or address of the argument. If you specify an input argument first, you can omit the <code>input</code> keyword.
output	The C/C++ function can only write the value or address of the argument.
inout	The C/C++ function can both read and write the value or address of the argument.

Table 56 C/C++ Function Argument Types

keyword	Specifies
real	The C/C++ function reads or writes the address of a Verilog real data type.
reg	The C/C++ function reads or writes the value or address of a Verilog reg.
bit	The C/C++ function reads or writes the value or address of a Verilog reg in two state simulation.

Table 56 C/C++ Function Argument Types (Continued)

keyword	Specifies
int	The C/C++ function reads or writes the address of a C/C++ int data type.
pointer	The C/C++ function reads or writes the address that a pointer is pointing to.
string	The C/C++ function reads from or writes to the address of a string.

Example 1

```
extern "A" reg return_reg (input reg r1);
```

This example declares a C/C++ function named `return_reg`. This function returns the value of a scalar reg. When you call this function, the value of a scalar reg named `r1` is passed to the function. This function uses abstract access.

Example 2

```
extern "C" bit [7:0] return_vector_bit (bit [7:0] r3);
```

This example declares a C/C++ function named `return_vector_bit`. This function returns an 8-bit vector bit (a reg in two state simulation). When you call this function, the value of an 8-bit vector bit (a reg in two state simulation) named `r3` is passed to the function. This function uses direct access.

The keyword `input` is omitted. This keyword can be omitted if the first argument specified is an input argument.

Example 3

```
extern string return_string();
```

This example declares a C/C++ function named `return_string`. This function returns a character string and takes no arguments.

Example 4

```
extern void receive_string( input string r5);
```

This example declares a C/C++ function named `receive_string`. It is a void function. At some time earlier in the simulation, another C/C++ function passed the address of a character string to reg `r5`. When you call this function, it reads the address in reg `r5`.

Example 5

```
extern pointer return_pointer();
```

This example declares a C/C++ function named `return_pointer`. When you call this function, it returns a pointer.

Example 6

```
extern void receive_pointer (input pointer r6);
```

This example declares a C/C++ function named `receive_pointer`. When you call this function the address in reg `r6` is passed to the function.

Example 7

```
extern void memory_reorg (input bit [32:0] array [7:0] mem2, output bit [32:0] array [7:0] mem1);
```

This example declares a C/C++ function named `memory_reorg`. When you call this function, the values in memory `mem2` are passed to the function. After the function executes, new values are passed to memory `mem1`.

Example 8

```
extern void incr (inout bit [] r7);
```

This example declares a C/C++ function named `incr`. When you call this function, the value in bit `r7` is passed to the function. When it finishes executing, it passes a new value to bit `r7`. Bit width for vector bit `r7` is not specified. This allows you to use various sizes in the parameter declaration in the C/C++ function header.

Example 9

```
extern void passbig (input bit [63:0] r8,
                     output bit [63:0] r9);
```

This example declares a C/C++ function named `passbig`. When you call this function, the value in bit `r8` is passed by reference to the function because it is more than 32 bits; see [Using Direct Access](#). When it finishes executing, a new value is passed by reference to bit `r9`.

Calling the C/C++ Function

After declaring the C/C++ function, you can call it in your Verilog code. You call a void C/C++ function in the same manner as you call a Verilog task-enabling statement, that is, by entering the function name and its arguments, either on a separate line in an `always` or `initial` block, or in the procedural statements in a Verilog task or function declaration. Unlike Verilog tasks, you can call a C/C++ function in a Verilog function.

You call a non-void (returns a value) C/C++ function in the same manner as you call a Verilog function call, that is, by entering its name and arguments, either in an expression on the RHS of a procedural assignment statement in an `always` or `initial` block, or in a Verilog task or function declaration.

Examples

```
r2=return_reg(r1);
```

The value of scalar reg `r1` is passed to C/C++ function `return_reg`. It returns a value to reg `r2`.

```
r4=return_vector_bit(r3);
```

The value of vector bit `r3` is passed to C/C++ function `return_vector_bit`. It returns a value to vector bit `r4`.

```
r5=return_string();
```

The address of a character string is passed to reg `r5`.

```
receive_string(r5);
```

The address of a character string in reg `r5` is passed to C/C++ function `receive_string`.

```
r6=return_pointer();
```

The address pointed to in a pointer in C/C++ function `return_pointer` is passed to reg `r6`.

```
get_pointer(r6);
```

The address in reg `r6` is passed to C/C++ function `get_pointer`.

```
memory_reorg(mem1,mem2);
```

In this example, all the values in memory `mem2` are passed to C/C++ function `memory_reorg`, and when it finishes executing, it passes new values to memory `mem1`.

```
incr(r7);
```

In this example, the value of bit `r7` is passed to C/C++ function `incr`, and when it finishes executing, it passes a new value to bit `r7`.

Storing Vector Values in Machine Memory

If you are using direct access you must know how vector values are stored in memory. This information is also helpful if you are using abstract access.

Verilog four-state simulation values (1, 0, x, and z) are represented in machine memory with data and control bits. The control bit differentiates between the 1 and x and the 0 and z values, as shown in the following table:

Simulation Value	Data Bit	Control Bit
1	1	0

Simulation Value	Data Bit	Control Bit
x	1	1
0	0	0
z	0	1

When a routine returns Verilog data to a C/C++ function, how that data is stored depends on whether it is from a two-state or four-state value, and whether it is from a scalar, a vector, or from an element in a Verilog memory.

For a four-state vector (denoted by the keyword reg), the Verilog data is stored in type vec32, which for abstract access is defined as follows:

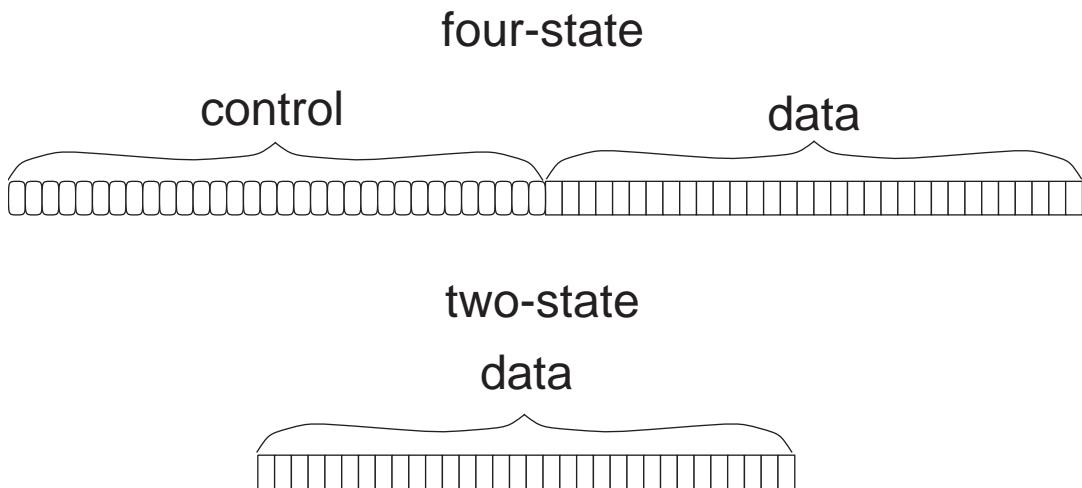
```
typedef unsigned int U;
typedef struct { U c; U d; } vec32;
```

So, type `vec32*` has two members of type `U`; member `c` is for control bits and member `d` is for data bits.

For a two-state vector bit, the Verilog data is stored in type `U*`.

Vector values are stored in arrays of chunks of 32 bits. For four-state vectors there are chunks of 32 bits for data values and 32 bits for control values. For two-state vectors, there are chunks of 32 bits for data values.

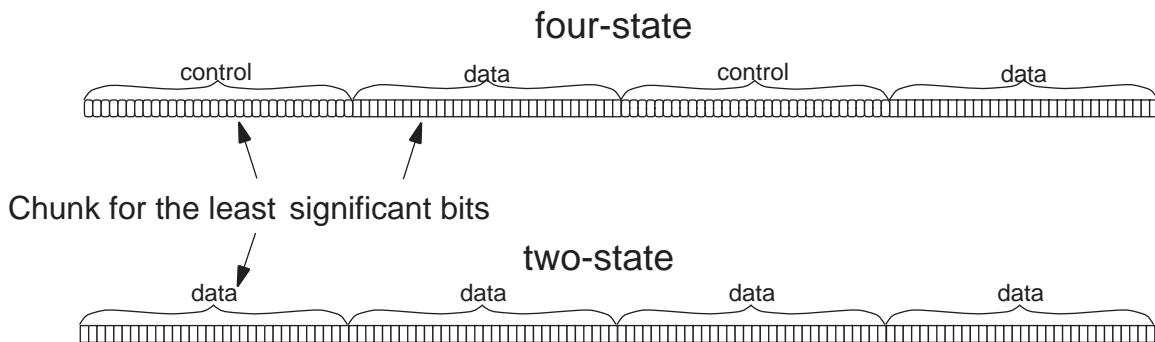
Figure 171 Storing Vector Values



Long vectors, more than 32 bits, have their value stored in more than one group of 32 bits and can be accessed by chunk. Short vectors, 32 bits or less, are stored in a single chunk.

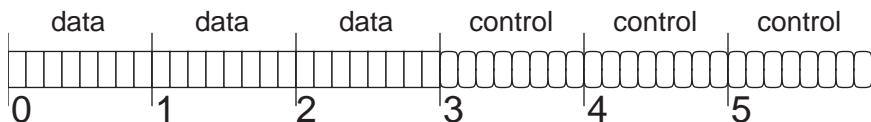
For long vectors, the chunk for the least significant bits come first, followed by the chunks for the more significant bits.

Figure 172 Storing Vector Values of More than 32 Bits



In an element in a Verilog memory, for each eight bits in the element, there is a data byte and a control byte with an additional set of bytes for remainder bit. Therefore, if a memory has 9 bits, it would need two data bytes and two control bytes. If it had 17 bits, it would need three data bytes and three control bytes. All the data bytes precede the control bytes. Two-state memories have both data and control bytes, but the bits in the control bytes always have a zero value.

Figure 173 Storing Verilog Memory Elements in Machine Memory



Converting Strings

There are no *true* strings in Verilog, and a string literal, like "some_text," is only a notation for vectors of bits, based on the same principle as binary, octal, decimal, and hexadecimal numbers. So there is a need for a conversion between the two representations of "strings": the C-style representation (which actually is a pointer to the sequence of bytes terminated with null byte) and the Verilog vector encoding a string.

DirectC includes the `vc_ConvertToString()` routine that you can use to convert a Verilog string to a C string. Its syntax is as follows:

```
void vc_ConvertToString(vec32 *, int, char *)
```

There are scenarios in which a string is created on the Verilog side and is passed to C code, and therefore, has to be converted from Verilog representation to C representation.

Consider the following example:

```
extern void WriteReport(string result_code, .... /* other stuff */);
```

Example of a valid call:

```
WriteReport("Passes", ....);
```

Example of incorrect code:

```
reg [100*8:1] message;  
. . .  
message = "Failed";  
. . .  
WriteReport(message, ....);
```

This call causes a core dump because the function expects a pointer and gets some random bits instead.

It might happen that a string, or different strings, are assigned to a signal in Verilog code and their values are passed to C.

For example,

```
task DoStuff(..., result_code); ... output reg [100*8:1] result_code;  
begin  
. . .  
if (...) result_code = "Bus error";  
. . .  
if (...) result_code = "Erroneous address";  
. . .  
else result_code = "Completed");  
end  
endtask  
  
reg [100*8:1] message;  
. . .  
DoStuff(..., message);
```

You cannot directly call the function as follows:

```
WriteReport(message, ...)
```

Following are the solutions:

Solution 1: Write a C wrapper function, pass `message` to this function and perform the conversion of vector-to-C string in C, calling `vc_ConvertToString`.

Solution 2: Perform the conversion on the Verilog side. This requires some additional effort, as the memory space for a C string has to be allocated as follows:

```
extern "C" string malloc(int);
extern "C" void vc_ConvertToString(reg [], int, string);
// this function comes from DirectC library

reg [31:0] sptr;
.
.
.

// allocate memory for a C-string
sptr = malloc(8*100+1);
//100 is the width of 'message', +1 is for NULL terminator
// perform conversion
vc_ConvertToString(message, 800, sptr);
WriteReport(sptr, ...);
```

Avoiding a Naming Problem

In a module definition, do not call an external C/C++ function with the same name as the module definition. The following is an example of the type of source code you should avoid:

```
extern void receive_string (input string r5);
.

.

.

module receive_string;
.

.

.

always @ r5
begin
.

.

.

receive_string(r5);
.

.

.

end
endmodule
```

Using Pass by Reference

You can use pass by reference with DirectC. The following source files: `main.v` and `pythag.c`, illustrate using pass by reference.

main.v

```
extern void pythag(inout real);
module main;
real p;
initial begin
    p = 7.89;
    pythag(p);
    $finish;
end
endmodule
```

pythag.c

```
#include <stdio.h>
void pythag(double *p)
{
    printf ("Passed real value from verilog p=%f \n", *p);
}
```

You can try out this example with the following command-line:

```
% vcs +vc main.v pythag.c -R -l somv.log
```

At runtime, VCS displays the following:

```
Passed real value from verilog p=7.890000
```

Using Direct Access

Direct access for C/C++ routines whose formal parameters are of the following types:

int	int*	double*	void*	void**
char*	char**	scalar	scalar*	
U*	vec32	UB*		

Some of these type identifiers are standard C/C++ types; those that are not, are defined with the following `typedef` statements:

```
typedef unsigned int U;
typedef unsigned char UB;
typedef unsigned char scalar;
typedef struct {U c; U d;} vec32;
```

The type identifier you use depends on the corresponding argument direction, type, and bit-width that you specified in the declaration of the function in your Verilog code. The following rules apply:

- Direct access passes all output and inout arguments by reference, so their corresponding formal parameters in the C/C++ function must be pointers.
- Direct access passes a Verilog bit by value only if it is 32 bits or less. If it is larger than 32 bits, direct access passes the bit by reference so the corresponding formal parameters in the C/C++ function must be pointers if they are larger than 32 bits.
- Direct access passes a scalar reg by value. It passes a vector reg direct access by reference, so the corresponding formal parameter in the C/C++ function for a vector reg must be a pointer.
- An open bit-width for a reg makes it possible for you to pass a vector reg, so the corresponding formal parameter for a reg argument, specified with an open bit-width, must be a pointer. Similarly, an open bit-width for a bit makes it possible for you to pass a bit larger than 32 bits, so the corresponding formal parameter for a bit argument specified with an open bit width must be a pointer.
- Direct access passes by value the following types of input arguments: `int`, `string`, and `pointer`.
- Direct access passes input arguments of type `real` by reference.

The following tables show the mapping between the data types you use in the C/C++ function and the arguments you specify in the function declaration in your Verilog code.

Table 57 For Input Arguments

argument type	C/C++ formal parameter data type	Passed by
<code>int</code>	<code>int</code>	value
<code>real</code>	<code>double*</code>	reference
<code>pointer</code>	<code>void*</code>	value
<code>string</code>	<code>char*</code>	value
<code>bit</code>	<code>scalar</code>	value
<code>reg</code>	<code>scalar</code>	value
<code>bit [] - 1-32 bit wide vector</code>	<code>U</code>	value
<code>bit [] - open vector, any vector wider than 32 bits</code>	<code>U*</code>	reference

Table 57 For Input Arguments (Continued)

argument type	C/C++ formal parameter data type	Passed by
reg [] - 1-32 bit wide vector	vec32*	reference
array [] - open vector, any vector wider than 32 bits	UB*	reference

Table 58 For Output and Inout Arguments

argument type	C/C++ formal parameter data type	Passed by
int	int*	reference
real	double*	reference
pointer	void**	reference
string	char**	reference
bit	scalar*	reference
reg	scalar*	reference
bit [] - any vector, including open vector	U*	reference
reg[] - any vector, including open vector	vec32*	reference
array[] - any array, 2 state or 4 state, including open array	UB*	reference

In direct access, the return value of the function is always passed by value. The data type of the returned value is the same as the input argument.

Example 1

Consider the following C/C++ function declared in the Verilog source code:

```
extern reg return_reg (input reg r1);
```

In this example, the function named `return_reg` returns the value of a scalar reg. The value of a scalar reg is passed to it. The header of the C/C++ function is as follows:

```
extern "C" scalar return_reg(scalar reti);
scalar return_reg(scalar reti);
```

If `return_reg()` is a C++ function, it must be protected from name mangling, as follows:

```
extern "C" scalar return_reg(scalar reti);
```

Note:

The `extern "C"` directive has been omitted in subsequent examples for brevity.

A scalar reg is passed by value to the function so the parameter is not a pointer. The parameter's type is scalar.

Example 2

Consider the following C/C++ function declared in the Verilog source code:

```
extern "C" bit [7:0] return_vector_bit (bit [7:0] r3);
```

In this example, the function named `return_vector_bit` returns the value of a vector bit. The "`C`" entry specifies direct access. Typically, a declaration includes this when some other functions use abstract access. The value of an 8-bit vector bit is passed to it. The header of the C/C++ function is as follows:

```
U return_vector_bit(U returner);
```

A vector bit is passed by value to the function because the vector bit is less than 33 bits so the parameter is not a pointer. The parameter's type is `U`.

Example 3

Consider the following C/C++ function declared in the Verilog source code:

```
extern void receive_pointer ( input pointer r6 );
```

In this example, the function named `receive_pointer` does not return a value. The argument passed to it is declared as a pointer. The header of the C/C++ function is as follows:

```
void receive_pointer(*pointer_receiver);
```

A pointer is passed by value to the function so the parameter is a pointer of type `void`, a generic pointer. In this example, it is not required to know the type of data that it points to.

Example 4

Consider the following C/C++ function declared in the Verilog source code:

```
extern void memory_rewriter (input bit [1:0] array [1:0]
                           mem2, output bit [1:0] array [1:0] mem1);
```

In this example, the function named `memory_rewriter` has two arguments, one declared as an input, the other as an output. Both arguments are bit memories. The header of the C/C++ function is as follows:

```
void memory_rewriter(UB *out[2], *in[2]);
```

Memories are always passed by reference to a C/C++ function so the parameter named `in` is a pointer of type `UB` with the size that matched the memory range. The parameter named `out` is also a pointer, because its corresponding argument is declared to be output. Its type is also `UB` because it outputs to a Verilog memory.

Example 5

Consider the following C/C++ function declared in the Verilog source code:

```
extern void incr (inout bit [] r7);
```

In this example, the function named `incr`, that does not return a value, has an argument declared as `inout`. No bit-width is specified, but the `[]` entry for the argument specifies that it is not a scalar bit. The header of the C/C++ function is as follows:

```
void incr (U *p);
```

Open bit-width parameters are always passed to by reference. A parameter whose corresponding argument is declared to be `inout` is passed to and from by reference. So there are two reasons for parameter `p` to be a pointer. It is a pointer to type `U` because its corresponding argument is a vector bit.

Example 6

Consider the following C/C++ function declared in the Verilog source code:

```
extern void passbig1 (input bit [63:0] r8,
                      output bit [63:0] r9);
```

In this example, the function named `passbig1`, that does not return a value, has input and output arguments declared as `bit` and larger than 32 bits. The header of the C/C++ function is as follows:

```
void passbig (U *in, U *out)
```

In this example, the parameters `in` and `out` are pointers to type `U`. They are pointers because their corresponding arguments are larger than 32 bits and type `U` because their corresponding arguments are type `bit`.

Example 7

Consider the following C/C++ function declared in the Verilog source code:

```
extern void passbig2 (input reg [63:0] r10,
                      output reg [63:0] r11);
```

In this example, the function named `passbig2`, that does not return a value, has input and output arguments declared as non-scalar `reg`. The header of the C/C++ function is as follows:

```
void passbig2(vec32 *in, vec32 *out)
```

In this example, the parameters `in` and `out` are pointers to type `vec32`. They are pointers because their corresponding arguments are non-scalar type `reg`.

Example 8

Consider the following C/C++ function declared in the Verilog source code:

```
extern void reality (input real real1, output real real2);
```

In this example, the function named `reality`, that does not return a value, has `input` and `output` arguments of declared type `real`. The header of the C/C++ function is as follows:

```
void reality (double *in, double *out)
```

In this example, the parameters `in` and `out` are pointers to type `double` because their corresponding arguments are type `real`.

Using the vc_hdrs.h File

When you compile/elaborate your design for DirectC (by including the `+vc` elaboration/compile-time option), VCS writes a file in the current directory named `vc_hdrs.h`. In this file, there are `extern` declarations for all the C/C++ functions that you declared in your Verilog code. For example, if you compile/elaborate the Verilog code that contains all the C/C++ declarations in the examples in this section, the `vc_hdrs.h` file contains the following `extern` declarations:

```
extern void memory_rewriter(UB* mem2, /*OUT*/UB* mem1);
extern U return_vector_bit(U r3);
extern void receive_pointer(void* r6);
extern void incr(/*INOUT*/U* r7);
extern void* return_pointer();
extern scalar return_reg(scalar r1);
extern void reality(double* real1, /*OUT*/double* real2);
extern void receive_string(char* r5);
extern void passbig2(vec32* r8, /*OUT*/vec32* r9);
extern char* return_string();
extern void passbig1(U* r8, /*OUT*/U* r9);
```

These declarations contain the `/*OUT*/` comment in the parameter specification if its corresponding argument in your Verilog code is of type `output` in the declaration of the function.

These declarations contain the `/*INOUT*/` comment in the parameter specification if its corresponding argument in your Verilog code is of type `inout` in the declaration of the function.

You can copy from these `extern` declarations to the function headers in your C code. If you copy from these declarations, you will always use the right type of parameter in your function header and you do not have to learn the rules for direct access. However, it is recommended to copy the `extern` declaration by VCS.

Access Routines for Multi-Dimensional Arrays

DirectC requires that Verilog multi-dimensional arrays be linearized (turned into arrays of the same size, but with only one dimension). VCS provides routines for obtaining information about Verilog multi-dimensional arrays when using direct access. This section describes these routines.

UB *vc_arrayElemRef(UB*, U, ...)

The `UB*` parameter points to an array, either a single dimensional array or a multi-dimensional array, and the `U` parameters specify indices in the multi-dimensional array. This routine returns a pointer to an element of the array or `NULL` if the indices are outside the range of the array or there is a null pointer.

```
U dgetelem(UB *mem_ptr, int i, int j) {
    int indx;
    U k;
    /* remaining indices are constant */
    UB *p = vc_arrayElemRef(mem_ptr,i,j,0,1);
    k = *p;
    return(k);
}
```

There are specialized versions of this routine for one-dimensional, two-dimensional, and three-dimensional arrays:

```
UB *vc_array1ElemRef(UB*, U)
UB *vc_array2ElemRef(UB*, U, U)
UB *vc_array3ElemRef(UB*, U, U, U)
```

U vc_getSize(UB*,U)

This routine is similar to the `vc_mdaSize()` routine used in abstract access. It returns the following values:

- If the `U` type parameter has a value of 0, it returns the number of indices in an array.
- If the `U` type parameter has a value greater than 0, it returns the number of values in the index specified by the parameter. There is an error condition if this parameter is out of the range of indices.

If the `UB` pointer is null, this routine returns 0.

Using Abstract Access

In abstract access, VCS creates a descriptor for each argument in a function call. The corresponding formal parameters in the function uses a specially defined pointer to these descriptors called `vc_handle`. In abstract access, you use these “handles” to pass data and values by reference to and from these descriptors.

Abstract access is useful when do not have to worry about the type you use for parameters, because you always use a special pointer type called `vc_handle`.

In abstract access, VCS creates a descriptor for every argument that you enter in the function call in your Verilog code. The `vc_handle` is a pointer to the descriptor for the argument. It is defined as follows:

```
typedef struct VeriC_Descriptor *vc_handle;
```

Using `vc_handle`

In the function header, the `vc_handle` for a Verilog reg, bit, or memory is based on the order that you declare the `vc_handle` and the order that you entered its corresponding reg, bit, or memory in the function call in your Verilog code. For example, you could have declared the function and called it in your Verilog code as follows:

```
extern "A" void my_function( input bit [31:0] r1,  
                           input bit [32:0] r2);  
  
module dev1;  
  reg [31:0] bit1;  
  reg [32:0] bit2;  
  initial  
  begin
```

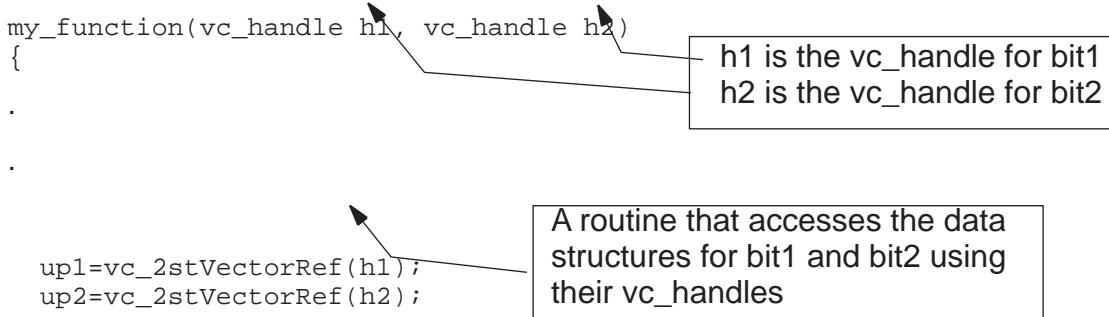
Declare the function

```
    .  
    .  
    .  
    my_function(bit1,bit2);  
    .  
    .  
    .
```

Enter first bit1 then bit2 as arguments
in the function call

This is using abstract access, so VCS created descriptors for `bit1` and `bit2`. These descriptors contain information about their value, but also other information such as whether they are scalar or vector, and whether they are simulating in two-state or four-state simulation.

The corresponding header for the C/C++ function is as follows:



After declaring the `vc_handles`, you can use them to pass data to and from these descriptors.

Using Access Routines

Abstract access provides a set of access routines that enable your C/C++ function to pass values to and from the descriptors for the Verilog reg, bit, and memory arguments in the function call.

These access routines use the `vc_handle` to pass values by reference, but the `vc_handle` is not the only type of parameter for many of these routines. These routines also have the following types of parameters:

- Scalar — an unsigned char
- Integers — uninterpreted 32 bits with no implied semantics
- Other types of pointers — primitive types “string” and “pointer”
- Real numbers

The naming convention used for access routines indicate their function. Routine names beginning with `vc_get` are for retrieving data from the descriptor for the Verilog parameter. Routine names beginning with `vc_put` are for passing new values to these descriptors.

These routines can convert Verilog representation of simulation values and strings to string representation in C/C++. Strings can also be created in a C/C++ function and passed to Verilog, however, you must make sure that they can be overwritten in Verilog. Therefore, copy them to local buffers if you want them to persist.

The following are the access routines, their parameters, and return values, and examples of how they are used. For the summary of access routines, see [Summary of Access Routines](#).

int vc_isScalar(vc_handle)

Returns a 1 value if the `vc_handle` is for a one-bit reg or bit; returns a 0 value for a vector reg or bit or any memory including memories with scalar elements. For example:

```
extern "A" void scalarfinder(input reg r1,
                             input reg [1:0] r2,
                             input reg [1:0] array [1:0] r3,
                             input reg array [1:0] r4);

module top;
reg r1;
reg [1:0] r2;
reg [1:0] r3 [1:0];
reg r4 [1:0];
initial
scalarfinder(r1,r2,r3,r4);
endmodule
```

In this example, a routine named `scalarfinder` and input a scalar reg, a vector reg and two memories (one with scalar elements) are declared.

The declaration contains the "A" specification for abstract access. You typically include it in the declaration when other functions use direct access, that is, you have a mix of functions with direct and abstract access.

```
#include <stdio.h>
#include "DirectC.h"

scalarfinder(vc_handle h1, vc_handle h2, vc_handle h3, vc_handle h4)
{
    int i1 = vc_isScalar(h1),
        i2 = vc_isScalar(h2),
        i3 = vc_isScalar(h3),
        i4 = vc_isScalar(h4);
    printf("\n i1=%d i2=%d i3=%d i4=%d\n\n", i1, i2, i3, i4);
}
```

Parameters `h1`, `h2`, `h3`, and `h4` are `vc_handles` to regs `r1` and `r2` and memories `r3` and `r4`, respectively. The function displays the following:

i1=1 i2=0 i3=0 i4=0

int vc_isVector(vc_handle)

This routine returns a 1 value if the `vc_handle` is to a vector reg or bit. It returns a 0 value for a vector bit or reg or any memory. For example, using the Verilog code from the previous example, and the following C/C++ function:

```
scalarfinder(vc_handle h1, vc_handle h2, vc_handle h3, vc_handle h4)
{
    int i1 = vc_isVector(h1),
        i2 = vc_isVector(h2),
```

```
i3 = vc_isVector(h3),
i4 = vc_isVector(h4);
printf("\n i1=%d i2=%d i3=%d i4=%d\n\n",i1,i2,i3,i4);
```

The function displays the following:

```
i1=0 i2=1 i3=0 i4=0
```

int vc_isMemory(vc_handle)

This routine returns a 1 value if the `vc_handle` is to a memory. It returns a 0 value for a bit or reg that is not a memory. For example, using the Verilog code from the previous example and the following C/C++ function:

```
#include <stdio.h>
#include "DirectC.h"

scalarfinder(vc_handle h1, vc_handle h2, vc_handle h3, vc_handle h4)
{
    int i1 = vc_isMemory(h1),
        i2 = vc_isMemory(h2),
        i3 = vc_isMemory(h3),
        i4 = vc_isMemory(h4);
    printf("\n i1=%d i2=%d i3=%d i4=%d\n\n",i1,i2,i3,i4);
}
```

The function displays the following:

```
i1=0 i2=0 i3=1 i4=1
```

int vc_is4state(vc_handle)

This routine returns a 1 value if the `vc_handle` is to a reg or memory that simulates with four states. It returns a 0 value for a bit or a memory that simulates with two states. For example, the following Verilog code uses metacomments to specify four- and two-state simulation:

```
extern void statefinder (input reg r1,
                        input reg [1:0] r2,
                        input reg [1:0] array [1:0] r3,
                        input reg array [1:0] r4,
                        input bit r5,
                        input bit [1:0] r6,
                        input bit [1:0] array [1:0] r7,
                        input bit array [1:0] r8);

module top;
reg /*4value*/ r1;
reg /*4value*/ [1:0] r2;
reg /*4value*/ [1:0] r3 [1:0];
reg /*4value*/ r4 [1:0];
reg /*2value*/ r5;
reg /*2value*/ [1:0] r6;
```

```
reg /*2value*/ [1:0] r7 [1:0];
reg /*2value*/ r8 [1:0];
initial
statefinder(r1,r2,r3,r4,r5,r6,r7,r8);
endmodule
```

The C/C++ function that calls the `vc_is4state` routine is as follows:

```
#include <stdio.h>
#include "DirectC.h"

statefinder(vc_handle h1, vc_handle h2, vc_handle h3,
            vc_handle h4, vc_handle h5, vc_handle h6,
            vc_handle h7, vc_handle h8)
{
    printf("\nThe vc_handles to 4state are:");
    printf("\nh1=%d h2=%d h3=%d h4=%d\n\n",
           vc_is4state(h1), vc_is4state(h2),
           vc_is4state(h3), vc_is4state(h4));
    printf("\nThe vc_handles to 2state are:");
    printf("\nh5=%d h6=%d h7=%d h8=%d\n\n",
           vc_is4state(h5), vc_is4state(h6),
           vc_is4state(h7), vc_is4state(h8));
}
```

The function prints the following:

```
The vc_handles to 4state are:
h1=1 h2=1 h3=1 h4=1
```

```
The vc_handles to 2state are:
h5=0 h6=0 h7=0 h8=0
```

int vc_is2state(vc_handle)

This routine does the opposite of the `vc_is4state` routine. For example, using the Verilog code from the previous example and the following C/C++ function:

```
#include <stdio.h>
#include "DirectC.h"

statefinder(vc_handle h1, vc_handle h2, vc_handle h3,
            vc_handle h4, vc_handle h5, vc_handle h6,
            vc_handle h7, vc_handle h8)
{
    printf("\nThe vc_handles to 4state are:");
    printf("\nh1=%d h2=%d h3=%d h4=%d\n\n",
           vc_is2state(h1), vc_is2state(h2),
           vc_is2state(h3), vc_is2state(h4));
    printf("\nThe vc_handles to 2state are:");
    printf("\nh5=%d h6=%d h7=%d h8=%d\n\n",
           vc_is2state(h5), vc_is2state(h6),
```

```
    vc_is2state(h7), vc_is2state(h8));
}
```

The function displays the following:

The vc_handles to 4state are:
h1=0 h2=0 h3=0 h4=0

The vc_handles to 2state are:
h5=1 h6=1 h7=1 h8=1

int vc_is4stVector(vc_handle)

This routine returns a 1 value if the vc_handle is to a vector reg. It returns a 0 value if the vc_handle is to a scalar reg, scalar or vector bit, or memory. For example, using the Verilog code from the previous example, and the following C/C++ function:

```
#include <stdio.h>
#include "DirectC.h"

statefinder(vc_handle h1, vc_handle h2,
            vc_handle h3, vc_handle h4,
            vc_handle h5, vc_handle h6,
            vc_handle h7, vc_handle h8)
{
    printf("\nThe vc_handle to a 4state Vector is:");
    printf("\nh2=%d \n\n", vc_is4stVector(h2));
    printf("\nThe vc_handles to 4state scalars or
          memories and 2state are:");
    printf("\nh1=%d h3=%d h4=%d h5=%d h6=%d h7=%d h8=%d\n\n",
           vc_is4stVector(h1), vc_is4stVector(h3),
           vc_is4stVector(h4), vc_is4stVector(h5),
           vc_is4stVector(h6), vc_is4stVector(h7),
           vc_is4stVector(h8));
}
```

The function displays the following:

The vc_handle to a 4state Vector is:
h2=1

The vc_handles to 4state scalars or
memories and 2state are:
h1=0 h3=0 h4=0 h5=0 h6=0 h7=0 h8=0

int vc_is2stVector(vc_handle)

This routine returns a 1 value if the vc_handle is to a vector bit. It returns a 0 value if the vc_handle is to a scalar bit, scalar or vector reg, or to a memory. For example, using the Verilog code from the previous example and the following C/C++ function:

```
#include <stdio.h>
#include "DirectC.h"
```

```

statefinder(vc_handle h1, vc_handle h2,
            vc_handle h3, vc_handle h4,
            vc_handle h5, vc_handle h6,
            vc_handle h7, vc_handle h8)
{
    printf("\nThe vc_handle to a 2state Vector is:");
    printf("\nh6=%d \n\n", vc_is2stVector(h6));
    printf("\nThe vc_handles to 2state scalars or
           memories and 4state are:");
    printf("\nh1=%d h2=%d h3=%d h4=%d h5=%d h7=%d h8=%d\n\n",
           vc_is2stVector(h1), vc_is2stVector(h2),
           vc_is2stVector(h3), vc_is2stVector(h4),
           vc_is2stVector(h5), vc_is2stVector(h7),
           vc_is2stVector(h8));
}

```

The function displays the following:

```
The vc_handle to a 2state Vector is:
h6=1
```

```
The vc_handles to 2state scalars or
           memories and 4state are:
h1=0 h2=0 h3=0 h4=0 h5=0 h7=0 h8=0
```

int vc_width(vc_handle)

Returns the width of a vc_handle. For example:

```

void memcheck_int(vc_handle h)
{
    int i;

    int mem_size = vc_arraySize(h);

    /* determine minimal needed width, assuming signed int */
    for (i=0; (1 << i) < (mem_size-1); i++) ;

    if (vc_width(h) < (i+1)) {
        printf("Register too narrow to be assigned %d\n", (mem_size-1));
        return;
    }

    for(i=0;i<8;i++) {
        vc_putMemoryInteger(h,i,i*4);
        printf("memput : %d\n",i*4);
    }
    for(i=0;i<8;i++) {
        printf("memget:: %d \n",vc_getMemoryInteger(h,i));
    }
}

```

int vc_arraySize(vc_handle)

Returns the number of elements in a memory or multi-dimensional array. The previous example also shows the usage of `vc_arraySize()`.

scalar vc_getScalar(vc_handle)

Returns the value of a scalar reg or bit. For example:

```
void rotate_scalars(vc_handle h1, vc_handle h2, vc_handle h3)
{
    scalar a;

    a = vc_getScalar(h1);
    vc_putScalar(h1, vc_getScalar(h2));
    vc_putScalar(h2, vc_getScalar(h3));
    vc_putScalar(h3, a);
    return;
}
```

void vc_putScalar(vc_handle, scalar)

Passes the value of a scalar reg or bit to a `vc_handle` by reference. The previous example also shows the usage of `vc_putScalar()`.

char vc_toChar(vc_handle)

Returns the 0, 1, x, or z character. For example:

```
void print_scalar(vc_handle h) {
    printf("%c", vc_toChar(h));
    return;
}
```

int vc_toInteger(vc_handle)

Returns an int value for a `vc_handle` to a scalar bit or a vector bit of 32 bits or less. For a vector reg or a vector bit with more than 32 bits this routine returns a 0 value and displays the following warning message:

DirectC interface warning: 0 returned for 4-state value (`vc_toInteger`)

The following is an example of Verilog code that calls a C/C++ function that uses this routine:

```
extern void rout1 (input bit onebit, input bit [7:0] mobits);

module top;
reg /*2value*/ onebit;
reg /*2value*/ [7:0] mobits;
initial
begin
rout1(onebit,mobits);
onebit=1;
```

```
mobits=128;
rout1(onebit,mobits);
end
endmodule
```

Note that the function declaration specifies that the parameters are of type bit. It includes metacomments for two-state simulation in the declaration of reg onebit and mobits. There are two calls to the function rout1, before and after values are assigned in this Verilog code.

The following C/C++ function uses this routine:

```
#include <stdio.h>
#include "DirectC.h"

void rout1 (vc_handle onebit, vc_handle mobits)
{
printf("\n\nonebit is %d mobits is %d\n\n",
      vc_toInteger(onebit), vc_toInteger(mobits));
}
```

This function displays the following:

```
onebit is 0 mobits is 0
```

```
onebit is 1 mobits is 128
```

char *vc_toString(vc_handle)

Returns a string that contains the 1, 0, x, and z characters. For example:

```
extern void vector_printer (input reg [7:0] r1);

module test;
reg [7:0] r1,r2;

initial
begin
#5 r1 = 8'bzx01zx01;
#5 vector_printer(r1);
#5 $finish;
end
endmodule

void vector_printer (vc_handle h)
{
vec32 b,*c;
c=vc_4stVectorRef(h);
b=*c;
printf("\n b is %x[control] %x[data]\n\n",b.c,b.d);
```

```
printf("\n b is %s \n\n",vc_toString(h));
}
```

In this example, a vector reg is assigned a value that contains x and z values, as well as, 1 and 0 values. In the abstract access C/C++ function, there are two ways of displaying the value of the reg:

- Recognize that type `vec32` is defined as follows in the `DirectC.h` file:

```
typedef struct {U c; U d;} vec32;
```

In machine memory, there are control, as well as, data bits for Verilog data to differentiate X from 1 and Z from 0 data, so there are `c` (control) and `d` (data) data variables in the structure and you must specify which variable when you access the `vec32` type.

- Use the `vc_toString` routine to display the value of the reg that contains X and Z values.

This example displays:

```
b is cc[control] 55[data]
```

```
b is zx01zx01
```

char *vc_toStringF(vc_handle, char)

Returns a string that contains the 1, 0, x, and z characters and allows you to specify the format or radix for the display. The `char` parameter can be '`b`', '`o`', '`d`', or '`x`'.

So, if you modify the C/C++ function in the previous example, it is as follows:

```
void vector_printer (vc_handle h)
{
vec32 b,*c;
c=vc_4stVectorRef(h);
b=*c;
printf("\n b is %s \n\n",vc_toStringF(h,'b'));
printf("\n b is %s \n\n",vc_toStringF(h,'o'));
printf("\n b is %s \n\n",vc_toStringF(h,'d'));
printf("\n b is %s \n\n",vc_toStringF(h,'x'));
}
```

This example now displays:

```
b is zx01zx01
```

```
b is XZX
```

```
b is X
```

```
b is XX
```

void vc_putReal(vc_handle, double)

Passes by reference a real (double) value to a `vc_handle`. For example:

```
void get_PI(vc_handle h)
{
    vc_putReal(h, 3.14159265);
}
```

double vc_getReal(vc_handle)

Returns a real (double) value from a `vc_handle`. For example:

```
void print_real(vc_handle h)
{
    printf("[print_real] %f\n", vc_getReal(h));
}
```

void vc_putValue(vc_handle, char *)

This function passes, by reference, through the `vc_handle`, a value represented as a string containing the 0, 1, x, and z characters. For example:

```
extern void check_vc_putvalue(output reg [] r1);

module tester;
reg [31:0] r1;

initial
begin
check_vc_putvalue(r1);
$display("r1=%0b",r1);
$finish;
end
endmodule
```

In this example, the C/C++ function is declared in the Verilog code specifying that the function passes a value to a four-state reg (and, therefore, can hold X and Z values).

```
#include <stdio.h>
#include "DirectC.h"

void check_vc_putvalue(vc_handle h)
{
    vc_putValue(h,"10xz");
}
```

The `vc_putValue` routine passes the string "10xz" to the reg `r1` through the `vc_handle`.
The Verilog code displays:

`r1=10xz`

void vc_putValueF(vc_handle, char *, char)

This function passes by reference, through the `vc_handle`, a value for which you specify a radix with the third parameter. The valid radices are '`b`', '`o`', '`d`', and '`x`'. For example the following Verilog code declares a function named `assigner` that uses this routine:

```
extern void assigner (output reg [31:0] r1,
                      output reg [31:0] r2,
                      output reg [31:0] r3,
                      output reg [31:0] r4);

module test;
reg [31:0] r1,r2,r3,r4;
initial
begin
assigner(r1,r2,r3,r4);
$display("r1=%0b in binary r1=%0d in decimal\n",r1,r1);
$display("r2=%0o in octal r2 =%0d in decimal\n",r2,r2);
$display("r3=%0d in decimal r3=%0b in binary\n",r3,r3);
$display("r4=%0h in hex r4= %0d in decimal\n\n",r4,r4);
$finish;
end
endmodule
```

The following is the C/C++ function:

```
#include <stdio.h>
#include "DirectC.h"

void assigner (vc_handle h1, vc_handle h2, vc_handle h3, vc_handle h4)
{
    vc_putValueF(h1,"10",'b');
    vc_putValueF(h2,"11",'o');
    vc_putValueF(h3,"10",'d');
    vc_putValueF(h4,"aff",'x');
}
```

The Verilog code displays the following:

```
r1=10 in binary r1=2 in decimal
r2=11 in octal r2 =9 in decimal
r3=10 in decimal r3=1010 in binary
r4=aff in hex r4= 2815 in decimal
```

void vc_putPointer(vc_handle, void*) void *vc_getPointer(vc_handle)

These functions pass a generic type of pointer or string to a `vc_handle` by reference. Do not use these functions for passing Verilog data (the values of Verilog signals). Use them for passing C/C++ data instead. `vc_putPointer` passes this data by reference to Verilog and `vc_getPointer` receives this data in a pass by reference from Verilog. You can also use these functions for passing Verilog strings.

For example:

```
extern void passback(output string, input string);
extern void printer(input pointer);

module top;
reg [31:0] r2;
initial
begin
passback(r2,"abc");
printer(r2);
end
endmodule
```

This Verilog code passes the string `abc` to the `passback` C/C++ function by reference, and that function passes it by reference to reg `r2`. The Verilog code then passes it by reference to the C/C++ function `printer` from reg `r2`.

```
passback(vc_handle h1, vc_handle h2)
{
vc_putPointer(h1, vc_getPointer(h2));
}

printer(vc_handle h)
{
printf("Procedure printer prints the string value %s\n\n",
      vc_getPointer (h));
}
```

The function named `printer` prints the following:

```
Procedure printer prints the string value abc
```

void vc_StringToVector(char *, vc_handle)

Converts a C string (a pointer to a sequence of ASCII characters terminated with a null character) into a Verilog string (a vector with 8-bit groups representing characters). For example:

```
extern "C" stringFullPath(string filename);
// find full path to the file
// C string obtained from C domain

extern "A" void s2v(string, output reg[]);
```

```
// string-to-vector
// wrapper for vc_StringToVector().

`define FILE_NAME_SIZE 512

module Test;
    reg [`FILE_NAME_SIZE*8:1] file_name;
    // this file_name will be passed to the Verilog code that expects
    // a Verilog-like string
    .
    .
    .
    initial begin
        s2vFullPath("myStimulusFile"), file_name); // C-string to Verilog-string
        // bits of 'file_name' represent now 'Verilog string'
    end
    .
    .
    .
endmodule
```

The C code is as follows:

```
void s2v(vc_handle hs, vc_handle hv) {
    vc_StringToVector((char *)vc_getPointer(hs), hv);
}
```

void vc_VectorToString(vc_handle, char *)

Converts a vector value to a string value.

int vc_getInteger(vc_handle)

Same as vc_toInteger.

void vc_putInteger(vc_handle, int)

Passes an int value by reference through a vc_handle to a scalar reg or bit or a vector bit that is 32 bits or less. For example:

```
void putter (vc_handle h1, vc_handle h2, vc_handle h3, vc_handle h4)
{
    int a,b,c,d;
    a=1;
    b=2;
    c=3;
    d=9999999;

    vc_putInteger(h1,a);
    vc_putInteger(h2,b);
    vc_putInteger(h3,c);
    vc_putInteger(h4,d);
}
```

vec32 *vc_4stVectorRef(vc_handle)

Returns a `vec32` pointer to a four-state vector. Returns NULL if the specified `vc_handle` is not to a four-state vector reg. For example:

```
typedef struct vector_descriptor {
    int width; /* number of bits */
    int is4stte; /* TRUE/FALSE */
} VD;

void WriteVector(vc_handle file_handle, vc_handle a_vector) {
    FILE *fp;
    int n, size;
    vec32 *v;
    VD vd;
    fp = vc_getPointer(file_handle);

    /* write vector's size and type */
    vd.is4state = vc_is4stVector(a_vector);
    vd.width = vc_width(a_vector);
    size = (vd.width + 31) >> 5; /* number of 32-bit chunks */
    /* printf("writing: %d bits, is 4 state: %d, #chunks: %d\n",
       vd.width, vd.is4state, size); */
    n = fwrite(&vd, sizeof(vd), 1, fp);
    if (n != 1) {
        printf("Error: write failed.\n");
    }

    /* write the vector into a file; vc_*stVectorRef
       is a pointer to the actual Verilog vector */
    if (vc_is4stVector(a_vector)) {
        n = fwrite(vc_4stVectorRef(a_vector), sizeof(vec32),
                   size, fp);
    } else {
        n = fwrite(vc_2stVectorRef(a_vector), sizeof(U),
                   size, fp);
    }
    if (n != size) {
        printf("Error: write failed for vector.\n");
    }
}
```

U *vc_2stVectorRef(vc_handle)

Returns a U pointer to a bit vector that is larger than 32 bits. If you specify a short bit vector (32 bits or fewer) this routine returns a NULL value. For example:

```
extern void big_2state( input bit [31:0] r1,
                        input bit [32:0] r2);

module test;
reg [31:0] r1;
reg [32:0] r2;
```

In this example, the Verilog code declares a 32-bit vector bit, `r1`, and a 33-bit vector bit, `r2`. The values of both are passed to the C/C++ function `big_2state`.

When you pass the short bit vector `r1` to `vc_2stVectorRef`, it returns a null value because it has fewer than 33 bits. This is not the case when you pass bit vector `r2` because it has more than 32 bits. Note that from right to left, the first 32 bits of `r2` have a value of 2 and the MSB 33rd bit has a value of 1. This is significant in how the C/C++ stores this data.

```

#include <stdio.h>
#include "DirectC.h"

big_2state(vc_handle h1, vc_handle h2)
{
    U u1,*up1,u2,*up2;
    int i;
    int size;

    up1=vc_2stVectorRef(h1);
    up2=vc_2stVectorRef(h2);
    if (up1){ /* check for the null value returned to up1 */
        u1=*up1;} else{
        u1=0;
        printf("\nShort 2 state vector passed to up1\n");
    }
    if (up2){ /* check for the null value returned to up2 */
        size = vc_width (h2); /* to find out the number of bits */
                                /* in h2 */
        printf("\n width of h2 is %d\n",size);
        size = (size + 31) >> 5; /* to get number of 32-bit chunks */
    printf("\n the number of chunks needed for h2 is %d\n\n",
          size);
        printf("loading into u2");
        for(i = size - 1; i >= 0; i--) {
            u2=up2[i]; /* load a chunk of the vector */
            printf(" %x",up2[i]);}
        printf("\n");}
    else{
        u2=0;
        printf("\nShort 2 state vector passed to up2\n");}
}

```

In this example, the short bit vector is passed to the `vc_2stVectorRef` routine, so it returns a null value to pointer `up1`. Then the long bit vector is passed to the

`vc_2stVectorRef` routine, so it returns a pointer to the Verilog data for vector bit `r2` to pointer `up2`.

It checks for the null value in `up1`. If it does not have a null value, whatever it points to is passed to `u1`. If it does have a null value, the function prints a message about the short bit vector. In this example, you can expect it to display this message.

Later in the function, it checks for the null value in `up2` and the size of the long bit vector that is passed to the second parameter. Because Verilog values are stored in 32-bit chunks in C/C++, the function finds out how many chunks are needed to store the long bit vector. It then loads one chunk at a time into `u2` and prints the chunk starting with the most significant bits. This function displays the following:

```
Short 2 state vector passed to up1
width of h2 is 33
the number of chunks needed for h2 is 2
loading into u2 1 2
```

void vc_get4stVector(vc_handle, vec32 *) void vc_put4stVector(vc_handle, vec32 *)

Passes a four-state vector by reference to a `vc_handle` to and from an array in C/C++ function. `vc_get4stVector` receives the vector from Verilog and passes it to the array and `vc_put4stVector` passes the array to Verilog.

These routines work only if there are sufficient elements in the array for all the bits in the vector. The array must have an element for every 32 bit in the vector plus an additional element for any remaining bits. For example:

```
extern void copier (input reg [67:0] r1,
                    output reg [67:0] r2);

module top;

reg [67:0] r1,r2;

initial
begin
    r1 [67:65] = 3'b111;
    r1 [64:33] = 32'bzzzzzzzzzzzzzzzzzzzzzzzzzzzzzz;
    r1 [32:0] = 32'b00000000000000000000000000000000;
    copier(r1,r2);
    $display("r1=%0b\n",r1);
    $display("r2=%0b\n",r2);

end
endmodule
```

In this example, there are two 68-bit regs. Values are assigned to all the bits of one reg and both of these regs are parameters to the C/C++ function named `copier`.

```
copier(vc_handle h1, vc_handle h2)
{
vec32 holder[3];
vc_get4stVector(h1,holder);
vc_put4stVector(h2,holder);
}
```

This function declares a `vec32` array of three elements named `holder`. It uses three elements because its parameters are 68-bit regs so you need an element for every 32 bits and one more for the remaining four bits.

The Verilog code displays the following:

void vc_get2stVector(vc_handle, U *) void vc_put2stVector(vc_handle, U *)

Passes a two-state vector by reference to a `vc_handle` to and from an array in C/C++ function. `vc_get2stVector` receives the vector from Verilog and passes it to the array and `vc_put4stVector` passes the array to Verilog.

There routines, such as the `vc_get4stVector` and `vc_put4stVector` routines, work only if there are sufficient elements in the array for all the bits in the vector. The array must have an element for every 32 bit in the vector plus an additional element for any remaining bits.

The differences between these routines and the `vc_get4stVector` and `vc_put4stVector` routines are the type of data they pass, two-state or four-state simulation values, and the type you declare for the array in the C/C++ function.

UB *vc_MemoryRef(vc_handle)

Returns a pointer of type UB that points to a memory in Verilog. For example:

```
extern void mem_doe ( input reg [1:0] array [3:0]
                      memory1, output reg [1:0] array
                      [31:0] memory2);

module top;
reg [1:0] memory1 [3:0];
reg [1:0] memory2 [31:0];
initial
begin
memory1 [3] = 2'b11;
memory1 [2] = 2'b10;
memory1 [1] = 2'b01;
memory1 [0] = 2'b00;
```

```
mem_doir(memory1,memory2);
$display("memory2[31]=%0d",memory2[31]);
end
endmodule
```

In this example, two memories, one with 4 addresses, `memory1`, the other with 32 addresses, `memory2` are declared. You assign values to the addresses of `memory1`, and then pass both memories to the C/C++ function `mem_doir`.

```
#include <stdio.h>
#include "DirectC.h"

void mem_doir(vc_handle h1, vc_handle h2)
{
    UB *p1, *p2;
    int i;

    p1 = vc_MemoryRef(h1);
    p2 = vc_MemoryRef(h2);

    for ( i = 0; i < 8; i++) {
        memcpy(p2,p1,8);
        p2 += 8;
    }
}
```

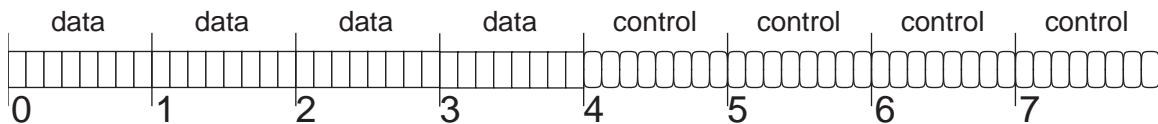
The purpose of the C/C++ function `mem_doir` is to copy the four elements in Verilog memory `memory1` into the 32 elements of `memory2`.

The `vc_MemoryRef` routines return pointers to the Verilog memories and the machine memory locations they point to are also pointed to by pointers `p1` and `p2`. Pointer `p1` points to the location of Verilog memory `memory1`, and `p2` points to the location of Verilog memory `memory2`.

The function uses a `for` loop to copy the data from Verilog memory `memory1` to Verilog memory `memory2`. It uses the standard `memcpy` function to copy a total of 64 bytes by copying eight bytes eight times.

This example copies a total of 64 bytes because each element of `memory2` is only two bits wide, however, for every eight bits in an element in machine memory there are two bytes, one for data and another for control. The bits in the control byte specify whether the data bit with a value of 0 is actually 0 or Z, or whether the data bit with a value of 1 is actually 1 or X.

Figure 174 Storing Verilog Memory Elements in Machine Memory



In an element in a Verilog memory, for each eight bits in the element there is a data byte and a control byte with an additional set of bytes for a remainder bit. Therefore, if a memory has 9 bits it would need two data bytes and two control bytes. If it has 17 bits it would need three data bytes and three control bytes. All the data bytes precede the control bytes.

Therefore, `memory1` needs 8 bytes of machine memory (four for data and four for control) and `memory2` needs 64 bytes of machine memory (32 for data and 32 for control). Therefore, the C/C++ function needs to copy 64 bytes.

The Verilog code displays the following:

```
memory2[31]=3
```

UB *vc_MemoryElemRef(vc_handle, U indx)

Returns a pointer to an element (word, address or index) of a Verilog memory. You specify the `vc_handle` of the memory and the element. For example:

```
extern void mem_elem_doir( inout reg [25:1] array [3:0] memory1);

module top;
reg [25:1] memory1 [3:0];
initial
begin
memory1 [0] = 25'bz00000000xxxxxxxx11111111;
$display("memory1 [0] = %0b\n", memory1[0]);
mem_add_doir(memory1);
$display("\nmemory1 [3] = %0b", memory1[3]);
end
endmodule
```

In this example, there is a Verilog memory with four addresses, each element has 25 bits. This means that the Verilog memory needs eight bytes of machine memory because there is a data byte and a control byte for every eight bits in an element, with an additional data and control byte for any remainder bits.

In this example, in element 0 the 25 bits are assigned from right to left, eight 1 bits, eight unknown x bits, eight 0 bits, and one high impedance z bit.

```
#include <stdio.h>
#include "DirectC.h"
```

```

void mem_elem_doe(vc_handle h)
{
    U indx;
    UB *p1, *p2, t [8];

    indx = 0;
    p1 = vc_MemoryElemRef(h, indx);
    indx = 3;
    p2 = vc_MemoryElemRef(h, indx);
    memcpy(p2,p1,8);

    memcpy(t,p2,8);
    printf(" %d from t[0], %d from t[1]\n",
           (int)t[0], (int)t[1]);
    printf(" %d from t[2], %d from t[3]\n",
           (int)t[2], (int)t[3]);
    printf(" %d from t[4], %d from t[5]\n",
           (int)t[4], (int)t[5]);
    printf(" %d from t[6], %d from t[7]\n",
           (int)t[6], (int)t[7]);
}

```

C/C++ function `mem_elem_doe` uses the `vc_MemoryElemRef` routine to return pointers to addresses 0 and 3 in Verilog `memory1` and pass them to UB pointers `p1` and `p2`. The standard `memcpy` routine then copies the eight bytes for address 0 to address 3.

The remainder of the function is additional code to show you data and control bytes. The eight bytes pointed to by `p2` are copied to array `t` and then the elements of the array are printed.

The combined Verilog and C/C++ code displays the following:

```

memory1 [0] = z00000000xxxxxxxx11111111
255 from t[0], 255 from t[1]
0 from t[2], 0 from t[3]
0 from t[4], 255 from t[5]
0 from t[6], 1 from t[7]

```

```

memory1 [3] = z00000000xxxxxxxx11111111

```

As you can see, function `mem_elem_doe` passes the contents of the Verilog memory `memory1` element 0 to element 3.

In array `t`, the elements contain the following:

-
- [0] The data bits for the eight 1 values assigned to the element.
 - [1] The data bits for the eight X values assigned to the element

- [2] The data bits for the eight 0 values assigned to the element
 - [3] The data bit for the Z value assigned to the element
 - [4] The control bits for the eight 1 values assigned to the element
 - [5] The control bits for the eight X values assigned to the element
 - [6] The control bits for the eight 0 values assigned to the element
 - [7] The control bit for the Z value assigned to the element
-

scalar vc_getMemoryScalar(vc_handle, U indx)

Returns the value of a one-bit memory element. For example:

```
extern void bitflipper (inout reg array [127:0] mem1);

module test;
reg mem1 [127:0];
initial
begin
mem1 [0] = 1;
$display("mem1[0]=%0d",mem1[0]);
bitflipper(mem1);
$display("mem1[0]=%0d",mem1[0]);
$finish;
end
endmodule
```

In this example of Verilog code, a memory with 128 one-bit elements, assign a value to element 0 is declared, and display its value before and after you call a C/C++ function named `bitflipper`.

```
#include <stdio.h>
#include "DirectC.h"

void bitflipper(vc_handle h)
{
scalar holder=vc_getMemoryScalar(h, 0);
holder = ! holder;
vc_putMemoryScalar(h, 0, holder);
}
```

In this example, a variable of type scalar, named `holder`, to hold the value of the one-bit Verilog memory element is declared. The routine `vc_getMemoryScalar` returns the value of the element to the variable. The value of `holder` is inverted and then the variable is included as a parameter in the `vc_putMemoryScalar` routine to pass the value to that element in the Verilog memory.

The Verilog code displays the following:

```
mem[0]=1
mem[0]=0
```

void vc_putMemoryScalar(vc_handle, U indx, scalar)

Passes a value of type scalar to a Verilog memory element. You specify the memory by `vc_handle` and the element by the `indx` parameter. This routine is used in the previous example.

int vc_getMemoryInteger(vc_handle, U indx)

Returns the integer equivalent of the data bits in a memory element whose bit-width is 32 bits or less. For example:

```
extern void mem_elem_halver (inout reg [] array [] memX);

module test;
reg [31:0] mem1 [127:0];
reg [7:0] mem2 [1:0];
initial
begin
mem1 [0] = 999;
mem2 [0] = 8'b1111xxxx;
$display("mem1[0]=%0d",mem1[0]);
$display("mem2[0]=%0d",mem2[0]);
mem_elem_halver(mem1);
mem_elem_halver(mem2);
$display("mem1[0]=%0d",mem1[0]);
$display("mem2[0]=%0d",mem2[0]);
$finish;
end
endmodule
```

In this example, when the C/C++ function is declared on your Verilog code it does not specify a bit-width or element range for the inout argument to the `mem_elem_halver` C/C++ function, because in the Verilog code you call the C/C++ function twice with a different memory each time and these memories have different bit widths and different element ranges.

Notice that you assign a value that included x values to the 0 element in memory `mem2`.

```
#include <stdio.h>
#include "DirectC.h"

void mem_elem_halver(vc_handle h)
{
int i =vc_getMemoryInteger(h, 0);
i = i/2;
```

```
vc_putMemoryInteger(h, 0, i);
}
```

This C/C++ function inputs the value of an element and then outputs half that value. The `vc_getMemoryInteger` routine returns the integer equivalent of the element you specify by `vc_handle` and index number to an int variable `i`. The function halves the value in `i`. Then the `vc_putMemoryInteger` routine passes the new value by value to the specified memory element.

The Verilog code displays the following before the C/C++ function is called twice with the different memories as the arguments:

```
mem1[0]=999
mem2[0]=X
```

Element `mem2[0]` has an X value because half of its binary value is x and the value is displayed with the `%d` format specification and, in this example, a partially unknown value is an unknown value. After the second call of the function, the Verilog code displays:

```
mem1[1]=499
mem2[0]=127
```

This occurs because before calling the function, `mem1[0]` has a value of 999, and after the call it has a value of 499, which is as close as it can get to half the value with integer values.

Before calling the function, `mem2[0]` has a value of 8'b1111xxxx, however, the data bits for the element would all be 1s (11111111). It is the control bits that specify 1 from x and this routine only deals with the data bits. Therefore, the `vc_getMemoryInteger` routine returns an integer value of 255 (the integer equivalent of the binary 11111111) to the C/C++ function, which is why the function outputs the integer value 127 to `mem2[0]`.

void vc_putMemoryInteger(vc_handle, U indx, int)

Passes an integer value to a memory element that is 32 bits or fewer. You specify the memory by `vc_handle` and the element by the `indx` argument. This routine is used in the previous example.

void vc_get4stMemoryVector(vc_handle, U indx, vec32 *)

Copies the value in an Verilog memory element to an element in an array. This routine copies both the data and control bytes. It copies them into an array of type `vec32` which is defined as follows:

```
typedef struct { U c; U d; } vec32;
```

Therefore, type `vec32` has two members, `c` and `d`, for control and data information. This routine always copies to the 0 element of the array. For example:

```
extern void mem_elem_copier (inout reg [] array [] memX);
```

```
module test;
reg [127:0] mem1 [127:0];
reg [7:0] mem2 [64:0];
initial
begin
mem1 [0] = 999;
mem2 [0] = 8'b0000000z;
$display("mem1[0]=%0d",mem1[0]);
$display("mem2[0]=%0d",mem2[0]);
mem_elem_copier(mem1);
mem_elem_copier(mem2);
$display("mem1[32]=%0d",mem1[32]);
$display("mem2[32]=%0d",mem2[32]);
$finish;
end
endmodule
```

In the Verilog code, a C/C++ function is declared that is called twice. Note the value assigned to `mem2[0]`. The C/C++ function copies the values to another element in the memory.

```
#include <stdio.h>
#include "DirectC.h"

void mem_elem_copier(vc_handle h)
{
vec32 holder[1];
vc_get4stMemoryVector(h,0,holder);
vc_put4stMemoryVector(h,32,holder);
printf(" holder[0].d is %d holder[0].c is %d\n\n",
       holder[0].d,holder[0].c);
}
```

This C/C++ function declares an array of type `vec32`. You must declare an array for this type, but as shown in this example, it is specified that it has only one element. The `vc_get4stMemoryVector` routine copies the data from the Verilog memory element (in this example, specified as the 0 element) to the 0 element of the `vec32` array. It always copies to the 0 element. The `vc_put4stMemoryVector` routine copies the data from the `vec32` array to the Verilog memory element (in this case, element 32).

The call to `printf` is to describe how the Verilog data is stored in element 0 of the `vec32` array.

The Verilog and C/C++ code display the following:

```
mem1[0]=999
mem2[0]=Z
holder[0].d is 999 holder[0].c is 0

holder[0].d is 768 holder[0].c is 1
```

```
mem1[32]=999
mem2[32]=z
```

As you can see, the function does copy the Verilog data from one element to another in both memories. When the function is copying the 999 value, the `c` (control) member has a value of 0; when it is copying the 8'b0000000z value, the `c` (control) member has a value of 1 because one of the control bits is 1, the remaining are 0.

void vc_put4stMemoryVector(vc_handle, U indx, vec32 *)

Copies Verilog data from a `vec32` array to a Verilog memory element. This routine is used in the previous example.

void vc_get2stMemoryVector(vc_handle, U indx, U *)

Copies the data bytes, but not the control bytes, from a Verilog memory element to an array in your C/C++ function. For example, if you use the Verilog code from the previous example, but simulate in two-state and use the following C/C++ code:

```
#include <stdio.h>
#include "DirectC.h"

void mem_elem_copier(vc_handle h)
{
    U holder[1];
    vc_get2stMemoryVector(h, 0, holder);
    vc_put2stMemoryVector(h, 32, holder);
}
```

The only difference here is that you declare the array to be of type `U` instead and you do not copy the control bytes, because there are none in two-state simulation.

void vc_put2stMemoryVector(vc_handle, U indx, U *)

Copies Verilog data from a `U` array to a Verilog memory element. This routine is used in the previous example.

void vc_putMemoryValue(vc_handle, U indx, char *)

This routine works similar to the `vc_putValue` routine except that is for passing values to a memory element instead of to a reg or bit. You enter an argument to specify the element (index) to which you want the routine to pass the value. For example:

```
#include <stdio.h>
#include "DirectC.h"

void check_vc_putvalue(vc_handle h)
{
    vc_putMemoryValue(h, 0, "10xz");
}
```

void vc_putMemoryValueF(vc_handle, U indx, char, char *)

This routine works similar to the `vc_putValueF` routine except that it is for passing values to a memory element instead of to a reg or bit. You enter an argument to specify the element (index) to which you want the routine to pass the value. For example:

```
#include <stdio.h>
#include "DirectC.h"

void assigner (vc_handle h1, vc_handle h2, vc_handle h3, vc_handle h4)
{
    vc_putMemoryValueF(h1, 0, "10", 'b');
    vc_putMemoryValueF(h2, 0, "11", 'o');
    vc_putMemoryValueF(h3, 0, "10", 'd');
    vc_putMemoryValueF(h4, 0, "aff", 'x');
}
```

char *vc_MemoryString(vc_handle, U indx)

This routine works similar to the `vc_toString` routine except that it used is for passing values to/from memory elements instead of to a reg or bit. You enter an argument to specify the element (index) whose value you want the routine to pass. For example:

```
extern void memcheck_vec(inout reg[] array[]);

module top;
reg [0:7] mem[0:7];
integer i;

initial
begin
    for(i=0;i<8;i=i+1) begin
        mem[i] = 8'b00000111;
        $display("Verilog code says \\"mem [%0d] = %0b\\",
                 i,mem[i]);
    end

    memcheck_vec(mem);
end

endmodule
```

The C/C++ function that calls `vc_MemoryString` is as follows:

```
#include <stdio.h>
#include "DirectC.h"

void memcheck_vec(vc_handle h)
{
    int i;
```

```

for(i= 0; i<8;i++) {
    printf("C/C++ code says \"mem [%d] is %s
          \n",i,vc_MemoryString(h,i));
}
}

```

The Verilog and C/C++ code display the following:

```

Verilog code says "mem [0] = 111"
Verilog code says "mem [1] = 111"
Verilog code says "mem [2] = 111"
Verilog code says "mem [3] = 111"
Verilog code says "mem [4] = 111"
Verilog code says "mem [5] = 111"
Verilog code says "mem [6] = 111"
Verilog code says "mem [7] = 111"
C/C++ code says "mem [0] is 00000111 "
C/C++ code says "mem [1] is 00000111 "
C/C++ code says "mem [2] is 00000111 "
C/C++ code says "mem [3] is 00000111 "
C/C++ code says "mem [4] is 00000111 "
C/C++ code says "mem [5] is 00000111 "
C/C++ code says "mem [6] is 00000111 "
C/C++ code says "mem [7] is 00000111 "

```

char *vc_MemoryStringF(vc_handle, U indx, char)

This routine works similar to the vc_MemoryString function except that you specify a radix with the third parameter. The valid radices are 'b', 'o', 'd', and 'x'. For example:

```

extern void memcheck_vec(inout reg[] array[]);

module top;
reg [0:7] mem[0:7];

initial begin
mem[0] = 8'b00000111;
$display("Verilog code says \"mem[0]=%0b radix b\"",mem[0]);
$display("Verilog code says \"mem[0]=%0o radix o\"",mem[0]);
$display("Verilog code says \"mem[0]=%0d radix d\"",mem[0]);
$display("Verilog code says \"mem[0]=%0h radix h\"",mem[0]);
memcheck_vec(mem);
end

endmodule

```

The C/C++ function that calls vc_MemoryStringF is as follows:

```

#include <stdio.h>
#include "DirectC.h"

void memcheck_vec(vc_handle h)
{

```

```

printf("C/C++ code says \"mem [0] is %s radix b\"\n",
       vc_MemoryStringF(h,0,'b'));
printf("C/C++ code says \"mem [0] is %s radix o\"\n",
       vc_MemoryStringF(h,0,'o'));
printf("C/C++ code says \"mem [0] is %s radix d\"\n",
       vc_MemoryStringF(h,0,'d'));
printf("C/C++ code says \"mem [0] is %s radix x\"\n",
       vc_MemoryStringF(h,0,'x'));
}

```

The Verilog and C/C++ code display the following:

```

Verilog code says "mem [0]=111 radix b"
Verilog code says "mem [0]=7 radix o"
Verilog code says "mem [0]=7 radix d"
Verilog code says "mem [0]=7 radix h"
C/C++ code says "mem [0] is 00000111 radix b"
C/C++ code says "mem [0] is 007 radix o"
C/C++ code says "mem [0] is 7 radix d"
C/C++ code says "mem [0] is 07 radix x"

```

void vc_FillWithScalar(vc_handle, scalar)

This routine fills all the bits or a reg, bit, or memory with all 1, 0, x, or z values (you can choose only one of these four values).

You specify the value with the scalar argument, which can be a variable of the scalar type. The scalar type is defined in the `DirectC.h` file as:

```
typedef unsigned char scalar;
```

You can also specify the value with integer arguments as follows:

0	Specifies 0 values
1	Specifies 1 values
2	Specifies z values
3	Specifies x values

If you declare a scalar type variable, enter it as the argument, and assign only the 0, 1, 2, or 3 integer values to it, they specify filling the Verilog reg, bit, or memory with the 0, 1, z, or x values.

You can use the following definitions from the `DirectC.h` file to specify these values:

```
#define scalar_0 0
#define scalar_1 1
```

```
#define scalar_z 2
#define scalar_x 3
```

The following Verilog and C/C++ code shows you how to use this routine to fill a reg and a memory using the following values:

```
extern void filler (inout reg [7:0] r1,
                    inout reg [7:0] array [1:0] r2,
                    inout reg [7:0] array [1:0] r3);

module top;
reg [7:0] r1;
reg [7:0] r2 [1:0];
reg [7:0] r3 [1:0];
initial
begin
$display("r1 is %0b",r1);
$display("r2[0] is %0b",r2[0]);
$display("r2[1] is %0b",r2[1]);
$display("r3[0] is %0b",r3[0]);
$display("r3[1] is %0b",r3[1]);
filler(r1,r2,r3);
$display("r1 is %0b",r1);
$display("r2[0] is %0b",r2[0]);
$display("r2[1] is %0b",r2[1]);
$display("r3[0] is %0b",r3[0]);
$display("r3[1] is %0b",r3[1]);
end
endmodule
```

The C/C++ code for the function is as follows:

```
#include <stdio.h>
#include "DirectC.h"

filler(vc_handle h1, vc_handle h2, vc_handle h3)
{
scalar s = 1;
vc_FillWithScalar(h1,s);
vc_FillWithScalar(h2,0);
vc_FillWithScalar(h3,scalar_z);
}
```

The Verilog code displays the following:

```
r1 is xxxxxxxx
r2[0] is xxxxxxxx
r2[1] is xxxxxxxx
r3[0] is xxxxxxxx
r3[1] is xxxxxxxx
r1 is 11111111
r2[0] is 0
r2[1] is 0
```

```
r3[0] is zzzzzzzz
r3[1] is zzzzzzzz
```

char *vc_argInfo(vc_handle)

Returns a string containing the information about the argument in the function call in your Verilog source code. For example, if you have the following Verilog source code:

```
extern void show(reg [] array []);
module tester;
reg [31:0] mem [7:0];
reg [31:0] mem2 [16:1];
reg [64:1] mem3 [32:1];
initial begin
    show(mem);
    show(mem2);
    show(mem3);
end
endmodule
```

Verilog memories `mem`, `mem2`, and `mem3` are all arguments to the function named `show`. If that function is defined as follows:

```
#include <stdio.h>
#include "DirectC.h"

void show(vc_handle h)
{
    printf("%s\n", vc_argInfo(h)); /* notice \n after the string */
}
```

This routine displays the following:

```
input reg[0:31] array[0:7]
input reg[0:31] array[0:15]
input reg[0:63] array[0:31]
```

int vc_Index(vc_handle, U, ...)

Internally, a multi-dimensional array is always stored as a one-dimensional array and this makes a difference in how it can be accessed. In order to avoid duplicating many of the previous access routines for multi-dimensional arrays, the access process is split into two steps. The first step, which this routine performs, is to translate the multiple indices into a single index of a linearized array. The second step is for another access routine to perform an access operation on the linearized array.

This routine returns the index of a linearized array or returns -1 if the U-type parameter is not an index of a multi-dimensional array or the `vc_handle` parameter is not a handle to a multi-dimensional array of the `reg` data type.

```
/* get the sum of all elements from a 2-dimensional slice
   of a 4-dimensional array */
```

```

int getSlice(vc_handle vh_array, vc_handle vh_idx1, vc_handle vh_idx2)
{
    int sum = 0;
    int i1, i2, i3, i4, indx;

    i1 = vc_getInteger(vh_idx1);
    i2 = vc_getInteger(vh_idx2);
    /* loop over all possible indices for that slice */
    for (i3 = 0; i3 < vc_mdaSize(vh_array, 3); i3++) {

        for (i4 = 0; i4 < vc_mdaSize(vh_array, 4); i4++) {

            indx = vc_Index(vh_array, i1, i2, i3, i4);
            sum += vc_getMemoryInteger(vh_array, indx);
        }
    }
    return sum;
}

```

There are specialized, more efficient versions for two-dimensional and three-dimensional arrays. They are as follows:

`int vc_Index2(vc_handle, U, U)`

Specialized version of `vc_Index()` where the two U parameters are the indices in a two-dimensional array.

`int vc_Index3(vc_handle, U, U, U)`

Specialized version of `vc_Index()` where the two U parameters are the indices in a three-dimensional array.

U vc_mdaSize(vc_handle, U)

Returns the following:

- If the U-type parameter has a value of 0, it returns the number of indices in the multi-dimensional array.
- If the U-type parameter has a value greater than 0, it returns the number of values in the index specified by the parameter. There is an error condition if this parameter is out of the range of indices.
- If the vc_handle parameter is not an array, it returns 0.

Summary of Access Routines

Table 59 summarizes all the access routines described in the previous section.

Table 59 Access Routine Description

<code>int vc_isScalar(vc_handle)</code>	Returns a 1 value if the <code>vc_handle</code> is for a one-bit reg or bit. It returns a 0 value for a vector reg or bit or any memory including memories with scalar elements.
<code>int vc_isVector(vc_handle)</code>	This routine returns a 1 value if the <code>vc_handle</code> is to a vector reg or bit. It returns a 0 value for a vector bit or reg or any memory.
<code>int vc_isMemory(vc_handle)</code>	This routine returns a 1 value if the <code>vc_handle</code> is to a memory. It returns a 0 value for a bit or reg that is not a memory.
<code>int vc_is4state(vc_handle)</code>	This routine returns a 1 value if the <code>vc_handle</code> is to a reg or memory that simulates with four states. It returns a 0 value for a bit or a memory that simulates with two states.
<code>int vc_is2state(vc_handle)</code>	This routine does the opposite of the <code>vc_is4state</code> routine.
<code>int vc_is4stVector(vc_handle)</code>	This routine returns a 1 value if the <code>vc_handle</code> is to a vector reg. It returns a 0 value if the <code>vc_handle</code> is to a scalar reg, scalar or vector bit, or to a memory.
<code>int vc_is2stVector(vc_handle)</code>	This routine returns a 1 value if the <code>vc_handle</code> is to a vector bit. It returns a 0 value if the <code>vc_handle</code> is to a scalar bit, scalar or vector reg, or to a memory.
<code>int vc_width(vc_handle)</code>	Returns the width of a <code>vc_handle</code> .
<code>int vc_arraySize(vc_handle)</code>	Returns the number of elements in a memory.
<code>scalar vc_getScalar(vc_handle)</code>	Returns the value of a scalar reg or bit.
<code>void vc_putScalar(vc_handle, scalar)</code>	Passes the value of a scalar reg or bit to a <code>vc_handle</code> by reference.
<code>char vc_toChar(vc_handle)</code>	Returns the 0, 1, x, or z character.
<code>int vc_toInteger(vc_handle)</code>	Returns an int value for a <code>vc_handle</code> to a scalar bit or a vector bit of 32 bits or less.
<code>char *vc_toString(vc_handle)</code>	Returns a string that contains the 1, 0, x, and z characters.
<code>char *vc_toStringF(vc_handle, char)</code>	Returns a string that contains the 1, 0, x, and z characters and allows you to specify the format or radix for the display. The char parameter can be 'b', 'o', 'd', or 'x'.

Table 59 Access Routine Description (Continued)

<code>void vc_putReal(vc_handle, double)</code>	Passes by reference a real (double) value to a <code>vc_handle</code> .
<code>double vc_getReal(vc_handle)</code>	Returns a real (double) value from a <code>vc_handle</code> .
<code>void vc_putValue(vc_handle, char *)</code>	This function passes, by reference through the <code>vc_handle</code> , a value represented as a string containing the 0, 1, x, and z characters.
<code>void vc_putValueF(vc_handle, char, char *)</code>	This function passes by reference through the <code>vc_handle</code> a value for which you specify a radix with the third parameter. The valid radices are 'b', 'o', 'd', and 'x'.
<code>void vc_putPointer(vc_handle, void*)void *vc_getPointer(vc_handle)</code>	These functions pass, by reference to a <code>vc_handle</code> , a generic type of pointer or string. Do not use these functions for passing Verilog data (the values of Verilog signals). Use it for passing C/C++ data. <code>vc_putPointer</code> passes this data by reference to Verilog and <code>vc_getPointer</code> receives this data in a pass by reference from Verilog. You can also use these functions for passing Verilog strings.
<code>void vc_StringToVector(char *, vc_handle)</code>	Converts a C string (a pointer to a sequence of ASCII characters terminated with a null character) into a Verilog string (a vector with 8-bit groups representing characters).
<code>void vc_VectorToString(vc_handle, char *)</code>	Converts a vector value to a string value.
<code>int vc_getInteger(vc_handle)</code>	Same as <code>vc_toInteger</code> .
<code>void vc_putInteger(vc_handle, int)</code>	Passes an int value by reference through a <code>vc_handle</code> to a scalar reg or bit or a vector bit that is 32 bits or less.
<code>vec32 *vc_4stVectorRef(vc_handle)</code>	Returns a <code>vec32</code> pointer to a four state vector. Returns NULL if the specified <code>vc_handle</code> is not to a four-state vector reg.
<code>U *vc_2stVectorRef(vc_handle)</code>	This routine returns a U pointer to a bit vector that is larger than 32 bits. If you specify a short bit vector (32 bits or fewer), this routine returns a NULL value.
<code>void vc_get4stVector(vc_handle, vec32 *)void vc_put4stVector(vc_handle, vec32 *)</code>	Passes a four-state vector by reference to a <code>vc_handle</code> to and from an array in C/C++ function. <code>vc_get4stVector</code> receives the vector from Verilog and passes it to the array. <code>vc_put4stVector</code> passes the array to Verilog.

Table 59 Access Routine Description (Continued)

<pre>void vc_get2stVector(vc_handle, U *)void vc_put2stVector(vc_handle, U *)</pre>	Passes a two state vector by reference to a <code>vc_handle</code> to and from an array in C/C++ function. <code>vc_get2stVector</code> receives the vector from Verilog and passes it to the array. <code>vc_put2stVector</code> passes the array to Verilog.
<pre>UB *vc_MemoryRef(vc_handle)</pre>	Returns a pointer of type <code>UB</code> that points to a memory in Verilog.
<pre>UB *vc_MemoryElemRef(vc_handle, U indx)</pre>	Returns a pointer to an element (word, address or index) of a Verilog memory. You specify the <code>vc_handle</code> of the memory and the element.
<pre>scalar vc_getMemoryScalar(vc_handle, U indx)</pre>	Returns the value of a one-bit memory element.
<pre>void vc_putMemoryScalar(vc_handle, U indx, scalar)</pre>	Passes a value, of type <code>scalar</code> , to a Verilog memory element. You specify the memory by <code>vc_handle</code> and the element by the <code>indx</code> parameter.
<pre>int vc_getMemoryInteger(vc_handle, U indx)</pre>	Returns the integer equivalent of the data bits in a memory element whose bit-width is 32 bits or less.
<pre>void vc_putMemoryInteger(vc_handle, U indx, int)</pre>	Passes an integer value to a memory element that is 32 bits or fewer. You specify the memory by <code>vc_handle</code> and the element by the <code>indx</code> parameter.
<pre>void vc_get4stMemoryVector(vc_handle, U indx, vec32 *)</pre>	Copies the value in an Verilog memory element to an element in an array. This routine copies both the data and control bytes. It copies them into an array of type <code>vec32</code> .
<pre>void vc_put4stMemoryVector(vc_handle, U indx, vec32 *)</pre>	Copies Verilog data from a <code>vec32</code> array to a Verilog memory element.
<pre>void vc_get2stMemoryVector(vc_handle, U indx, U *)</pre>	Copies the data bytes, but not the control bytes, from a Verilog memory element to an array in your C/C++ function.
<pre>void vc_put2stMemoryVector(vc_handle, U indx, U *)</pre>	Copies Verilog data from a <code>U</code> array to a Verilog memory element.
<pre>void vc_putMemoryValue(vc_handle, U indx, char *)</pre>	This routine works like the <code>vc_putValue</code> routine except that it is for passing values to a memory element instead of to a reg or bit. You enter an parameter to specify the element (index) you want the routine to pass the value to.

Table 59 Access Routine Description (Continued)

void vc_putMemoryValueF(vc_handle, U indx, char, char *)	This routine works like the <code>vc_putValueF</code> routine except that it is for passing values to a memory element instead of to a reg or bit. You enter an parameter to specify the element (index) you want the routine to pass the value to.
char *vc_MemoryString(vc_handle, U indx)	This routine works like the <code>vc_toString</code> routine except that it is for passing values to from memory element instead of to a reg or bit. You enter an parameter to specify the element (index) you want the routine to pass the value of.
char *vc_MemoryStringF(vc_handle, U indx, char)	This routine works like the <code>vc_MemoryString</code> function except that you specify a radix with the third parameter. The valid radices are 'b', 'o', 'd', and 'x'.
void vc_FillWithScalar(vc_handle, scalar)	This routine fills all the bits or a reg, bit, or memory with all 1, 0, x, or z values (you can choose only one of these four values).
char *vc_argInfo(vc_handle)	Returns a string containing the information about the parameter in the function call in your Verilog source code.
int vc_Index(vc_handle, U, ...)	Returns the index of a linearized array, or returns -1 if the U-type parameter is not an index of a multi-dimensional array, or the <code>vc_handle</code> parameter is not a handle to a multi-dimensional array of the reg data type.
int vc_Index2(vc_handle, U, U)	Specialized version of <code>vc_Index()</code> where the two U parameters are the indices in a two-dimensional array.
int vc_Index3(vc_handle, U, U, U)	Specialized version of <code>vc_Index()</code> where the two U parameters are the indexes in a three-dimensional array.
U vc_mdaSize(vc_handle, U)	If the U type parameter has a value of 0, it returns the number of indices in multi-dimensional array. If the U type parameter has a value greater than 0, it returns the number of values in the index specified by the parameter. There is an error condition if this parameter is out of the range of indices. If the <code>vc_handle</code> parameter is not a multi-dimensional array, it returns 0.

Enabling C/C++ Functions

The `+vc` elaboration/compile-time option is required for enabling the direct call of C/C++ functions in your Verilog code. When you use this option you can enter the C/C++ source files on the `vcs` command line. These source files must have a `.c` extension.

There are suffixes that you can append to the `+vc` option to enable additional features. You can append all of them to the `+vc` option in any order. For example:

```
+vc+abstract+allhdrs+list
```

These suffixes specify the following:

`+abstract`

Specifies that you are using abstract access through `vc_handle` to the data structures for the Verilog arguments.

When you include this suffix, all functions use abstract access except those with "C" in their declaration; these exceptions use direct access.

If you omit this suffix, all functions use direct access except those with the "A" in their declaration; these exceptions use abstract access.

`+allhdrs`

Writes the `vc_hdrs.h` file that contains external function declarations that you can use in your Verilog code.

`+list`

Displays on the screen all the functions that you called in your Verilog source code. In this display, void functions are called procedures. The following is an example of this display:

The following external functions have been actually called:

```
procedure receive_string
procedure passbig2
function return_string
procedure passbig1
procedure memory_rewriter
function return_vector_bit
procedure receive_pointer
procedure incr
function return_pointer
function return_reg
```

[DirectC interface]

Mixing Direct And Abstract Access

If you want some C/C++ functions to use direct access and others to use abstract access, you can do so by using a combination of "A" or "C" entries for abstract or direct access in the declaration of the function and the use of the `+abstract` suffix. The following table shows the result of these combinations:

	no +abstract suffix	include the +abstract suffix
extern (no mode specified)	direct access	abstract access
extern "A"	abstract access	abstract access
extern "C"	direct access	direct access

Specifying the DirectC.h File

The C/C++ functions need the `DirectC.h` file to use abstract access. This file is located in `$VCS_HOME/include` (and there is a symbolic link to it at `$VCS_HOME/platform/lib/DirectC.h`). You need to instruct VCS where to look for it. You can accomplish this in the following three ways:

- Copy the `$VCS_HOME/include/DirectC.h` file to your current directory. VCS always looks for this file in your current directory.
- Establish a link in the current directory to the `$VCS_HOME/include/DirectC.h` file.
- Include the `-CC` option as follows:
`-CC "-I$VCS_HOME/include"`

Extended BNF for External Function Declarations

A partial EBNF specification for external function declaration is as follows:

```

source_text ::= description +
description ::= module | user_defined_primitive
| extern_function_declaration
extern_function_declaration ::= extern access_mode extern_func_type
extern_function_name ( list_of_extern_func_args ? ) ;
access_mode ::= ( "A" | "C" ) ?

```

Note:

If access mode is not specified, then the command line option `+abstract` rules; default mode is "C".]

```

extern_func_type ::= void | reg | bit | DirectC_primitive_type
| bit_vector_type
bit_vector_type ::= bit [ constant_expression : constant_expression ]
list_of_extern_func_args ::= extern_func_arg
( , extern_func_arg ) *
extern_func_arg ::= arg_direction ? arg_type optional_arg_name ?

```

Note:

Argument direction (that is, input, output, inout) applies to all arguments that follow it until the next direction occurs; the default direction is input.

```

arg_direction ::= input | output | inout
arg_type ::= bit_or_reg_type | array_type | DirectC_primitive_type
bit_or_reg_type ::= ( bit | reg ) optional_vector_range ?
optional_vector_range ::= [ ( constant_expression :
constant_expression ) ? ]
array_type ::= bit_or_reg_type array [ ( constant_expression :
constant_expression ) ? ]
DirectC_primitive_type ::= int | real | pointer | string

```

In this specification, **extern_function_name** and **optional_arg_name** are user-defined identifiers.

Using DPI Open Array

Open array is a DPI array formal argument for which the packed or unpacked dimension size (or both) is not specified and for which interface routines describe the size of corresponding actual arguments at runtime.

This section describes support for DPI open array in the following section:

- [Using Multi-Dimensional Array as an Actual Argument for DPI Function](#)

Using Multi-Dimensional Array as an Actual Argument for DPI Function

VCS supports fixed array, dynamic array as actual argument of DPI function whose formal argument is open array. Following is an example of dynamic Multi-Dimensional array and mixed multi-dimensional array as actual argument for DPI function.

Example 207 Usage of Dynamic Unpacked MDA as Actual Argument

The content of the `test.c` file is as follows:

```

#include "svdpi.h"
void foo(const svOpenArrayHandle h)

```

Chapter 26: C Language Interface
Using Multi-Dimensional Array as an Actual Argument for DPI Function

```
{
    int i,j;
    for (i= svLow(h, 1); i <= svHigh(h, 1); i++) {
        for (j= svLow(h, 2); j <= svHigh(h, 2); j++) {
            printf("h[%d] [%d] is %d\n", i,j,* (svLogicVecVal*) svGetArrElemPtr2(h,
i,j));
        }
    }
}
```

The content of the `test.sv` file is as follows:

```
module top;
import "DPI-C" function void foo(input logic [2:0]oa[][]);
logic [2:0]a_2x2 [][]; //Multi-dimensional dynamic array
logic [2:0]a_3x3 [1:3][]; //Multi-dimensional mixed array
task initialize();
    a_2x2 = new[2];
    for(int i=0;i<2;i++) begin
        a_2x2[i] = new[2];
        for(int j=0;j<2;j++) begin
            a_2x2[i][j] = i+j;
        end
    end
    for(int i=1;i<=3;i++) begin
        a_3x3[i] = new[3];
        for(int j=0;j<3;j++) begin
            a_3x3[i][j] = i*j;
        end
    end
endtask
initial begin
    initialize();
    $display("SV: foo(a_2x2)");
    foo(a_2x2);
    $display("SV: foo(a_3x3)");
    foo(a_3x3);
end
endmodule
```

Run the example using the following commands:

- % vcs -sverilog test.c test.sv
- % simv

The output generated is as follows:

```
SV: foo(a_2x2)
h[0][0] is 0
h[0][1] is 1
h[1][0] is 1
h[1][1] is 2
```

```
SV: foo(a_3x3)
h[1][0] is 0
h[1][1] is 1
h[1][2] is 2
h[2][0] is 0
h[2][1] is 2
h[2][2] is 4
h[3][0] is 0
h[3][1] is 3
h[3][2] is 6
```

Limitations

The limitations of this feature are as follows:

- Multiple-dimensional dynamic array whose base type is Unpacked structure/union is not supported as actual argument.
- Complex usages like concatenation operator are not supported as actual argument.

27

Support for VHDL 2002, 2008 and 2019

VCS supports the following VHDL standards:

- **VHDL 2002 Protected Type** - VHDL 2002 is supported with the `-vhdl102` switch or `VHDL_MODE=vhdl102` setup variable (which should be specified in the `synopsys_sim.setup` file).
- **VHDL 2008 Constructs** - VHDL 2008 is supported with the `-vhdl108` switch or `VHDL_MODE=vhdl108` setup variable (which should be specified in the `synopsys_sim.setup` file).
- **VHDL 2019 Conditional Analysis Tool Directives** - The VHDL 2019 conditional analysis tool directives are supported with the following options and the setup variables (the setup variable must be specified in the `synopsys_sim.setup` file):
 - Options: `-vhdl187`, `-vhdl193`, `-vhdl102`, or `-vhdl108` (`vhdlan`)
 - Setup variables: `VHDL_MODE = vhdl187`, `VHDL_MODE = vhdl193`, `VHDL_MODE = vhdl108`, or `VHDL_MODE = vhdl102`

VHDL 2002 Protected Type

Protected type is supported by VCS. For more information on protected type, refer to clause 3.5 in the VHDL 2002 LRM.

Use Model

Protected type is supported with the following options:

`-vhdl102` or `-vhdl108` (`vhdlan`) switch.

OR

`VHDL_MODE = vhdl108` or `VHDL_MODE = vhdl102` setup variable.

Limitations of VHDL 2002 Protected Type

VHDL 2002 protected type works with the following limitations:

- VHPI interface to protected types is not supported.
 - Protected type in subprograms is not supported.
 - UCLI support is not yet implemented.
-

VHDL 2008 Constructs

VCS supports the following VHDL 2008 constructs that are described in this section. To enable the VHDL 2008 constructs, specify the `-vhdl108` option in the `vhdlan` command line or set the `VHDL_MODE=vhdl108` variable in the `synopsys_sim.setup` file.

- [Array Types and Operators](#)
- [Adding Comments](#)
- [Use Clause and Aliases](#)
- [Support for Bit String Literals](#)
- [Support for TO_STRING Conversion](#)
- [Support for External Names](#)
- [Specifying The all Keyword in the Process Sensitivity List](#)
- [Support for Logical Unary Reduction Operator](#)
- [Support for Matching Relational Operators for Bit and std_ulogic](#)
- [Including Non-Static Expressions in Port Map](#)
- [Standard Environment Package](#)
- [Package Declaration and Instantiations](#)
- [Referencing Interface Lists](#)
- [Overriding the Value Assigned to a Signal](#)
- [Matching Case Statements](#)
- [Conditional Elaboration](#)
- [Condition Operator in an Expression](#)
- [Reading Output Port](#)

- 2008 IEEE Packages
- Resolved Elements
- Conditional and Selected Assignments
- Context Declaration
- Improved I/O
- Support for Implicitly Constrained Array Elements
- Support for Unconstrained Element Types
- Support for Enhanced Generics in Entity Interfaces
- Support for Slices in Array Aggregates
- Support for Type Conversion in VHDL 2008
- Support for Case Expression Subtype
- Support for Subtypes of Ports and Parameters
- Support for Static Composite Expressions
- Support for VHDL 2008 Enhanced Generics in Components
- Support for VHDL 2008 Local Packages
- Support for Unconstrained Elements in Generics and Generic Types

Array Types and Operators

As defined in the *VHDL 2008 LRM*, section 5.3.2.3 and 5.3.2.4, VCS supports predefined array types, STRING, BOOLEAN_VECTOR, BIT_VECTOR, INTEGER_VECTOR, REAL_VECTOR, and TIME_VECTOR. These data types are defined in STANDARD package. For more information about these data types and its definition, see the *IEEE Std. VHDL 2008 LRM*.

Example,

```
entity test1 is
port(w1 : out integer_vector(2 downto 0);
     x1 : out boolean_vector(2 downto 0);
     y1 : out real_vector(2 downto 0);
     t1 : out time_vector(2 downto 0);
     );
end test1;

architecture ar_test1 of test1 is
begin
  w1<=(10,20,30);
```

```
x1<=(TRUE, FALSE, TRUE);
y1<=(123.234, 9876.12345, 12.1);
t1 <= (10 ns, 20 ns, 60 ns);
end ar_test1;
```

Adding Comments

As defined in the *IEEE Std. VHDL 2008 LRM*, a comment is either a single-line comment or a delimited comment. A single-line comment starts with two adjacent hyphens and extends up to the end of the line. A delimited comment starts with a solidus (slash) character immediately followed by an asterisk character and extends up to the first subsequent occurrence of an asterisk character immediately followed by a solidus character.

Example1 - Single-line Comment

```
entity full_adder is
    port (A, B, CIN: in std_logic; sum, cout: out std_logic);
end full_adder;

architecture full_adder of full_adder is
begin
    SUM <= A xor B xor CIN; -- My comment for SUM
    COUT <= (A and B) or (B and CIN) or (CIN and A);
                                --My comment for CARRY
end full_adder;
```

Example 2 - Delimited Comment

```
entity full_adder is
    port (A, B, CIN: in std_logic; sum, cout: out std_logic);
end full_adder;

architecture full_adder of full_adder is
begin
    SUM <= A xor B xor CIN; /* My comment for SUM */
    COUT <= (A and B) or (B and CIN) or (CIN and A);
                                /* My comment for CARRY */
end full_adder;
```

Use Clause and Aliases

VCS supports use clauses as defined in the *IEEE Std. VHDL 1076-2008 LRM*, section 12.4. A use clause achieves direct visibility of declarations that are visible by selection. For information about the syntax and functionality, see the *IEEE Standard VHDL 1076-2008 LRM*.

Example,

```
package P is
    type color is (BLUE, GREEN, RED, YELLOW);
end package;
use work.P.color;
entity e is
end;
architecture arch of e is
    signal A1 : color ;
begin
    A1 <= GREEN ; //Now enum literal will be visible with
                    the use of type identifier with use
end architecture;
```

VCS also supports alias declarations as defined in VHDL 1076-2008 LRM, section 6.6. An alias declaration declares an alternate name for an existing named entity. For information about the syntax and functionality, see the *IEEE Standard VHDL 1076-2008 LRM*.

Example,

```
package P1 is
    type alu_type1 is (add,sub,mul,div);
end package;
use work.P1.alu_type1;
package P2 is
    alias alu_type is alu_type1;
end package;
use work.P2.alu_type;
entity e is
end;
architecture arch of e is
    signal a1, a2 : alu_type := add;
begin
    process
begin
    a1 <= a2;
    wait;
end process;
end;
```

Support for Bit String Literals

As defined in the *IEEE Standard VHDL 1076-2008 LRM*, section 15.8, a bit string literal is formed by a sequence of characters (possibly none) enclosed between two quotation marks used as bit string brackets, preceded by a base specifier. The bit string literal can also be preceded by an integer specifying the length of the value represented by the bit string literal.

```
bit_string_literal ::= [ integer ] base_specifier "[ bit_value ]"
```

Where,

```
bit_value ::= graphic_character {[ underline ] graphic_character }
baseSpecifier ::= B | O | X | UB | UO | UX | SB | SO | SX | D
```

For more information about the rules and definition of bit string literals, see the *IEEE Standard VHDL 1076-2008 LRM*.

VCS supports VHDL bit string literals to specify a value for vector of 0 and 1 elements in binary, octal, or hexadecimal form. You can use the bit string literals by adding the `-vhdl08` option in `vhdlan` command line. With this enhancement, you can:

- Specify elements other than 0 and 1:
 - In a binary literal, any non-bit character represents itself
 - In an octal literal, any non-octal-digit character is expanded to three occurrences of the character in the vector value
 - In a hexadecimal literal, any non-hexadecimal digit character is expanded to four occurrences of the character in the vector value
 - Embed underscore (_) for readability as it will not be expanded.
- Specify the exact length of the vector represented by a literal
- Specify if the literal represents an unsigned or signed number
- Specify a vector value in the decimal form

VHDL 2008 allows you to specify the length of a bit string literal. The length of a bit string literal is the length of its string literal value. If a bit string literal includes the integer immediately preceding the base specifier, the length of the bit string literal is the value of the integer. Else, the length is the number of characters in the expanded bit value. The string literal value is obtained by adjusting the expanded bit value to the length of the bit string literal.

Examples,

`6x"a"` is equivalent to `b"001010"`

`32b"10"` is equivalent to `b"00000000_00000000_00000000_00000010"`

`7o"123"` is equivalent to `b"1010011"`

VHDL 2008 adds signed bit string literals, which permits you to represent negative values in fixed-width representations. In a signed bit string literal, the leftmost element of the expanded bit value is duplicated, if the specified length is longer. If the specified length is shorter, elements might be truncated, if they are the same as the new leftmost element. For unsigned literals, any truncated elements must be 0.

Examples,

`16so"6"` is equivalent to `b"1111111_11111110"`

`6sx"fe"` is equivalent to `b"111110"`

`8sb"100111000"` is an error

VHDL 2008 allows you to specify bit string literals in decimal form with the expanded bit value having sufficient bits to contain the largest power of two (not greater than the value). Decimal bit string literals are always unsigned.

Examples,

`d"47"` is equivalent to `b"101111"`

`16d"25"` is equivalent to `b"00000000_00011001"`

`7d"255"` is an error

Note:

As the length of the expanded bit value is chosen so that the leftmost element is always 1, any specified length must not be less than that.

VHDL 2008 permits characters other than underscores and extended digits. This permits you to specify metalogical values for bit string literals to use them as aggregates for arrays of `std_ulogic`. If the radix is octal or hexadecimal, the character is duplicated as necessary.

Examples,

`8x"Z5"` is equivalent to `"ZZZZ0101"`

`7so"X3"` is equivalent to `"XXXX011"`

`6b"ABC"` is equivalent to `"000ABC"` (not valid as an array of `std_logic` but still permitted by the language)

Support for TO_STRING Conversion

VCS supports the `TO_STRING (x)` predefined function, as defined in the *VHDL 2008 LRM*, sections 5.2.6 and 5.7. The `TO_STRING (x)` function works with any scalar type and any one-dimensional array type where the element type is an enumerated type all of whose literals are character literals.

VCS supports the following VHDL 2008 overloaded versions of the `TO_STRING` predefined function:

- `TO_STRING (value: time; unit: time)` — use unit instead of the resolution limit
- `TO_STRING (value: real; digits: natural)` — if digits are greater than 0, then it controls the number of digits after decimal, and suppresses the exponent
- `TO_STRING (value: real; format: string)` — use a restricted subset of `printf(3)` conversions (no length modifiers, asterisks, or text other than a conversion)

Additionally, for all bit-based array types, you can define the octal and the hexadecimal string conversion functions as follows:

- `TO_OSTRING [BitArrayType return string]`
- `TO_HSTRING [BitArrayType return string]`

You can use the `TO_STRING (x)` functions by adding the `-vhdl108` option in `vhdlan` command line.

The `to_string/to_ostring/to_hstring` literals are built-in functions and following are some of the conversion examples:

TO_STRING	Expected Output
<code>to_string(bit('1'))</code>	“1”
<code>to_string(1.5 ms, 2ns)(time resolution 1 ns)</code>	“750000 ns”
<code>to_string(52.5, “%5.2f”)</code>	“52.50”
<code>to_ostring(bit_vector("111111"))</code>	“77”

Example

The following example illustrates the usage of `to_string` with `bit_vector` and `std_logic_vector`:

```
entity E is
end entity E;
architecture demo of e is
    signal s1 : string(1 to 5) := to_string(bit_vector'("10111"));
    signal s2 : string(1 to 5) := to_string(std_logic_vector'("1ZU0X"));
    begin
    process
        begin
            report to_string(s1);
            report to_string(s2);
        wait;
    end process;
end architecture;
```

```
    end process;
end architecture demo;
```

Support for External Names

VCS supports VHDL External Names, which is introduced in the 1076-2008 - IEEE Standard as specified in the section 8.7 of VHDL Language Reference Manual.

VHDL had never provided an easy way to access a signal or a shared variable inside a design hierarchy from any other levels of design/verification environment. With the introduction of External Names, this limitation is no longer valid and you can easily access a constant, a signal or a shared variable residing in any other VHDL design hierarchy.

In tune with the enhancements in the *IEEE Standard VHDL 1076-2008 LRM*, VCS also supports the *External Names* feature. For more information about *External Names* feature, see the *IEEE Standard VHDL 1076-2008 LRM*.

Example

Leaf module:

```
entity e is
end entity;
architecture a of e is
signal s : std_logic :='1';
begin
end a;
```

Top module instantiating the Leaf module:

```
entity tb is
end entity;
architecture arch of tb is
signal s1 : std_logic ;
begin
  inst : entity work.e; // instance
process is
begin
  s1 <= << signal .tb.inst.s:std_logic>>;//external name
  wait for 1 ns;
  wait;
end process;
end arch;
```

To run the example, use the following command:

```
% vhdlan -vhdl108 test.vhd
% vcs tb -R
```

Specifying The all Keyword in the Process Sensitivity List

VCS supports the usage of the VHDL 2008 `all` keyword in the process sensitivity list as defined in the section 11.3 of *IEEE Standard VHDL Language Reference Manual (IEEE Std 1076-2008)*. Use the `-vhdl08` option to enable this feature.

As per the VHDL 2008 LRM, you can use the `all` keyword in the process sensitivity list, as shown in the following command:

```
process_sensitivity_list ::= all | sensitivity_list
```

For example, consider the following process that has signals `s1` and `si` in the sensitivity list:

```
process (s1, si) is
begin
    s4(si) <= s1;
end process;
```

When you use `process (all)`, the preceding code change as follows:

```
process (all) is
begin
    s4(si) <= s1;
end process;
```

Support for Logical Unary Reduction Operator

VCS supports the use of unary logical operators `and`, `or`, `nand`, `nor`, `xor`, and `xnor` and are referred to as logical reduction operators, which is introduced in section 9.2.2 of *IEEE Standard VHDL 2008 Language Reference Manual*. Each such logical reduction operator can be applied for any one-dimensional array type whose element type is `bit` or `boolean` and this produces a `bit` or `boolean` result, respectively.

Example,

```
entity test is
end;

architecture arch of test is

signal bv1 : bit_vector(2 downto 0);
signal bs1,bs2,bs3,bs4,bs5,bs6,bs7,bs8 : bit;

begin
process
begin
    wait for 1 ns;
    bv1 <= "010";
    wait for 1 ns;
```

```

bs2 <= and bv1;
bs3 <= or bv1;
bs4 <= nand bv1;
bs5 <= nor bv1;
bs6 <= xor bv1;
bs7 <= xnor bv1;
wait;
end process;
end;
```

Support for Matching Relational Operators for Bit and std_ulogic

VCS supports the usage of matching relational operators `?=`, `?/=`, `?<`, `?<=`, `?>`, and `?>=` that returns bit or `std_ulogic` results, which is introduced in section 9.2.3 of *IEEE Standard VHDL 2008 Language Reference Manual*.

The result type of each matching relational operator is same as the type of the operands (for scalar operands) or the element type of the operands (for array operands). For more information about matching relational operators, see the *IEEE Std. VHDL 2008 LRM*.

Example

```

entity test is
end;

architecture arch of test is

signal b1,b2,b3,b4,b5,b6,b7,b8 :std_ulogic;

begin
process
begin
    wait for 1 ns;
    b1 <= '1';
    b2 <= '1';
    wait for 1 ns;
    b3 <= b1 ?= b2;
    b4 <= b1 ?/= b2;
    b5 <= b1 ?< b2;
    b6 <= b1 ?<= b2;
    b7 <= b1 ?> b2;
    b8 <= b1 ?>= b2;
    wait;
end process;
end;
```

Output of this example is as follows:

```

value of b3 is '1';
value of b4 is '0';
value of b5 is '0';
```

```
value of b6 is '1';
value of b7 is '0';
value of b8 is '1';
```

Including Non-Static Expressions in Port Map

VCS supports writing an expression in a port map to include non-static expressions involving the values of signals, as defined in section 6.5.7.3 of *IEEE Std. VHDL-2008 LRM*.

This enables you to include functional logic in a port map, and avoid the need to express the logic with a separate assignment statement and an intermediate signal. If the expression is not static, the port association is defined to be equivalent to association with an anonymous signal that is the target of a signal assignment with the expression on the right-hand side.

For example,

Consider the entity `test` with port “`a`”:

```
library ieee;
use ieee.std_logic_1164.all;

entity test is port(a : in std_logic);
end entity;

architecture arch of test is
begin
end arch;
```

And the entity `top`, where entity `test` is instantiated with port “`a`” having an expression in port mapping:

```
library ieee;
use ieee.std_logic_1164.all;

entity top is port(    a1 : in std_logic;
                      b1 : in std_logic);
end entity;
```

Because of this enhancement, you can include the non-static signal expression in a port map, in which the architecture is simpler as shown in the following code:

```
architecture arch of top is
component test
    port (a : in std_logic);
end component;
begin
    vh_inst : test port map ( a => a1 and b1 );
end arch;
```

Command Line:

```
% vhdlan -vhdl08 basic_sig_exp.vhd
% vcs top
% ./simv
```

Standard Environment Package

As defined in the section 16.5 of *IEEE Std. VHDL 2008 LRM*, package ENV contains declarations that provide a VHDL interface to the host environment. You can use the std.env.all package to provide a VHDL interface.

Example,

```
use IEEE.std_logic_1164.all;
use std.env.all;  /* Use the env package
entity e is
    port (clk : in bit );
end entity;
architecture arch of e is
begin
check : process(clk)
begin
    if (clk'event and clk = '1') then
        stop; // Call the stop
    end if;
end process;
end architecture;
```

Note:

Execution of STOP procedure causes the simulator to stop.

Package Declaration and Instantiations

Generics provide a channel for communicating information to a block, a package, or a subprogram from its environment. VCS supports generic classes (constants, types, packages, and subprograms) in only *packages* as defined in the following sections.

As defined in the VHDL 2008 LRM, section 13.1 and 13.2, you can declare packages as design units. They are separately analyzed into a design library, and can be referenced by any other design unit that names the library. Thus, they are applicable only in the global scope. Using VHDL-2008, you can declare the packages locally within the declarative region of an entity, architecture, block, process, subprogram, protected type body, or enclosing package. However, VCS supports the following generics only in a *global scope*.

- *Generic classes*: the following generic classes are supported in a package:
 - Generic constants
 - Generic types
 - Generic packages
 - Generic subprograms

Use Model

To use generics in your design source code, specify the `-vhdl08` option in the `vhdlan` command line, or enable VHDL 2008 using the `VHDL_MODE=vhdl08` variable in the `synopsys_sim.setup` file.

Example

The following example includes packages with all the supported generic constants and generic types:

```
library ieee;
use ieee.std_logic_1164.all;
package uninst_pack is
generic (g1 : integer);
function foo return integer;
end package uninst_pack;
package body uninst_pack is
function foo return integer is
begin
return g1 + 11;
end;
end package body uninst_pack;
package inst_pack is new work.uninst_pack generic map (g1 => 31);
-----
library ieee;
use ieee.std_logic_1164.all;
package global is
generic (type DT;
left , right : DT;
function inc(x : DT) return DT;
procedure chk(x : DT; y : DT);
package inst_pack is new work.uninst_pack generic map (<>));
use inst_pack.all;
procedure self_check;
end package global;
package body global is
procedure self_check is
variable v : DT := left;
begin
v := inc(v);
chk(v, right);
end procedure;
```

```

end package body global;
-----
library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_arith.all;
package normal_package is
  function incr (x : integer) return integer;
  function incr (x : std_logic_vector) return std_logic_vector;
  procedure util_assert(i1 : integer; i2 : integer);
  procedure util_assert(sv1 : std_logic_vector; sv2 : std_logic_vector);
end package normal_package;
package body normal_package is
  function incr(x : integer) return integer is
  begin
    report "--From Function incr1--";
    return x + 1;
  end;
  function incr(x : std_logic_vector) return std_logic_vector is
  begin
    report "--From Function incr2--";
    return unsigned(x) + 1;
  end;
  procedure util_assert(i1 : integer; i2 : integer) is
  begin
    assert i1 = i2 report to_string(i1) & " /= " & to_string(i2);
  end procedure;
  procedure util_assert(sv1 : std_logic_vector; sv2 : std_logic_vector) is
  begin
    assert sv1 = sv2 report to_string(sv1) & " /= " & to_string(sv2);
  end procedure;
end package body normal_package;
-----
use work.normal_package.all;
package global_inst is new work.global
generic map (
  DT => integer,
  left => 8, right => 9,
  inc => incr, chk => util_assert,
  inst_pack => work.inst_pack
);
use work.global_inst.all;
entity top is
end entity top;
architecture behav of top is
begin
  process is
  begin
    self_check;
    wait;
  end process;
end architecture behav;

```

To use an uninstantiated package, it is required to instantiate it with the actual generics, as shown in the following example:

```
package global_inst is new work.global
generic map (
DT => integer,
left => 8, right => 9,
inc => incr, chk => util_assert,
inst_pack => work.inst_pack
);
use work.global_inst.all;
```

Command Line

To run the example, use the following commands:

```
% vhdlan -vhdl108 generics_package.vhd
% vcs top -R
```

Note:

If the actual generic type is unconstrained and if a formal generic type is used in the context of the constrained type, VCS issues an error message.

Limitation

Uninstantiated package declarations and package instantiations are only allowed at the design unit level (that is, at a global scope). Local package declaration and instantiations are not supported.

Referencing Interface Lists

A generic interface list consists entirely of interface constant declarations, interface type declarations, interface subprogram declarations, and interface package declarations. VCS supports declaring the following interface constants, as defined in the VHDL 2008 Std. LRM sections, 6.5.6.1 and 6.5.6.2:

- Reference *generic constants* in a generic list
- Reference *generic type* in a declaration of a subsequent generic

You can use one formal generic declared within a given generic list to declare the subsequent generics in this list. With this enhancement, you can use the value of generic to constrain the size of a subsequent generic of an array type.

Use Model

For referencing interface lists in your design, specify the `-vhdl108` option in the `vhdlan` command line, or enable VHDL 2008 using the `VHDL_MODE=vhdl108` variable in the `synopsys_sim.setup` file.

Example

Example 208

```

library ieee;
use ieee.std_logic_1164.all;

entity test is
    generic (left : integer:=3;
             right: integer:=1;
             arr: bit_vector(left downto right):="110");
    port      (A : in bit_vector(left downto right));
end test;

architecture arch of test is
begin

    L1: process
    begin
        report to_string(left) & " " & to_string(right);
        report to_string(arr);
        report to_string(A);
    end process L1;

end arch ;

```

Command Line

To run the example, use the following commands:

```
% vhdlan -vhdl08 top.vhd
% vcs test -R
```

Limitations

The support for referencing generics in the generic list works with the following limitations:

- Overriding generics using the elaboration and runtime option

```
-gv|-gvalue generic_name=value and elaboration option
```

```
-gfile genericfile.txt
```

The values overridden using the elaboration/runtime options are not propagated to the subsequent generics in the list. In the following example, the value of the generic `size` when overridden using the `-gv size=16` option at elaboration time or runtime is not propagated to the subsequent generic, `left`. Thus, the generic, `left`, holds the previous value 7.

Example,

```

library ieee;
use ieee.std_logic_1164.all;
```

```

entity test is
    generic (
        size : integer:= 8;
        left : integer:= size-1;
        right : natural := 0
    );
    port (
        A : in bit_vector(left downto right)
    );
end test;
architecture arch of test is
begin
    L1: process
    begin
        report to_string(size);
        report to_string(left) & " " & to_string(right);
        report to_string(A);
    end process L1;
end arch;
```

Command Lines

To override at elaboration time, execute the following commands:

```
% vhdlan test.vhd -vhdl108
% vcs test -gv size=16
% ./simv
```

To override at elaboration time, execute the following commands:

```
% vhdlan test.vhd -vhdl108
% vcs test -debug_access=all -lca -gfile generic_file.txt
% ./simv
```

To override at runtime, execute the following commands:

```
% vhdlan test.vhd -vhdl108
% vcs test -debug_access=all -lca
% ./simv -gv size=16
```

- Overriding generics in a VHDL design unit instantiated under Verilog top (VL-VH scenario).

The values of VHDL generics when overridden from a Verilog-top design is not propagated to the subsequent generics in the list. In the following example, the value of generic, `size` when overridden to 16 is not propagated to subsequent generic, `left`. Thus, the generic, `left`, holds the previous value 7.

Example,

```
library ieee;
use ieee.std_logic_1164.all;
```

```

entity test is
  generic (
    size : integer:= 8;
    left : integer:= size-1;
    right : natural := 0
  );
  port (
    A : in bit_vector(left downto right)
  );
end test;
architecture arch of test is
begin
  L1: process
  begin
    report to_string(size);
    report to_string(left) & " " & to_string(right);
    report to_string(A);
  end process L1;
end arch;

module top;
  parameter NUM = 15;
  wire [7:0] arr;
  test #(.size(NUM)) test_inst(.A(arr));
endmodule

```

Command Lines

To run the example, use the following commands:

```

% vhdlan -vhdl08 -nc example3.vhd
% vlogan -sverilog -nc example3.v
% vcs top -nc -q
% ./simv

```

Overriding the Value Assigned to a Signal

Generally, it is required to override the value assigned to a signal during the verification cycle and force a different value onto the signal to verify various signal values. Overriding a signal enables you to:

- Set up a test scenario by forcing value to a state, which is normally achieved through a complex initialization sequence. Forcing the values allows you to bypass the sequence and thus reduces the verification time significantly.
- Inject erroneous values into the design to ensure that it detects errors.

VCS supports the VHDL 2008 standard for forcing and releasing the values of signals as defined in the VHDL 2008 LRM, section 10.5. With this capability, you can write VHDL testbench code to force and release signal values.

To enable the execution of force and release assignments in your testbench, specify the `-vhdl08` option in the `vhdlan` command line, or enable VHDL 2008 using the `VHDL_MODE=vhdl08` variable in the `synopsys_sim.setup` file.

This section discusses the following topics:

- [Forcing and Releasing Values of Signals](#)
- [Forcing and Releasing Ports of a Design](#)
- [Assigning Composite Value to a Collection of Signals](#)
- [Forcing and Releasing Assignment Written in a Subprogram](#)
- [Forcing and Releasing Multiple Concurrent Assignments](#)
- [Debugging the Force and Release Assignments](#)

Forcing and Releasing Values of Signals

Forcing a value is a sequential assignment written within a process that forms a part of the testbench. To force a signal with a force assignment, use the following syntax:

```
signal_name <= force expression;
```

This causes a delta cycle and forces the named signal to take on the value of the expression in that delta cycle, regardless of the value assigned to the signal by any normal signal assignment. The signal is considered to be active during the delta cycle. If the forcing value is different from the previous value, an event occurs on the signal. The processes sensitive to changes on the signal value, then respond to the value change in the normal way.

The rules relating the type of expression to the type of the target signal are applicable for force assignments. The target signal name can be a normal signal name, or it can be an external signal name, or an alias. Generally, using external names is a common use case, as you might often need to force an internal signal of a design from a testbench.

After the signal is forced, you can update the signal with another force assignment to change the overriding value, which causes the signal to become active again and possibly to have another event. Note that, you can force the updated signal as often as needed.

To stop forcing a signal, execute a release assignment with the following syntax:

```
signal_name <= release;
```

This causes a further delta cycle, with the signal being active. However, with the `release` assignment, the signal is no longer forced and the current values of its sources are used to

determine the signal value in the normal way. Thus, it enables you to take back the control of the signal.

Forcing and Releasing Ports of a Design

The ports are a form of signal. As defined in the VHDL 2008 LRM, you can force and release ports of a design. The values of forcing and releasing ports are governed by the driving value and the effective value.

The driving value is the value presented externally by an entity and is determined by the internal sources within the entity. The effective value is the value seen internally by an entity and is determined by whatever is externally connected to the port, whether it is an explicitly declared signal or a port of an enclosing entity. Depending on the port mode and the external connections, the driving and effective values may be different.

For example, an `inout` mode port of the type `std_logic` might drive a `0` value, but the externally connected signal might have another source driving a `1` value. In this case, the resolved value of the signal is `x`, and that value is seen as the effective value of the `inout` mode port.

VHDL 2008 allows you to force driving values and effective values of a port independently by including a force mode in an assignment. For explicitly declared signals, where the driving and effective values are the same, the distinction makes no difference.

For ports and signal parameters, to force the driving value, include the keyword `out` in the force assignment, as shown in the following syntax:

```
signal_name <= force out expression;
```

Alternatively, to force the effective value, include the keyword `in` in the force assignment, as shown in the following syntax:

```
signal_name <= force in expression;
```

To stop forcing the port's or signal parameter's driving value, use the `release` assignment with the keyword `out`, as shown in the following syntax:

```
signal_name <= release out;
```

Similarly, to stop the forced effective value, use the `release` assignment with the keyword `in`, as shown in the following syntax:

```
signal_name <= release in;
```

You can force and release the driving values of ports of the mode `out`, `inout`, and `buffer`. Ports and signal parameters of mode `in` can be forced or released only on the effective value. They do not have a driving value, therefore, force or release with mode `out` is not allowed.

The ports and signal parameters of all modes except the linkage have effective values, and therefore you can force and release the effective value of a port or a signal parameter of any mode except linkage.

If you omit the force mode (`out` or `in`) in a force or release assignment, a default force mode applies. For assignments to ports and signal parameters of mode `in` and to explicitly declared signals, the default force mode is `in`, forcing the effective value. For assignments to ports of mode `out`, `inout`, or `buffer`, and to signal parameters of mode `out` or `inout`, the default force mode is `out`, forcing the driving value.

Assigning Composite Value to a Collection of Signals

A testbench models a broken data driver connection by forcing a `z` value on the output part of the bidirectional port, while allowing the input part of the port to operate normally.

As defined in the VHDL 2008 LRM, VCS allows you to assign such composite value to a collection of signals. You can write the collection of signals in the form of an aggregate on the left-hand side of the assignment, as shown in the following syntax:

```
(carry_out, sum) <= ('0' & a) + ('0' & b);
```

Note that this form of aggregate assignment is legal in VHDL 2008. However, you cannot write an aggregate of signal names as the target of a force or release assignment to force or release each of the signal values. Instead, you must write a separate force or release assignment for each of the signals.

In scenarios where you define a resolved signal of a composite type, such as an array type (that is, a signal with multiple sources, each of which is a composite value), the resolution function for the signal takes an array of composite values and determines a composite value as the resolved value of the signal. You cannot write a force or a release assignment with an element of such a signal as the target. However, you can only force or release the signal as a whole.

Forcing and Releasing Assignment Written in a Subprogram

As defined in the VHDL 2008 LRM, VCS enables you to force and release assignment in a subprogram. A signal assignment written in a procedure that is not contained within a process can only assign to a signal parameter of the procedure.

VHDL 2008 allows force and release assignments in procedures outside of the processes to signals other than the signal parameters.

Forcing and Releasing Multiple Concurrent Assignments

Multiple forces and releases might occur for a given signal during a single simulation cycle, as they are sequential assignments written in processes. As defined in the VHDL 2008 LRM, if a force and release both occur, the effect is as though the release is

immediately overridden by the force, and so the signal remains forced. However, it is forced with the new force value.

As the effect of multiple forces is not defined in the LRM, it is recommended to avoid writing multiple forces in a testbench. The effect of multiple releases, however, is same as a single release, and a release assignment on a signal that is not forced has no effect.

Debugging the Force and Release Assignments

UCLI supports debugging of the VHDL 2008 force and release assignment statements. For EVCD dumping, language force is considered as a test bench driver, and same is applicable to VHDL 2008 force also.

The following UCLI commands support the VHDL 2008 force and release assignment statements:

- The `drivers` command lists the force assignment statements just as non-force assignment statements are listed.
- The `show` and `get` commands shows the value of signals with force assignment statements.
- The `stop -event` command stops the simulation when a force/release assignment statement triggers a value change.
- The `force -freeze` command overrides any active force assignment statement.
- The `release` command releases any active force assignment statement.

Matching Case Statements

VCS enables you to define the sequential matching `case` statement, as defined in the VHDL 2008 LRM, section 10.9. With this support, VCS allows you to write the matching `case` statement with do not care -.

You can specify the expression followed by sequential statements using `case-end case`, as shown in the following syntax:

```
case_statement ::= [ case_label : ]
case '?' expression is
case_statement_alternative
{ case_statement_alternative }
end case '?' [ case_label ] ';'
```

To enable the VHDL 2008 constructs, specify the `-vhdl08` option in the `vhdlan` command line, or enable VHDL 2008 using the `VHDL_MODE=vhdl08` variable in the `synopsys_sim.setup` file.

Example 1:

[Example 209](#) is an example with the matching `case` statement.

Example 209 Usage of matching case Statement

```

File: example.vhd
library ieee;
use ieee.std_logic_1164.all;

entity ent is
    port (
        sel : in std_ulogic_vector(2 downto 0);
        in1 : in std_ulogic_vector(3 downto 0);
        in2 : in std_ulogic_vector(3 downto 0);
        out1 : out std_ulogic_vector(3 downto 0)
    );
end entity ent;

architecture arch of ent is
begin
    process(sel,in1,in2)
    begin
        case ? sel is
            when "10-" => report "AND operation"; out1 <= in1 and in2;
            when "01-" => report "OR operation"; out1 <= in1 or in2;
            when "00-" => report "NAND operation"; out1 <= in1 nand in2;
            when others => null;
        end case ?;
    end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
entity top is
end entity top;

architecture arch of top is
    signal in1_tb,in2_tb,out1_tb : std_ulogic_vector(3 downto 0);
    signal sel_tb : std_ulogic_vector(2 downto 0);
begin
    ent_inst : entity work.ent(arch) port
    map(sel_tb,in1_tb,in2_tb,out1_tb);

    process
    begin
        in1_tb <= "1111"; in2_tb <= "0010";
        sel_tb <= "101";
        wait for 1 ns;
        report to_string(out1_tb);
        wait;
    end process;
end architecture arch;
```

You can run this example using the following command line:

```
% vhdlan -vhdl08 example.vhdl
% vcs top
% simv
```

VCS generates the following output:

```
Report NOTE at 0 NS in design unit ENT(ARCH) from
process /TOP/ENT_INST/_P0:
    "AND operation"
1 NS
Report NOTE at 1 NS in design unit TOP(ARCH) from process /TOP/_P0:
    "0010"
(simv): Simulation complete, time is 1 NS.
```

Conditional Elaboration

VCS enables you to define the sequential if-else-generate and case-generate statements, as defined in the VHDL 2008 LRM, section 11.8. With this support, VCS allows you to configure the alternatives.

You can specify the alternate sets of sequential or structural statements using if-else-generate, as shown in the following syntax:

```
generate_label : if [ alternative_label : ] condition generate
[block_declarative
begin ]

----- first alternative
[else [ alternative_label : ] generate
[ block declarative
begin ]

-----last alternative
[end [alternate_label];]
end generate [ generate_label ];
```

You can also include further conditions to test using the if-elsif -generate, as follows:

```
generate_label : if [ alternative_label : ] condition generate
[block_declarative
begin ]

----- first alternative
[end [alternate_label];]
{ elsif [ alternative_label : ] condition generate
[block declarative
begin ]
----- next alternative
[end [alternate_label];]
```

```

}
[else [ alternative_label : ] generate
[ block declarative
begin ]

-----last alternative
[end [alternate_label];]
end generate [ generate_label ];

```

Similarly, VCS also supports the `case-generate` statement that enables you to specify alternative sets of sequential or structural statements and the choice values for each alternative, as shown in the following syntax:

```

generate_label : case expression generate
when [ alternative_label : ] choice1 =>
[ block declarative
begin ]

----- first alternate
[end [alternate_label];]
when [ alternative_label : ] choice2 =>
[ block declarative
begin ]

----- next alternate
[end [alternate_label];]
when [ alternative_label : ] others =>
[ block declarative
begin ]

----- last alternate
[end [alternate_label];]
end generate [generate_label];

```

To enable the VHDL 2008 constructs, specify the `-vhdl08` option in the `vhdlan` command line, or enable VHDL 2008 using the `VHDL_MODE=vhdl08` variable in the `synopsys_sim.setup` file.

Example 1:

[Example 210](#) is an example with the `if-else-generate` statement.

Example 210 Usage of if-else-generate Statement

```

File: example1.vhd
library ieee;
use ieee.std_logic_1164.all;

entity test is
generic(j:integer :=2);
end test;

```

```

architecture arch of test is
begin
  IF1: if A1 : j>5 generate
    signal k : integer := 10;
    begin
      process
        begin
          report to_string(j);
          report to_string(k);
          wait;
        end process;
    elsif A2 : j <= 2  generate
      process
        begin
          report to_string(j+1);
          wait;
        end process;
    else  A3 : generate
      process
        begin
          report to_string(-j);
          wait;
        end process;
    end generate IF1;
  end arch;
```

You can run this example using the following command line:

```
% vhdlan -vhdl108 example1.vhd
% vcs test -R
% simv
```

VCS generates the following output:

```
Report NOTE at 0 NS in design unit TEST(ARCH) from process /TEST/IF1/_P0:
  "3"
(simv): Simulation complete, time is 0.
```

Example 2:

[Example 211](#) is an example with the `case-generate` statement.

Example 211 Usage of case-generate Statement

```
File: example2.vhd
library ieee;
use ieee.std_logic_1164.all;

entity test is
generic(s:std_ulogic:='X');
```

```

end test;

architecture arch of test is

begin

CASE1: case s generate
when A1 : 'U' | 'X' | '0' | '1' | 'Z' =>
process
begin
report to_string(s);
wait;
end process;
when A2 : 'W' | 'L' | 'H' =>
process
begin
report to_string('P');
wait;
end process;
when A3 : others =>
process
begin
report "others";
wait;
end process;
end generate CASE1;
end arch;
```

You can run this example using the following command line:

```
% vhdlan -vhdl08 example2.vhd
% vcs test -R
% simv
```

VCS generates the following output:

```
Report NOTE at 0 NS in design unit TEST(ARCH) from
process /TEST/CASE1/_P0:
  "X"
(simv): Simulation complete, time is 0.
```

For using cross-module references across the language boundaries or for tool specific configuration, you must use only generate label names without alternate label as shown in [Example 212](#):

Example 212 Usage of XMR across the if-else and case generate statements

```
File: example1.vhdl
library ieee;
use ieee.std_logic_1164.all;

entity ent1 is
port(
```

```

        in1 : in std_logic;
        in2 : in std_logic;
        sel : in std_logic;
        out1 : out std_logic
    );
end entity ent1;

architecture arch of ent1 is
begin
    process(in1,in2,sel)
    begin
        if(sel='1') then
            out1 <= in1;
        else
            out1 <= in2;
        end if;
    end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;

entity ent2 is
    generic (
        g0 : natural := 1;
        g1 : natural := 2;
        g2 : natural := 3
    );
    port(
        in1 : in std_logic;
        in2 : in std_logic;
        sel : in std_logic;
        out1 : out std_logic
    );
end entity ent2;

architecture arch of ent2 is
begin
    gen_lbl : if g0 = 2 generate
    elsif alt_lbl : g1 = 3 generate
    else last_lbl : generate
    begin
        gen_lbl2 : case g2 generate
        when first_alt : 4 =>
        when alt_lbl : others =>
            ent1_inst : entity ent1(arch) port map(in1,in2,sel,out1);
        end generate gen_lbl2;
        end last_lbl;
        end generate gen_lbl;
    end generate;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;

```

```

entity ent3 is
end entity ent3;

architecture arch of ent3 is
    signal in1_tb,in2_tb,sel_tb,out1_tb : std_logic;
begin

    ent2_inst : entity work.ent2(arch) port
map(in1_tb,in2_tb,sel_tb,out1_tb);

    process
    begin
        sel_tb <= '1'; in1_tb <= '1'; in2_tb <= '0';
        wait for 1 ns;
        sel_tb <= 'X'; in1_tb <= '0'; in2_tb <= '1';
        wait for 1 ns;
        sel_tb <= 'X'; in1_tb <= '1'; in2_tb <= '1';
        wait for 1 ns;
        sel_tb <= 'X'; in1_tb <= '0'; in2_tb <= '0';
        wait;
    end process;
end architecture arch;

File: example2.v
`timescale 1ns/1ns
module top;
ent3 ent3_inst ();
    initial $monitor("@time = %0t
%b",$time,top.ent3_inst.ent2_inst.gen_lbl.gen_lbl2.ent1_inst.out1); // 
xmr to output port
endmodule
File: xprop.cfg
instance {top.ent3_inst.ent2_inst.gen_lbl.gen_lbl2.ent1_inst}
{xpropOn}; // enabling xprop on ent1_inst using xprop config
file

```

You can run this example using the following command line:

```
% vhdlan -vhdl08 example1.vhd
% vlogan -sverilog example2.v
% vcs top -xprop=xprop.cfg
% simv
```

VCS generates the following output:

```
@time = 0 1
@time = 1 x
@time = 2 1
@time = 3 0
```

Condition Operator in an Expression

VCS enables you to use condition operator `??` in an expression, as defined in the VHDL 2008 LRM, section 9.2.9. Prior to VHDL 2008 a condition was required to have a Boolean value which was not convenient to model the designs that used `bit` or `std_ulogic` as control signals as shown:

```
if ctrl_sig = '1' then ...
```

Now, you can convert the `bit` or the `std_ulogic` value to a `boolean` value. For `bit` value, `??` converts `1` to `true` and `0` to `false`. For `std_ulogic` value, `??` converts both `1` and `H` to `true` and all other values to `false`.

You can rewrite the if-statement condition shown above using the following implicit and explicit ways:

```
if ?? ctrl_sig then ... -- explicit way
```

```
if ctrl_sig then ... -- implicit way
```

To enable the VHDL 2008 constructs, specify the `-vhdl08` option in the `vhdlan` command line, or enable VHDL 2008 using the `VHDL_MODE=vhdl08` variable in the `synopsys_sim.setup` file.

Example 1:

[Example 213](#) is an example with explicit condition operator.

Example 213 Usage of explicit condition operator

```
File: example1.vhd
library ieee;
use ieee.std_logic_1164.all;

entity ent is
    port ( in1 : in std_logic;
           in2 : in std_logic;
           sel : in std_logic;
           out1 : out std_logic
    );
end entity ent;

architecture arch of ent is
begin
    process(sel,in1,in2)
    begin
        if ?? sel then -- Using Conditional Operator Explicitly
            out1 <= in1;
        else
            out1 <= in2;
        end if;
    end process;
end architecture;
```

```

        end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;

entity top is
end entity top;

architecture arch of top is
    signal in1_tb,in2_tb,out1_tb,sel_tb : std_logic;
begin
    ent_inst : entity work.ent(arch) port
map(in1_tb,in2_tb,sel_tb,out1_tb);

    process
    begin
        in1_tb <= '1'; in2_tb <= '0'; sel_tb <= '1';
        wait for 1 ns;
        report "out1 = " & to_string(out1_tb);
        wait;
    end process;
end architecture arch;

```

Example 2:

[Example 214](#) is an example with implicit condition operator.

Example 214 Usage of Implicit Condition Operator

```

File: example2.vhd
library ieee;
use ieee.std_logic_1164.all;

entity ent is
    port ( in1 : in std_logic;
           in2 : in std_logic;
           sel : in std_logic;
           out1 : out std_logic
    );
end entity ent;

architecture arch of ent is
begin
    process(sel,in1,in2)
    begin
        if sel then -- Using Conditional Operator Implicitly
            out1 <= in1;
        else
            out1 <= in2;
        end if;
    end process;
end architecture arch;

```

```

library ieee;
use ieee.std_logic_1164.all;

entity top is
end entity top;

architecture arch of top is
    signal in1_tb,in2_tb,out1_tb,sel_tb : std_logic;
begin
    ent_inst : entity work.ent(arch) port
map(in1_tb,in2_tb,sel_tb,out1_tb);

    process
    begin
        in1_tb <= '1'; in2_tb <= '0'; sel_tb <= '1';
        wait for 1 ns;
        report "out1 = " & to_string(out1_tb);
        wait;
    end process;
end architecture arch;

```

You can run these examples using the following command line:

```
% vhdlan -vhdl08 example1.vhd
```

Or

```
% vhdlan -vhdl08 example2.vhd
% vcs top
% simv
```

VCS generates the following output:

```

1 NS
Report NOTE at 1 NS in design unit TOP(ARCH) from process /TOP/_P0:
    "out1 = 1"
(simv): Simulation complete, time is 1 NS.

```

Reading Output Port

VCS enables you to read out-mode ports, as defined in the VHDL 2008 LRM, section 9.2.9. Prior to VHDL 2008, internal reading of out-mode ports driving value was not allowed. In order to achieve this, the common practice followed was to restore the out-mode port value in an internal signal and drive on to out-mode ports. Now, it is no longer required to restore to an internal signal.

To enable the VHDL 2008 constructs, specify the `-vhdl08` option in the `vhdlan` command line, or enable VHDL 2008 using the `VHDL_MODE=vhdl08` variable in the `synopsys_sim.setup` file.

Example 1:

[Example 215](#) is an example for reading output parameter.

Example 215 Usage for Reading Output Parameter

```

File: example.vhdl
library ieee;
use ieee.std_logic_1164.all;

entity outModeRead is
    port(
        in1 : in std_logic_vector(0 to 3);
        in2 : in std_logic_vector(0 to 3);
        out_port : out std_logic_vector(0 to 3)
    );
end entity outModeRead;

architecture arch of outModeRead is
    procedure proc(in1,in2 : in std_logic_vector(0 to 3);out1
    : out std_logic_vector(0 to 3)) is
begin
    out1 := in1 or in2; -- reading the output
    parameter of procedure
    on LHS
end procedure proc;
begin
    process(in1, in2)
    variable sig : std_logic_vector(0 to 3);
    begin
        proc(in1,in2,sig);
        out_port <= out_port or sig; -- reading the output
        parameter of
        entity on LHS
    end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;

entity top is
end entity top;

architecture arch of top is
    signal in1_tb,in2_tb,out_port_tb : std_logic_vector(0 to 3);
begin
    inst : entity work.outModeRead(arch) port
    map(in1_tb,in2_tb,out_port_tb);
    process
    begin
        in1_tb <= "1101"; in2_tb <= "1100";
        wait for 1 ns;
        report to_string(out_port_tb);
    end process;
end architecture arch;

```

```
wait;
end process;
end architecture arch;
```

You can run this example using the following command line:

```
% vhdlan -vhdl08 example.vhd1
% vcs top
% simv
```

VCS generates the following output:

```
1 NS
Report NOTE at 1 NS in design unit TOP(ARCH) from process /TOP/_P0:
"11U1"
(simv): Simulation complete, time is 1 NS.
```

2008 IEEE Packages

The 2008 IEEE package includes:

- All the standard packages including pre-defined types in the VHDL Standard LRM.
- Several new packages including the fixed-point and the floating-point numbers represented as vectors of `std_ulogic` elements and a package providing access to the simulation environment.
- Operations to the standard packages that provide a consistent feature set across the suite. These operations provide a consistent set of conversion functions and I/O operations, such as `to_string`, `read`, `write`, and so on.

Overview of Additional IEEE Packages

This section provides an overview of enhancement to the IEEE packages, supported data types, and the corresponding operations that are included in VHDL 2008.

Table 60 VHDL 2008 Supported IEEE Packages

IEEE Packages	Supported Data Types	Supported Operations	Supported Functions and Procedures
std_logic_1164	1.STD_LOGIC 2.STD_ULOGIC 3.STD_LOGIC_VEC TOR 4.STD_ULOGIC_VECT OR 5.X01 6.X01Z 7.UX01 8.UX01Z	<ul style="list-style-type: none"> logical (and, nand, or, nor, xor, xnor, not) shift and rotate (sll,srl,rol,ror) conditional (??) 	<ul style="list-style-type: none"> Conversion (To_bit,To_bitvector,To_St dULogic,To_StdLogicVect or,To_StdULogicVector) Strength strippers (TO_01,To_X01,To_X01Z, To_UX01) Edge detection (rising_edge,falling_edge) Unknown detection(Is_X) String conversion (to_string, to_ostring,to_hstring,READ,WRITE,OREAD,OWRIT E,HREAD,HWRITE)
numeric_bit	1.signed 2.unsigned	<ul style="list-style-type: none"> arithmetic (+,-,*,/,rem, mod, abs) logical (and, nand, or, nor, xor, xnor, not) comparison(>,<,=,/=, ≥,≤) matching relational (?>,?<,?=,?/=,? ≥,?≤) shift and rotate (sll, srl, rol, ror, sla, sra) matching relational (?>,?<,?=,?/=,? ≥,?≤) 	<ul style="list-style-type: none"> arithmetic (find_leftmost,find_rightmo st) comparison (MAXIMUM,MINIMUM) shift and rotate (SHIFT_LEFT,SHIFT_RIG HT,ROTATE_LEFT,ROTA TE_RIGHT) Resize (RESIZE) Conversion (TO_INTEGER,TO_UNSI GNED,TO_SIGNED) Edge detection (rising_edge, falling_edge) String conversion (to_string, to_ostring, to_hstring, READ,WRITE,OREAD,O WRITE,HREAD,HWRITE)

Table 60 VHDL 2008 Supported IEEE Packages (Continued)

IEEE Packages	Supported Data Types	Supported Operations	Supported Functions and Procedures
numeric_std	1.SIGNED 2.UNSIGNED 3.UNRESOLVED_SIGNED 4.UNRESOLVED_UNSIGNED	<ul style="list-style-type: none"> arithmetic (+,-,*./,rem, mod, abs) logical (and, nand, or, nor, xor, xnor, not) comparison(>,<,=,/=, >=,<=) matching relational(?)>,<?, ?=,?/=?>=?<=?=) shift and rotate (sll, srl, rol, ror, sla, sra) 	<ul style="list-style-type: none"> arithmetic (find_leftmost,find_rightmost) comparison (MAXIMUM,MINIMUM) shift and rotate (SHIFT_LEFT,SHIFT_RIGHT,ROTATE_LEFT,ROTATE_RIGHT) Resize (RESIZE) Conversion (TO_INTEGER,TO_UNSIGNED,TO_SIGNED) Edge detection (rising_edge, falling_edge) Match functions (STD_MATCH) Translation (TO_01, TO_X01,TO_X01Z, TO_UX01,IS_X) String conversion (to_string, to_ostring, to_hstring, READ,WRITE,OREAD,O WRITE,HREAD,HWRITE).
numeric_bit_unsigned	• BIT_VECTOR	<ul style="list-style-type: none"> arithmetic (+,-,*./,rem,mod) comparison (>,<,=,/=,>=,<=) matching relational ()?>,<?,?=,?/=?>=?<=?=) shift and rotate (sll,srl,rol,ror,sla,sra) 	<ul style="list-style-type: none"> arithmetic (find_leftmost,find_rightmost) comparison (MAXIMUM,MINIMUM) shift and rotate (SHIFT_LEFT,SHIFT_RIGHT,ROTATE_LEFT,ROTATE_RIGHT) Resize (RESIZE) Conversion (TO_INTEGER, TO_bitvector)

Table 60 VHDL 2008 Supported IEEE Packages (Continued)

IEEE Packages	Supported Data Types	Supported Operations	Supported Functions and Procedures
numeric_std_unsigned	1.STD_LOGIC 2.STD_ULOGIC 3.STD_LOGIC_VEC_TOR 4.STD_ULOGIC_VECTORTOR	<ul style="list-style-type: none"> arithmetic (+,-,* ,/,rem,mod) comparison (>,<,=,/=,>=,<=) matching relational (?>,<?,?>=?/=?>=?<=) shift and rotate (sla,sra) 	<ul style="list-style-type: none"> arithmetic (find_leftmost,find_rightmost) comparison (MAXIMUM,MINIMUM) shift and rotate (SHIFT_LEFT,SHIFT_RIGHT,ROTATE_LEFT,ROTATE_RIGHT) Resize (RESIZE) Conversion (TO_INTEGER, TO_stdlogicvector, to_stdlogicvector)
fixed_pkg	1.UFIXED 2.UNRESOLVED_U FIXED 3.SFIXED 4.UNRESOLVED_S FIXED	<ul style="list-style-type: none"> arithmetic(+,-,* ,/,rem, mod,abs) logical (and,nand,or,nor,xor, xnor,not) comparison(>,<,=,/=, >=,<=) matching relational(>,<?,?>=?/=?>=?<=) shift and rotate (sll,srl,rol,ror,sla,sra) 	<ul style="list-style-type: none"> arithmetic (divide, reciprocal, remainder, modulo, add_carry, scalb, Is_Negative) comparison (std_match, MAXIMUM, MINIMUM, find_leftmost, find_rightmost) shift and rotate (SHIFT_LEFT,SHIFT_RIGHT,ROTATE_LEFT,ROTATE_RIGHT) Resize (RESIZE) Conversion (to_ufixed,to_unsigned,to_real,to_integer,to_sfixed,to_signed,to_slv,to_suv,to_UFix,to_SFix) Range generation (ufixed_high,ufixed_low,sfixed_high,sfixed_low,UFix_high,UFix_low,SFix_high, SFix_low,saturate) Translation (TO_01, TO_X01,TO_X01Z, TO_UX01,IS_X) String conversion(to_string, to_ostring,to_hstring,READ,WRITE,OREAD,OWRITE,HREAD,HWRITE,from_string,from_ostring,from_hstring)

Table 60 VHDL 2008 Supported IEEE Packages (Continued)

IEEE Packages	Supported Data Types	Supported Operations	Supported Functions and Procedures
float_pkg	1.float, UNRESOLVED_float 2.float32, UNRESOLVED_float32 3.float64, UNRESOLVED_float64 4.float128, UNRESOLVED_float128	<ul style="list-style-type: none"> arithmetic (+,-,*,/,rem, mod, abs) logical (and, nand, or, nor, xor, xnor, not) comparison (>,<,=,/=,>=,<=) matching relational (?>,<?=,?/=?>=?<=) shift and rotate (sll,srl,rol,ror) 	<ul style="list-style-type: none"> validity check (Classfp) arithmetic (add, subtract, multiply, divide, reciprocal, remainder, modulo, dividebyp2, mac, sqrt, Copysign, Scalb, Logb, Nextafter, Unordered, Finite, Isnan, zerofp, nanfp, qnanfp, pos_inffp, neg_inffp, neg_zerofp) comparison (eq,ne,lt,gt,le,ge, std_macth, find_rightmost, find_leftmost, maximum, minimum) shift and rotate (SHIFT_LEFT, SHIFT_RIGHT, ROTATE_LEFT, ROTATE_RIGHT) Resize (RESIZE) Conversion (to_float32, to_float64, to_float128, to_slv, to_suv, to_float, to_unsigned, to_signed, to_ufixed, to_sfixed, to_real, to_integer, realtobits, bitstoreal, break_number, normalize) Result range generation (ufixed_high, ufixed_low, sfixed_high, sfixed_low, UFix_high, UFix_low, SFix_high, SFix_low, saturate)
			<ul style="list-style-type: none"> Translation (TO_01, TO_X01, TO_X01Z, TO_UX01, IS_X) String conversion (to_string, to_ostring, to_hstring, READ, WRITE, OREAD, OWRITE, HREAD, HWRITE, from_string, from_ostring, from_hstring)

Resolved Elements

VCS supports resolved elements, as defined in *VHDL 2008 LRM Section 6.3*.

The support for resolution is enhanced to include array elements and record elements.

Using this enhancement, `std_logic_vector` is automatically converted to `std_ulogic_vector` (or vice versa). This indicates that `std_logic_vector` is now a subtype of `std_ulogic_vector`.

The following is the general syntax for a subtype declaration:

```

subtype_declaration ::= 
    subtype identifier is subtype_indication ;
subtype_indication ::= 
    [ resolution_indication ] type_mark [ constraint ]
resolution_indication ::= 
    resolution_function_name | ( element_resolution )

element_resolution ::= array_element_resolution | record_resolution

array_element_resolution ::= resolution_indication

record_resolution ::= record_element_resolution { ,
    record_element_resolution }

record_element_resolution ::= record_element_simple_name
    resolution_indication

type_mark ::= 
    type_name
    | subtype_name

constraint ::= 
    range_constraint
    | array_constraint
    | record_constraint

element_constraint ::= 
    array_constraint
    | record_constraint

```

Usage Example

The following is an example for array elements resolution:

```

entity DUT is
    port(in1,in2,in3 : in bit_vector(0 to 7);
        out1 : out bit_vector(0 to 7)
    );
end entity DUT ;

architecture arch of DUT is
    type bit_vector_vector is array (natural range <>) of bit_vector(0 to
7);
    function func1 (arg : bit_vector) return bit is
        variable result : bit := '0';

```

```

begin
    for i in arg'range loop
        result := result or arg(i);
    end loop;
    return result;
end function func1;
-- subtype declaration with array elements
subtype res_bit_nest is ((func1)) bit_vector_vector(0 to 0);
signal sig : res_bit_nest;
begin
sig(0) <= in1;
sig(0) <= in2;
sig(0) <= in3;
out1 <= sig(0);
end architecture arch;

entity top is
end entity top;

architecture arch of top is
    signal in1,in2,in3,out1 : bit_vector(0 to 7);
begin
inst : entity work.DUT(arch) port map(in1,in2,in3,out1);

process
begin
    in1 <= "11000011"; in2 <= "10101010"; in3 <= "10011001";
    wait for 1 ns;
    report to_string(out1);
    wait;
end process;
end architecture arch;

```

To run the example, use the following commands:

```
% vhdlan -vhdl108 test.vhdl
% vcs top
% simv
```

The following is the output:

```
1 NS
Report NOTE at 1 NS in design unit TOP(ARCH) from process /TOP/_P0:
"11111011"
(simv): Simulation complete, time is 1 NS.
```

The following is an example for record elements resolution:

```

library ieee;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;

package pack is
```

```

type bit_vector_vector is array (natural range <>) of bit_vector(0 to
3);
type signed_vector is array (natural range <>) of signed(0 to 3);
function wor (arg : bit_vector) return bit;
function wxor (arg : signed) return std_ulogic;
type REC is record
    E1 : bit_vector_vector(0 to 0);
    E2 : signed_vector(0 to 0);
end record;
-- subtype declaration with record elements
subtype resolved_rec is (E1((wor)),E2((wxor))) REC;
end package pack;

package body pack is

function wor (arg : bit_vector) return bit is
    variable result : bit := '0';
begin
    for i in arg'range loop
        result := result or arg(i);
    end loop;
    return result;
end function wor;

function wxor (arg : signed) return std_ulogic is
    variable result : std_ulogic := '0';
begin
    for i in arg'range loop
        result := result nor arg(i);
    end loop;
    return result;
end function wxor;

end package body pack;

library ieee;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;
use work.pack.all;

entity DUT is
    port(in1,in2 : in bit_vector(0 to 3);
         in3,in4 : in signed(0 to 3);
         out1 : out bit_vector(0 to 3);
         out2 : out signed(0 to 3)
    );
end entity DUT;

architecture arch of DUT is
    signal sig : resolved_rec;
begin

    sig.E1(0) <= in1;

```

```

sig.E1(0) <= in2;
sig.E2(0) <= in3;
sig.E2(0) <= in4;
out1 <= sig.E1(0);
out2 <= sig.E2(0);

end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;

entity top is
end entity top ;

architecture arch of top is
    signal in1,in2,out1 : bit_vector(0 to 3);
    signal in3,in4,out2 : signed(0 to 3);
begin

inst : entity work.DUT(arch) port map (in1,in2,in3,in4,out1,out2);

process
begin
    in1 <= "1100"; in2 <= "1010"; in3 <= "1001"; in4 <= "1101";
    wait for 1 ns;
    report "out1 = " & to_string(out1) & " out2 = " & to_string(out2);
    wait;
end process;
end architecture arch;

```

To run the example, use the following commands:

```

Command lines to run
% vhdlan -vhdl108 test.vhdl
% vcs top
% simv

```

The output is as follows:

```

1 NS
Report NOTE at 1 NS in design unit TOP(ARCH) from process /TOP/_P0:
    "out1 = 1110 out2 = 0000"

```

Conditional and Selected Assignments

VCS supports the conditional and selected assignments, as defined in *VHDL 2008 LRM Sections 10.5 and 10.6*.

According to VHDL 2008 LRM, you can use conditional and selected signal and variable assignments inside processes and subprograms, which allows the consistency between sequential and concurrent assignments.

The conditional signal assignment represents an equivalent if statement that assigns values to signals or that forces or releases signals. The general syntax is as follows:

```
conditional_signal_assignment ::=  

  conditional_waveform_assignment  

  | conditional_force_assignment  

conditional_waveform_assignment ::=  

  target <= [ delay_mechanism ] conditional_waveforms ;  

conditional_waveforms ::=  

  waveform when condition  

  { else waveform when condition }  

  [ else waveform ]  

conditional_force_assignment ::=  

  target <= force [ force_mode ] conditional_expressions ;  

conditional_expressions ::=  

  expression when condition  

  { else expression when condition }  

  [ else expression ]
```

The selected signal assignment represents an equivalent case statement that assigns values to signals or that forces or releases signals. The syntax is as follows:

```
selected_signal_assignment ::=  

  selected_waveform_assignment  

  | selected_force_assignment  

selected_waveform_assignment ::=  

  with expression select [ ? ]  

  target <= [ delay_mechanism ] selected_waveforms ;  

selected_waveforms ::=  

  { waveform when choices , }  

  waveform when choices  

selected_force_assignment ::=  

  with expression select [ ? ]  

  target <= force [ force_mode ] selected_expressions ;  

selected_expressions ::=  

  { expression when choices , }  

  expression when choices
```

For more information about the syntax and the functionality, see the *IEEE Standard VHDL 1076-2008 LRM*.

Use Model

To use Conditional and selected assignments in your design source code, specify the `-vhdl08` option in the `vhdlan` command line, or enable VHDL 2008 using the `VHDL_MODE=vhdl08` variable in the `synopsys_sim.setup` file.

Usage Example

The following example illustrates the usage of conditional signal assignment inside process:

```

library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_signed.all;

entity counter is
  port(
    clk    : in std_logic;
    rst    : in std_logic;
    count : out std_logic_vector(3 downto 0)
  );
end entity counter;

architecture arch of counter is
begin
  process(clk)
  begin
    if (rising_edge(clk)) then
      --- conditional signal assignment
      count <= (others=>'0') when rst else count + 1;
    end if;
  end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_textio.all;
use std.textio.all;

entity tb is
end entity tb;

architecture arch of tb is
  signal clk,rst : std_logic;
  signal count : std_logic_vector(3 downto 0);
begin
  inst : entity work.counter(arch) port map(clk,rst,count);
  process
  begin
    rst <= '0';
    wait for 5 ns;
    rst <= '1';
    wait for 5 ns;
    rst <= '0';
    wait for 100 ns;
    wait;
  end process;

```

```

process
begin
  if( now /= 100 ns) then
    clk <= '0'; wait for 5 ns;
    clk <= '1'; wait for 5 ns;
  else
    wait;
  end if;
end process;

process(count)
  variable l : line;
begin
  write(l,justify("@time = " & to_string(now) & " Count = " &
to_string(count)));
  writeline(output,l);
end process;
end architecture arch;

```

To run the example, use the following commands:

```
% vhdlan -vhdl08 test.vhdl
% vcs tb
% simv
```

The output is as follows:

```

@time = 0 ns Count = UUUU
@time = 5 ns Count = 0000
@time = 15 ns Count = 0001
@time = 25 ns Count = 0010
@time = 35 ns Count = 0011
@time = 45 ns Count = 0100
@time = 55 ns Count = 0101
@time = 65 ns Count = 0110
@time = 75 ns Count = 0111
@time = 85 ns Count = 1000
@time = 95 ns Count = 1001

```

The following example illustrates the usage of selected signal assignment inside process:

```

library ieee;
use ieee.std_logic_1164.all;

entity dut is
  port (
    in1 , in2 : in std_logic;
    sel : in std_logic;
    out1 : out std_logic
  );
end entity dut;

architecture arch of dut is

```

```

begin
  process(all)
  begin
    with sel select          -- selected signal assignment
      out1 <= in1 and in2 when '0',
      in1 or in2 when '1',
      'Z' when others;
  end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_textio.all;
use std.textio.all;

entity tb is
end entity tb;

architecture arch of tb is
  signal in1,in2,sel,out1 : std_logic;
begin
  inst : entity work.dut(arch) port map(in1,in2,sel,out1);
  process
    variable l : line;
  begin
    in1 <= '0'; in2 <= '1'; sel <= '0';
    wait for 1 ns;
    write(l,"out1 = " & to_string(out1));
    writeline(output,l);
    in1 <= '0'; in2 <= '1'; sel <= '1';
    wait for 1 ns;
    write(l,"out1 = " & to_string(out1));
    writeline(output,l);
    wait;
  end process;
end architecture arch;

```

To run the example, use the following commands:

```
% vhdlan -vhdl08 test.vhdl
% vcs tb
% simv
```

The output is as follows:

```
out1 = 0
out1 = 1
```

Note:

Conditional and selected assignments are also supported for variables.

Limitation

The following is the limitation for conditional and selected assignments feature:

- VHDL 2008 condition operator ?? is not yet supported for condition coverage.

Context Declaration

VCS supports the context declaration and context clauses, as defined in *VHDL 2008 LRM Sections 13.3 and 13.4*.

Based on the VHDL 2008 LRM, you can define context declaration and its context clauses which can be referenced by design units. This allows you to gather a collection of libraries and use clauses in one declaration instead of repeating the collection for every design unit.

The benefits of context declaration are:

- You can avoid writing multiple long lists of library and use clauses in large design.
- You can organize contexts of design units that share common library and use clauses in a more efficient and consistent manner.
- You can implement built-in contexts for common purposes.

The following is the general syntax for a context declaration:

```
context_declaration ::=  
    context identifier is  
        context_clause  
    end [ context ] [ context_simple_name ] ;  
  
context_clause ::= { context_item }  
  
context_item ::=  
    library_clause  
    | use_clause  
    | context_reference  
  
context_reference ::=  
    context selected_name { , selected_name } ;
```

Usage Example

The following example shows the syntax of a context declaration:

File1.vhd

```
context context_clause1 is  
    library ieee;
```

```

use ieee.std_logic_1164.all;
end context context_clause1;

File2.vhd

context context_clause2 is
    library lib1;
    context lib1.context_clause1;
    library ieee;
    use ieee.std_logic_arith.all;
    use std.textio.all;
    use ieee.std_logic_textio.all;
end context context_clause2;

context work.context_clause2;

entity dut is
    port(
        in1 : in unsigned(0 to 3);
        in2 : in unsigned(0 to 3);
        out1 : out unsigned(0 to 3)
    );
end entity dut;

architecture arch of dut is
begin
    process(in1, in2)
    begin
        case in1(0) is
            when '1' => out1 <= in1 + in2;
            when '0' => out1 <= in2 - in1;
            when others => out1 <= in1;
        end case;
    end process;
end architecture arch;

context work.context_clause2;

entity top is
end entity top;

architecture arch of top is
    signal in1,in2,out1 : unsigned(0 to 3);
begin
    inst : entity work.dut(arch) port map(in1,in2,out1);

    process
        variable l : line;
    begin
        in1 <= "1101" ; in2 <= "1001";
        wait for 1 ns;
        write(l,"out1 = " & to_string(conv_std_logic_vector(out1,4)));
    end process;
end architecture arch;

```

```
writeline(output,1);
wait;
end process;
end architecture arch;
```

synopsys_sim.setup file

```
WORK > DEFAULT
DEFAULT : ./work
lib1 : ./lib1
```

To run the example, use the following commands:

```
% mkdir work lib1
% vhdlan -vhdl08 file1.vhd -work lib1
% vhdlan -vhdl08 file2.vhd
% vcs top
% simv
```

The following is the output:

```
out1 = 0110
(simv): Simulation complete, time is 1 NS.
```

Improved I/O

VHDL provides a set of enhancements to the I/O functions and string conversion features. You can easily debug the signal value change and also use read and write procedures by implementing these new functions and features.

The support for textual I/O is enhanced by including operations for all the standard types in the respective IEEE 2008 packages as defined in *VHDL 2008 LRM Section 16.4*. This support is also extended for octal and hexadecimal I/O to read and write data by enhancing the string I/O capability.

The following is the list of functions:

- **TO_STRING (x)** function - This function provides the ability to work with any scalar type and any one-dimensional array type where the element type is an enumerated type all of whose literals are character literals.
- Justify function - This function provides the ability to justify a string representation in a field of given width. It is useful for tabular formatting of output.
- Octal Read/write operations - This function provides the octal read and write capability. If the number of bits to be read is not divisible by three, then the number read is resized accordingly. In the case of write operation, padding bits are included.

- Hexadecimal Read/write operations - This function provides the hexadecimal read and write capability. If the number of bits is not divisible by four, then the number read is resized accordingly. In the case of write operation, padding bits are included.
- Tee procedure - It provides the ability to write a line both to the file output and to a separate named file and allows to avoid replicated write operations.
- Flush procedure - It provides the ability to effect all previous calls to the write procedure for a file be completed.

Usage Example

The following is an example for some of the I/O functions:

```

library ieee;
use ieee.std_logic_1164.all,ieee.std_logic_unsigned.all;

entity counter is
    port(
        clk,rst,en : in std_logic;
        count : out std_logic_vector(3 downto 0)
    );
end entity counter;

architecture arch of counter is
begin
    process(clk,rst)
    begin
        if(rst) then
            count <= (others => '0');
        elsif(rising_edge(clk)) then
            if(en) then
                count <= count + 1;
            end if;
        end if;
    end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
use std.textio.all;

entity top is
end entity top;

architecture arch of top is
    signal clk,rst,en : std_logic;
    signal count : std_logic_vector(3 downto 0);
begin
    counter_inst : entity work.counter(arch) port map(clk,rst,en,count);
    clk_proc : process

```

```

begin
    if now = 100 ns then
        wait;
    else
        clk <= '0'; wait for 5 ns;
        clk <= '1'; wait for 5 ns;
    end if;
end process clk_proc;

process
begin
    rst <= '1'; en <= '0'; wait for 3 ns;
    rst <= '0'; en <= '1'; wait for 100 ns;
    wait;
end process;

process(count)
    variable wl,rl : line;
    variable var : std_logic_vector(3 downto 0);
    file fpw : text open WRITE_MODE is "file.out";
    file fpr : text open READ_MODE is "file.out";
begin
    write(wl,justify(to_string(now,ns),right,10));
    write(wl,justify(to_string(count),right,15));
    tee(fpw,wl);
end process;

end architecture arch;

```

To run the example, use the following commands:

```
% vhdlan -vhdl108 test.vhdl
% vcs top
% simv
```

The output is as follows:

0ns	UUUU
0ns	0000
5ns	0001
15ns	0010
25ns	0011
35ns	0100
45ns	0101
55ns	0110
65ns	0111
75ns	1000
85ns	1001
95ns	1010

Support for Implicitly Constrained Array Elements

VHDL provides two kinds of composite types, namely, arrays and records. All the elements are of the same type in the case of array elements. The elements are of different types in the case of record elements.

VCS supports the use of implicitly constrained array element types as defined in VHDL 2008 LRM, section 5.3.

The following element types are supported from this release:

- Partially constrained subtypes
- Implicitly constrained array subtypes on:
 - Ports
 - Constants
 - Generics
 - Parameters

Usage Example

```
package global is
    type BitVecArrTyp is array(natural range <>) of bit_vector;
end package global;

entity ent1 is
    port(in1, in2 : in work.global.BitVecArrTyp;
         out1 : out work.global.BitVecArrTyp);
end entity ent1;

architecture arch of ent1 is
begin
    process(in1, in2)
    begin
        for idx in in1'range loop
            out1(idx) <= in1(idx) and in2(idx);
        end loop;
    end process;
end architecture arch;

entity top is
end entity top;

architecture arch of top is
    signal in1, in2, out1 : work.global.BitVecArrTyp(0 to 1)(3 downto 0);
begin
    ent1_inst : entity work.ent1(arch) port map (in1, in2, out1);
```

```

in1 <= ("1010", "0101");
in2 <= ("1110", "0111");

process (out1)
begin -- process
    report "out1 : ("" & to_string(out1(0)) &
            "", "" & to_string(out1(1)) & "")";
end process;
end architecture arch;

```

The following are the commands to run the test case:

```

vhdlan -vhdl08 DocExample.vhd
vcs top
./simv

```

Output

```

Report NOTE at 0 NS in design unit TOP(ARCH) from process /TOP/_P2:
    "out1 : ("0000", "0000")"
Report NOTE at 0 NS in design unit TOP(ARCH) from process /TOP/_P2:
    "out1 : ("1010", "0101")"
(simv): Simulation complete, time is 0.

```

Support for Unconstrained Element Types

VCS supports the use of unconstrained element types as defined in the *VHDL 2008 LRM, Section 5.3*. This feature is enabled with the `-vhdl08` switch at the `vhdlan` stage.

The following element types are supported:

- Partially constrained array subtypes
- Implicitly constrained array subtypes on,
 - Ports
 - Constants
 - Generics
 - Parameters
 - New built-in array attributes
- Unconstrained or partial array element constrained aliases
- Unconstrained array element access type
- Unconstrained array element type external names

- Explicitly constrained record element subtypes on,
 - Ports
 - Constants
 - Generics
 - Parameters

The following is the general syntax for definition of an array type and a record type:

Array Type:

```

array_type_definition ::= 
    unbounded_array_definition |      constrained_array_definition

unbounded_array_definition ::= 
    array ( index_subtype_definition { ,      index_subtype_definition } ) 
        of element_subtype_indication

constrained_array_definition ::= 
    array index_constraint of element_subtype_indication

index_subtype_definition ::= type_mark range <>

array_constraint ::= 
    index_constraint [ array_element_constraint ]
    | ( open ) [ array_element_constraint ]

array_element_constraint ::= element_constraint

index_constraint ::= ( discrete_range { , discrete_range } )

discrete_range ::= discrete_subtype_indication | range

```

Record Type:

```

record_type_definition ::= 
    record
        element_declaration
        { element_declaration }
    end record [ record_type_simple_name ]

element_declaration ::= 
    identifier_list : element_subtype_definition ;

identifier_list ::= identifier { , identifier }

element_subtype_definition ::= subtype_indication

record_constraint ::= 
    ( record_element_constraint { ,      record_element_constraint } )

```

```
record_element_constraint ::= record_element_simple_name
                           element_constraint
```

Usage Example

The following is an example for unconstrained array element type:

Example 216 Unconstrained element type

```
library ieee;
use ieee.std_logic_1164.all;

package global is
  type StdVecArrTyp is array(natural range <>) of std_logic_vector;
end package global;

library ieee;
use ieee.std_logic_1164.all;

entity ent1 is
  port(
    in1 : in work.global.StdVecArrTyp(open)(open);
    in2 : in work.global.StdVecArrTyp(open)(open);
    out1 : out work.global.StdVecArrTyp(open)(open)
  );
end entity ent1;

architecture arch of ent1 is
begin
  process(in1,in2)
  begin
    for idx in in1'range loop
      out1(idx) <= in1(idx) and in2(idx);
    end loop;
  end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;

entity top is
end entity top;

architecture arch of top is
  signal in1,in2,out1 : work.global.StdVecArrTyp(0 to 1)(3 downto 0);
begin
  ent1_inst : entity work.ent1(arch)
    port map(in1 => in1, in2 => in2, out1 => out1);
  process
    use ieee.std_logic_textio.all;
    use std.textio.all;
    variable l : line;
```

```

begin
    in1(0) <= "1101"; in1(1) <= "1111";
    in2(0) <= "1101"; in2(1) <= "1001";
    wait for 1 ns;
    for i in in1'range loop
        write(l,string'(" in1(") & to_string(i) & string'(")      : ") &
to_string(in1(i)));
        write(l,string'(" in2(") & to_string(i) & string'(")      : ") &
to_string(in2(i)));
        write(l,string'(" out1(")& to_string(i) & string'(")      : ") &
to_string(out1(i)));
        writeline(output,l);
    end loop;
    wait;
end process;
end architecture arch;

```

The following are the commands to execute:

```
% vhdlan -vhdl108 test.vhd
% vcs top
% simv
```

Output

```
in1(0) : 1101 in2(0) : 1101 out1(0) : 1101
in1(1) : 1111 in2(1) : 1001 out1(1) : 1001
(simv): Simulation complete, time is 1 NS.
```

The following is an example of unconstrained record element type:

Example 217 Unconstrained record element type

```

library ieee;
use ieee.std_logic_1164.all;

package pkg is
    type slv1d is array(natural range <>) of std_logic_vector;
    type rec_typ is record
        r0 : slv1d;
        r1 : std_logic_vector;
    end record rec_typ;
end package pkg;

library ieee;
use ieee.std_logic_1164.all;
use work.pkg.all;

entity ent is
    port (
        in1 : in rec_typ(r0(0 to 1)(3 downto 0),r1(3 downto 0));
        out1 : out rec_typ(r0(0 to 1)(3 downto 0),r1(3downto 0))
    );

```

```

end entity ent;

architecture arch of ent is
begin
    process(in1)
    begin
        for i in in1.r0'range loop
            for j in in1.r0'element'range loop
                out1.r0(i)(j) <= not in1.r0(i)(j);
            end loop;
        end loop;
        out1.r1 <= not in1.r1;
    end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_textio.all;
use std.textio.all;
use work.pkg.all;

entity top is
end entity top;

architecture arch of top is
    signal in1,out1 : rec_typ(r0(0 to 1)(3 downto 0),r1(3      downto 0));
begin
    inst : entity work.ent(arch) port map(in1,out1);
    process
        variable l : line;

    begin
        in1.r0(0) <= "1101"; in1.r0(1) <= "1111"; in1.r1 <=      "0101";

        wait for 1 ns;
        for i in out1.r0'range loop
            write(l,string'(" out1.r0(") & to_string(i) &
            string(') : ") & to_string(out1.r0(i)));
        end loop;
        write(l,string'(" out1.r1 : ") & to_string(out1.r1));
        writeline(output,l);
        wait;
    end process;
end architecture arch;

```

The following are the commands to execute:

```
% vhdlan -vhdl08 test.vhd
% vcs top
% simv
```

Output

```
out1.r0(0) : 0010 out1.r0(1) : 0000 out1.r1 : 1010
```

Limitations

The following are the limitations for unconstrained element types:

- ▶ Unbounded records constrained by expression or association are not supported for the following objects:

- Ports, parameters, generic constants, alias, access, external, and type conversion

This includes parameters to operator overload and therefore comparison of aggregates of implicitly constrained unbounded record type is not supported.

1. Record aggregates assigned to record objects and those associated with record interface elements that have been explicitly constrained are not supported.
2. Formal sub-element association of record elements where the record element is constrained at the point of the interface element declaration (Port/Parameter/Generic).
3. Type conversion for unconstrained element type(arrays or records) is not supported.
4. Unbounded record and array element types associated with type generics are not supported.

Support for Enhanced Generics in Entity Interfaces

VCS supports the use of enhanced generics in entity interfaces as defined in Section 6.5 of *VHDL 2008 LRM*.

With the support of enhanced generics in VHDL 2008, you can declare generic types, sub-programs, packages, and generic constants in an entity interface. This helps in declaring abstract and reusable entities with different types and design specialization.

Usage Example

These examples show the use of generic types and functions.

Example 1:

Using a Generic Type in an Entity Interface:

Example 218 Usage of generic type in entity interface

```
entity gen_mux is
    generic( type T );
    port(
        sel    : in  bit;
        I0,I1 : in  T ;
```

```

      Y      : out T
    );
end entity gen_mux;

architecture arch of gen_mux is
begin
  Y <= I0 when sel else I1;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
use std.textio.all;
use ieee.std_logic_textio.all;

entity top is
end entity top;

architecture arch of top is
  signal sel : bit;
  signal a,b,Y : std_logic_vector(2 downto 0);
begin
  inst : entity work.gen_mux(arch)
    generic map (
      T => std_logic_vector(2 downto 0) )
    port map(
      sel => sel,
      I0 => a,
      I1 => b,
      Y => Y
    );
  Process
    Variable l : line;
Begin
  a <= "110"; b <= "011"; sel <= '1';
  wait for 1 ns;
  write(l,"sel = " & to_string(sel) & " a = " &
        to_string(a) & " b = " & to_string(b) & " y =
        " & to_string(y));
  writeline(output,l);
  a <= "110"; b <= "011"; sel <= '0';
  wait for 1 ns;
  write(l,"sel = " & to_string(sel) & " a = " &
        to_string(a) & " b = " & to_string(b) & " y =
        " & to_string(y));
  writeline(output,l);
  wait;
  end process;
end architecture arch;

```

Following are the commands to run the example:

```
vhdlan -vhdl08 test.vhd
vcs top
./simv
```

Output

```
sel = 1 a = 110 b = 011 y = 110
sel = 0 a = 110 b = 011 y = 011
```

Example 2:

Using a Sub-program in an Entity Interface:

Example 219 Usage of sub-program in entity interface

```
library ieee;
use ieee.std_logic_1164.all;

entity gen_counter is
    generic( type T; constant C : T ; function      count_increment(in1 :
T) return T );
    port(
        clk,rst : in std_logic;
        count : out T
    );
end entity gen_counter;

architecture arch of gen_counter is
begin
    process(clk,rst)
        variable rst_value : T := C;
    begin
        if(rst) then
            count <= C;
        elsif(rising_edge(clk)) then
            count <=count_increment(count);
        end if;
    end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_unsigned.all;
use std.textio.all;
use ieee.std_logic_textio.all;

entity top is
end entity top;

architecture arch of top is
    signal clk,rst: std_logic;
```

```

signal count : std_logic_vector(1 downto 0);
function increment(in1 : in std_logic_vector) return std_logic_vector
is
begin
    begin
        return (in1 + 1);
    end function increment;
begin
    inst : entity work.gen_counter(arch)
generic map (
    T => std_logic_vector(1 downto 0),
    C => (others => '0'),
    count_increment => increment
)
port map(
    clk => clk,
    rst => rst,
    count => count
);
    process
    begin
        if(now = 50 ns) then
            wait;
        else
            clk <= '0'; wait for 5 ns;
            clk <= '1'; wait for 5 ns;
        end if;
    end process;
Process
begin
    rst <= '1';
    wait for 3 ns;
    rst <= '0';
    wait for 101 ns;
    wait;
end process;
process(count)
Variable l : line;
begin
    write(l, "rst = " & to_string(rst) & " clk = " & to_string(clk) & "
count = " & to_string(count)&" time = " & to_string(now) );
    writeline(output,l);
end process;
end architecture arch;

```

Following are the commands to run the example:

```

vhdlan -vhdl08 -q test.vhd
vcs top
./simv

```

Output

```
rst = U clk = U count = UU time = 0 ns
rst = 1 clk = 0 count = 00 time = 0 ns
rst = 0 clk = 1 count = 01 time = 5 ns
rst = 0 clk = 1 count = 10 time = 15 ns
rst = 0 clk = 1 count = 11 time = 25 ns
rst = 0 clk = 1 count = 00 time = 35 ns
rst = 0 clk = 1 count = 01 time = 45 ns
```

Example 3

Using the Generic Package Instantiation in an Entity Interface:

Example 220 Usage of generic package instantiation in an entity interface

```
package pkg is
    generic(type T);
        function func(sel : in bit; in1,in2 : in T) return T ;
    end package pkg;

package body pkg is
    function func(sel : in bit; in1,in2 : in T) return T is
    begin
        if(sel) then
            return in1;
        else
            return in2;
        end if;
    end function func;
end package body pkg;

library ieee;
use ieee.std_logic_1164.all;

entity gen_mux is
    generic(type Te;package pkg_inst is new work.pkg
map(T=>Te));
    port(
        sel: in bit;
        in1,in2 : in Te;
        out1 : out Te
    );
end entity gen_mux;

architecture arch of gen_mux is
    use pkg_inst.all;
begin
    out1 <= func(sel,in1,in2);
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
```

```

package pkg_inst is new work.pkg generic map(
    T => std_logic_vector(1 downto 0)
);

library ieee;
use ieee.std_logic_1164.all;
use std.textio.all;
use ieee.std_logic_textio.all;

entity top is
end entity top;

architecture arch of top is
    signal sel : bit;
    signal in1,in2,out1,temp: std_logic_vector(1 downto 0);
begin
    inst : entity work.gen_mux(arch)
        generic map (Te => std_logic_vector(1 downto 0),pkg_inst =>
work.pkg_inst)
            port map(
                sel => sel,
                in1 => in1,
                in2 => in2,
                out1 => out1
            );
    Process
    begin
        sel <= '1'; in1 <= "00";in2<= "01";
        wait for 3 ns;
        sel <= '0'; in1 <= "10";in2<= "11";
        wait for 3 ns;
        wait;
    end process;

    process(out1)
        variable l : line;
    begin
        write(l,"sel = "&to_string(sel)&" in1 =
"&to_string(in1)&" in2 = "&to_string(in2)&" out1 = "&to_string(out1)&" time =
"&to_string(now));
        writeline(output,l);
    end process;
end architecture arch;

```

Following are the commands to run the example:

```
% vhdlan -vhdl08 test.vhd
% vcs top
% simv
```

Output

```
sel = 0 in1 = UU in2 = UU out1 = UU time = 0 ns
sel = 1 in1 = 00 in2 = 01 out1 = 00 time = 0 ns

sel = 0 in1 = 10 in2 = 11 out1 = 11 time = 3 ns
```

Limitation

Following is a limitation for this feature:

- Enhanced generics are not supported in component declarations.

Support for Slices in Array Aggregates

VHDL 2008 supports the use of slices in array aggregates as defined in Section 9.3.3.3 of VHDL 2008 LRM.

An array aggregate allows you to create an array from a collection of various elements. In VHDL 2008, you can form array aggregates from a combination of individual elements and slices of the array.

VHDL 2008 also allows you to write an aggregate as the target of signal assignment statement. The names in the aggregate can be a mixture of element-typed signals and array-typed signals.

An example of bit_vector array aggregate can be written as:

`('1', "1100", '0')`

This forms a six-element vector from the single element '1', the vector value "1100", and the single element '0'. The vector value forms a slice of the final aggregate value.

Usage Example

Example 221 SliceAggrEx1.vhd

```
Library ieee;
Use ieee.std_logic_1164.all;

Entity ent is
End entity ent;

Architecture arch of ent is
signal Sig1:std_logic_vector(7 downto 0) := ('0',"011011","1");
Begin
    process begin
        wait for 1 ns;
        report "sig1:" & to_string(Sig1);
        wait;
```

```
    end process;
End architecture arch;
```

The following are the commands to run the test case:

```
% vhdlan -vhdl08 -q sliceAggrEx1.vhd
% vcs ent -q
% simv -q
```

Output

```
Report NOTE at 1 NS in design unit ENT(ARCH) from process /ENT/_P0:
"sig1:00110111"
```

Example 2:

Example 222 SliceAggrEx2.vhd

```
Library ieee;
Use ieee.std_logic_1164.all;
use ieee.numeric_std.all;
Entity ent is
Port (
    a, b : in unsigned(7 downto 0);
    sum : out unsigned(7 downto 0);
    c_out :out std_ulogic
);
End entity ent;

Architecture arch of ent is
Begin
    (c_out,sum) <= ('0' & a ) + ('0' & b);
End architecture arch;

Library ieee;
Use ieee.std_logic_1164.all;
use ieee.numeric_std.all;
entity tb is
end entity;

architecture a of tb is
    signal a, b , sum : unsigned(7 downto 0);
    signal c_out : std_ulogic;
begin
    process begin
        a <= "00101110";
        b <= "10110110";
        wait for 5 ns;
        a <= "00101110";
        b <= "10110110";
        wait for 5 ns;
        wait;
```

```

    end process;

    inst : entity work.ent port map (a => a , b=> b , sum => sum , c_out
=> c_out);

        process(c_out,sum) begin
            report "c_out=" & to_string(c_out) & " sum=" & to_string(sum);
        end process;
end architecture;
```

The following are the commands to run the test case:

```
% vhdlan -vhdl108 -q sliceAggrEx2.vhd
% vcs tb -q
% simv -q
```

Output

```
Report NOTE at 0 NS in design unit TB(A) from process /TB/_P1:
"c_out=U sum=UUUUUUUU"
Report NOTE at 0 NS in design unit TB(A) from process /TB/_P1:
"c_out=X sum=XXXXXXXX"
Report NOTE at 0 NS in design unit TB(A) from process /TB/_P1:
"c_out=0 sum=11100100"
```

Example 3:

This example shows the usage of inner most sub-aggregate of a multidimensional array aggregate.

Example 223 Usage of inner most sub-aggregate

```

Library ieee;
Use ieee.std_logic_1164.all;
use ieee.numeric_std.all;
entity tb is
end entity;

architecture a of tb is
    signal clk : bit := '0';

    type typeArr2D is array (natural range 0 to 1 )of std_logic_vector(3
downto 0);
    type typeArr3D is array (natural range <>)of typeArr2D;

    constant sigArr2D_1 : typeArr2D := (0 => "0001" , 1 => "0010" );
    constant sigArr3D_1 : typeArr3D(1 downto 0) :=
(sigArr2D_1 ,sigArr2D_1 );
    signal sigArr3D_2 : typeArr3D(5 downto 0);
    signal sigArr3D_3 : typeArr3D(2 downto 0) := ( sigArr2D_1 ,
sigArr3D_1);

begin
```

```

process(clk) begin
    if (clk = '1')then
        sigArr3D_2 <= (sigArr2D_1,sigArr3D_1,sigArr3D_3);
    end if;
end process;

process begin
    wait for 5 ns;
    clk <= '1';
    wait for 5 ns;
    report "sigArr3D_2(0) [0=> (" & to_string(sigArr3D_2(0)(0)) &
") , 1=( " & to_string(sigArr3D_2(0)(1)) & ")";
    report "sigArr3D_2(1) [0=> (" & to_string(sigArr3D_2(1)(0)) &
") , 1=( " & to_string(sigArr3D_2(1)(1)) & ")";
    report "sigArr3D_2(2) [0=> (" & to_string(sigArr3D_2(2)(0)) &
") , 1=( " & to_string(sigArr3D_2(2)(1)) & ")";
    report "sigArr3D_2(3) [0=> (" & to_string(sigArr3D_2(3)(0)) &
") , 1=( " & to_string(sigArr3D_2(3)(1)) & ")";
    report "sigArr3D_2(4) [0=> (" & to_string(sigArr3D_2(4)(0)) &
") , 1=( " & to_string(sigArr3D_2(4)(1)) & ")";
    report "sigArr3D_2(5) [0=> (" & to_string(sigArr3D_2(5)(0)) &
") , 1=( " & to_string(sigArr3D_2(5)(1)) & "]";
    wait;
end process;
end architecture;

```

Output

```

Report NOTE at 10 NS in design unit TB(A) from process /TB/_P1:
    "sigArr3D_2(0) [0=> (0001) , 1=(0010)]"
Report NOTE at 10 NS in design unit TB(A) from process /TB/_P1:
    "sigArr3D_2(1) [0=> (0001) , 1=(0010)]"
Report NOTE at 10 NS in design unit TB(A) from process /TB/_P1:
    "sigArr3D_2(2) [0=> (0001) , 1=(0010)]"
Report NOTE at 10 NS in design unit TB(A) from process /TB/_P1:
    "sigArr3D_2(3) [0=> (0001) , 1=(0010)]"
Report NOTE at 10 NS in design unit TB(A) from process /TB/_P1:
    "sigArr3D_2(4) [0=> (0001) , 1=(0010)]"
Report NOTE at 10 NS in design unit TB(A) from process /TB/_P1:
    "sigArr3D_2(5) [0=> (0001) , 1=(0010)]"
(simv): Simulation complete, time is 10 NS.

```

Limitation

Following is the limitation for this feature:

- This feature is only applicable for one-dimensional array aggregate or the inner most sub-aggregate of a multidimensional array aggregate.

Support for Type Conversion in VHDL 2008

VHDL 2008 extends the support for use of different element types in the source and target array expression types.

VHDL 2008 supports the following enhancements in type conversion:

- An array can be converted to another array whether the index subtype of the target and operand is closely related or not. In the case of arrays, only the element types need to be closely related.
- If the target sub-type is an array type for which the index ranges are not defined, then the bounds of the result are checked whether they belong to the corresponding index subtype of the target type. If the target subtype is an array sub-type for which the index ranges are defined, then each element is checked for the operand if there is a matching element of the target sub-type.
- A record can be converted to its sub-type.

Usage Example

Example for Type Conversion

Example 224 Type conversion

```
entity ent is
end entity;

architecture arch of ent is
    type exp_type is (int, undef, trap, ovf, div0);
    type exp_vector is array (exp_type) of bit;

    signal d_bitVect: bit_vector(31 downto 0);
    signal exp_reg : exp_vector;

    signal realVect :real_vector(0 to 3);
    signal intVect :integer_vector(0 to 3);
begin

    d_bitVect(31 downto 27) <= bit_vector( exp_reg );
    intVect <=integer_vector(realVect);

    process begin
        wait for 1 ns;
        exp_reg <= "10010";
        realVect <= ( 0=> 1.25 , 1=>2.67 , 2=>2.85 , 3=>6.75);

        wait for 1 ns;
        report "exp_reg=" & to_string(exp_reg);
    end process;
end;
```

```

        report "d_bitVect(31 downto 27)=" & to_string(d_bitVect(31 downto
27));
        report "realVect=" & to_string(realVect(0)) & "," &
to_string(realVect(1)) & "," & to_string(realVect(1)) & "," &
to_string(realVect(2)) & ")" ;
        report "intVect=" & to_string(intVect(0)) & "," &
to_string(intVect(1)) & "," & to_string(intVect(1)) & "," &
to_string(intVect(2)) & ")" ;
        wait;
    end process;
end architecture;

```

The following are the commands to run the test case:

```
% vhdlan -vhdl108 -q test.vhd
% vcs ent -q
% simv -q
```

Output

```

2 NS
Report NOTE at 2 NS in design unit ENT(ARCH) from process /ENT/_P2:
  "exp_reg=10010"
Report NOTE at 2 NS in design unit ENT(ARCH) from process /ENT/_P2:
  "d_bitVect(31 downto 27)=10010"
Report NOTE at 2 NS in design unit ENT(ARCH) from process /ENT/_P2:
  "realVect=(1.25e+0,2.67e+0,2.67e+0,2.85e+0)"
Report NOTE at 2 NS in design unit ENT(ARCH) from process /ENT/_P2:
  "intVect=(1,3,3,3)"
(simv): Simulation complete, time is 2 NS.

```

Limitation

Following is the limitation for this feature:

- Type conversion is not supported for unconstrained array elements and record elements.

Support for Case Expression Subtype

VCS supports the following enhancements for VHDL 2008:

- The case expression need not have a locally static index range. It must have the choices that have same length as the case expression.
- An array aggregate containing others is allowed as a choice in a case statement, if the index range of the case expression is locally static.

Example for Case Expression

The following example shows case expression with concatenation (which are not locally static), but the case choices are locally static.

Example 225 Usage of case expression

```
entity ent is
end entity;

architecture arch of ent is
  signal sel1 : bit_vector (1 downto 0);
  signal sel2 : bit_vector (1 downto 0);
  signal out1 : bit_vector (3 downto 0);

  signal clk : bit := '0';
  signal endSim : bit := '0';
begin
  process (clk) begin
    if (clk = '1') then
      case (sel1 & sel2) is
        when "0000" => out1 <= "0000";
        when "0001" => out1 <= "0001";
        when "0010" => out1 <= "0010";
        when "0100" => out1 <= "0100";
        when "1000" => out1 <= "1000";
        when others => out1 <= "1111";
      end case;
    end if;
  end process;

  process begin
    wait for 4 ns;
    sel1 <= "00";
    sel2 <= "01";
    wait for 10 ns;
    sel1 <= "00";
    sel2 <= "10";
    wait for 10 ns;
    sel1 <= "01";
    sel2 <= "00";
    wait for 10 ns;
    sel1 <= "10";
    sel2 <= "00";
    wait for 10 ns;
    sel1 <= "11";
    sel2 <= "11";
    wait for 10 ns;
    endSim <= '1';
    wait;
  end process;

  process begin
```

```

clk <= '0';
wait for 5 ns;
clk <= '1';
wait for 5 ns ;
if (endSim = '1') then
    wait;
end if;
end process;

process (out1) begin
    report "out1 =" & to_string(out1);
end process;

end architecture;

```

The following are the commands to run the test case:

```
% vhdlan -vhdl08 -q test.vhd
% vcs ent -q
% simv -q
```

Output

```

Report NOTE at 0 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =0000"
5 NS
Report NOTE at 5 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =0001"
15 NS
Report NOTE at 15 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =0010"
25 NS
Report NOTE at 25 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =0100"
35 NS
Report NOTE at 35 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =1000"
45 NS
Report NOTE at 45 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =1111"
(simv): Simulation complete, time is 60 NS.
```

This example shows the case choices with concatenation (not locally static), but the case expression is locally static.

```

entity ent is
end entity;

architecture arch of ent is
    signal sell1 : bit_vector (3 downto 0);
    signal out1 : bit_vector (3 downto 0);
```

```

signal clk : bit :='0';
signal endSim : bit :='0';
begin
    process (clk) begin
        if (clk ='1') then
            case (sel1 ) is
                when "00"&"00" => out1 <= "0000";
                when '0' & "001" => out1 <= "0001";
                when "00"&"10" => out1 <= "0010";
                when "01"&"00" => out1 <= "0100";
                when "10"&"00" => out1 <= "1000";
                when others => out1 <= "1111";
            end case;
        end if;
    end process;

    process begin
        wait for 4 ns;
        sel1 <= "0001";
        wait for 10 ns;
        sel1 <= "0010";
        wait for 10 ns;
        sel1 <= "0100";
        wait for 10 ns;
        sel1 <= "1000";
        wait for 10 ns;
        sel1 <= "1111";
        wait for 10 ns;
        endSim <= '1';
        wait;
    end process;

    process begin
        clk <= '0';
        wait for 5 ns;
        clk <= '1';
        wait for 5 ns ;
        if (endSim = '1') then
            wait;
        end if;
    end process;

    process (out1) begin
        report "out1 =" & to_string(out1);
    end process;
end architecture;
```

The following are the commands to run the test case:

```
% vhdlan -vhdl08 -q test1.vhd
% vcs ent -q
% simv -q
```

Output

```

Report NOTE at 0 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =0000"
5 NS
Report NOTE at 5 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =0001"
15 NS
Report NOTE at 15 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =0010"
25 NS
Report NOTE at 25 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =0100"
35 NS
Report NOTE at 35 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =1000"
45 NS
Report NOTE at 45 NS in design unit ENT(ARCH) from process /ENT/_P3:
    "out1 =1111"
(simv): Simulation complete, time is 60 NS.

```

Support for Subtypes of Ports and Parameters

VCS supports the following enhancements for VHDL 2008:

- The subtype rules for scalar ports and scalar signal parameters are allowed to support runtime subtype checks.
- The subtype of the signal must be compatible with the subtype of the port for a port of mode ‘in’ or ‘inout’. Also, the subtype of the port must be compatible with the subtype of the signal for a port of mode ‘out’ or ‘inout’.

Example for Subtypes of Ports and Parameters

Example 226 Subtypes of ports and parameters

```

entity leaf is
    port (in1 : in integer;
          out1 : out positive );
end entity;

architecture arch of leaf is
begin

end architecture;

entity ent is
end entity;

architecture arch of ent is
    signal sigInt : positive;

```

```

signal sigOut : integer;
begin
    intsl : entity work.leaf port map( sigInt,sigOut);

    process begin
        sigInt <= 10;
        wait for 1 ns;
        report "sigOut=" & to_string(sigOut);
        wait;
    end process;
end architecture;
```

The following are the commands to run the test case:

```
% vhdlan -vhdl108 -q test.vhd
% vcs ent -q
% simv -q
```

Output

```
Report NOTE at 1 NS in design unit ENT(ARCH) from process /ENT/_P0:
    "sigOut=1"
(simv): Simulation complete, time is 1 NS.
```

Support for Static Composite Expressions

VCS provides additional support for the use of locally static expressions as defined in *VHDL 2008 LRM, Section 9.4*. A locally static expression can be of a composite type, if the subtype of each of the primaries in the expression is locally static and has a locally static subtype.

VCS expands the support for the following kinds of locally static expressions:

- Composite types, such as arrays and records.
- Allowed primaries in a locally expression can be array or record aggregates, indexed array elements, array slices, selected record elements and so on.
- The operators in an expression can either be any of the predefined operators or functions or those defined in the standard packages, such as `std_logic_1164`, `numeric_bit`, `numeric_std`, `numeric_bit_unsigned`, and `numeric_std_unsigned`.

Usage Examples

The following examples show the use of static composite expressions in case and select statements:

Use of Case Choice and Constant Initializations

Example 227 Usage of case choice and constant initializations

```

library ieee;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;
entity e is
port(
  p1:in bit_vector(3 downto 0);
  p2:in std_logic_vector(3 downto 0);
  p3:in unsigned(3 downto 0));
end entity;
architecture arch_e of e is
  constant kbv1:bit_vector(3 downto 0):=(others => '0');
  constant kbv2:bit_vector(3 downto 0):='0'&(kbv1(2 downto 0) xor
"001");

  constant kst1:std_logic_vector(3 downto 0):=(others => '0');
  constant kst2:std_logic_vector(2 downto
0):=To_StdLogicVector(kbv2(2 downto 0));

  constant kun1:unsigned(2 downto 0):=(others => '0');
  constant kun2:unsigned(3 downto 0):=resize(unsigned'("10"),4);
  constant kun3:unsigned(3 downto 0):=SHIFT_RIGHT("1010",1);

begin
  -- predefined types bit-vector
  process(p1)begin
    case(p1)is
      when kbv1                      => report
      "p1:bit_vector value is 0"; -- 0000
      when "00"&"01"                  => report
      "p1:bit_vector value is 1"; -- 0001
      when ("0001" sll 1)            => report
      "p1:bit_vector value is 2"; -- 0010
      when ("0111" and "1011")       => report
      "p1:bit_vector value is 3"; -- 0011
      when kbv2(1 downto 0)&"00"     => report
      "p1:bit_vector value is 4"; -- 0100
      when (2=>kbv2(0),0=>'1',others=>'0') => report
      "p1:bit_vector value is 5"; -- 0101
      when others                     => report
      "p1:bit_vector value is unknown";
      end case;
    end process;
  -- std_logic_vector

```

```

process(p2)begin
    case(p2)is
        when kst1                               => report
            "p2:std_logic_vector value is 0"; -- 0000
            when '0'&kst2                      => report
            "p2:std_logic_vector value is 1"; -- 0001
            when ("0001" sll 1)                  => report
            "p2:std_logic_vector value is 2"; -- 0010
            when ("0111" and "1011")              => report
            "p2:std_logic_vector value is 3"; -- 0011
            when kst2(1 downto 0)&"00"          => report
            "p2:std_logic_vector value is 4"; -- 0100
            when (2=>kst2(0),0=>'1',others=>'0') => report
            "p2:std_logic_vector value is 5"; -- 0101
            when others                          => report
            "p2:std_logic_vector value is unknown";
                end case;
        end process;
        -- signed/unsigned from numeric_std
process(p3)begin
    case(p3)is
        when kun1&'0'                           => report
            "p3:unsigned value is 0"; -- 0000
            when to_unsigned(1,4)                 => report
            "p3:unsigned value is 1"; -- 0001
            when kun2                           => report
            "p3:unsigned value is 2"; -- 0010
            when ("0111" and "1011")              => report
            "p3:unsigned value is 3"; -- 0011
            when SHIFT_LEFT(kun2,1)               => report
            "p3:unsigned value is 4"; -- 0100
            when kun3                           => report
            "p3:unsigned value is 5"; -- 0101
            when others                          => report
            "p3:unsigned value is unknown";
                end case;
        end process;
end arch_e;

library ieee;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;
entity tb is
end entity;
architecture arch_tb of tb is
    signal p1: bit_vector(3 downto 0);
    signal p2: std_logic_vector(3 downto 0);
    signal p3: unsigned(3 downto 0);
begin
    dut_inst:entity work.e port map (p1,p2,p3);
    process begin
        wait for 5 ns;
        p1 <= "0000";wait for 5 ns;

```

```

    p1 <= "0001";wait for 5 ns;
    p1 <= "0010";wait for 5 ns;
    p1 <= "0011";wait for 5 ns;
    p1 <= "0100";wait for 5 ns;
    p1 <= "0101";wait for 5 ns;
    p1 <= "0110";wait for 5 ns;

    p2 <= "0000";wait for 5 ns;
    p2 <= "0001";wait for 5 ns;
    p2 <= "0010";wait for 5 ns;
    p2 <= "0011";wait for 5 ns;
    p2 <= "0100";wait for 5 ns;
    p2 <= "0101";wait for 5 ns;
    p2 <= "0110";wait for 5 ns;

    p3 <= "0000";wait for 5 ns;
    p3 <= "0001";wait for 5 ns;
    p3 <= "0010";wait for 5 ns;
    p3 <= "0011";wait for 5 ns;
    p3 <= "0100";wait for 5 ns;
    p3 <= "0101";wait for 5 ns;
    p3 <= "0110";wait for 5 ns;
    wait;
  end process;
end architecture;

```

Output

```

Report NOTE at 0 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P0:
  "p1:bit_vector value is 0"
Report NOTE at 0 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
  "p2:std_logic_vector value is unknown"
Report NOTE at 0 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P2:
  "p3:unsigned value is unknown"
10 NS
Report NOTE at 10 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P0:
  "p1:bit_vector value is 1"
15 NS
Report NOTE at 15 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P0:
  "p1:bit_vector value is 2"
20 NS
Report NOTE at 20 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P0:
  "p1:bit_vector value is 3"
25 NS
Report NOTE at 25 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P0:
  "p1:bit_vector value is 4"

```

```

30 NS
Report NOTE at 30 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P0:
  "p1:bit_vector value is 5"
35 NS
Report NOTE at 35 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P0:
  "p1:bit_vector value is unknown"
40 NS
Report NOTE at 40 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
  "p2:std_logic_vector value is 0"
45 NS
Report NOTE at 45 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
  "p2:std_logic_vector value is 1"
50 NS
Report NOTE at 50 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
  "p2:std_logic_vector value is 2"
55 NS
Report NOTE at 55 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
  "p2:std_logic_vector value is 3"
60 NS
Report NOTE at 60 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
  "p2:std_logic_vector value is 4"
65 NS
Report NOTE at 65 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
  "p2:std_logic_vector value is 5"
70 NS
Report NOTE at 70 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
  "p2:std_logic_vector value is unknown"
75 NS
Report NOTE at 75 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P2:
  "p3:unsigned value is 0"
80 NS
Report NOTE at 80 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P2:
  "p3:unsigned value is 1"
85 NS
Report NOTE at 85 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P2:
  "p3:unsigned value is 2"
90 NS
Report NOTE at 90 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P2:
  "p3:unsigned value is 3"
95 NS

```

```

Report NOTE at 95 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P2:
  "p3:unsigned value is 4"
100 NS
Report NOTE at 100 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P2:
  "p3:unsigned value is 5"
105 NS
Report NOTE at 105 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P2:
  "p3:unsigned value is unknown"
(simv): Simulation complete, time is 110 NS.

```

Use of Select Choice and Constant Initialization

Example 228 Usage of select choice and constant initialization

```

library ieee;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;
entity e is
port(
  p3:in unsigned(3 downto 0));
end entity;
architecture arch_e of e is
  constant kun1:unsigned(2 downto 0):=(others => '0');
  constant kun2:unsigned(3 downto
  0):=resize(unsigned'"10",4);
  constant kun3:unsigned(3 downto
  0):=SHIFT_RIGHT("1010",1);

  signal s_out :integer:=6;
begin
  with p3 select  s_out <=
    0 when  kun1&'0',
    1 when  to_unsigned(1,4),
    2 when  kun2,
    3 when ("0111" and "1011"),
    4 when  SHIFT_LEFT(kun2,1),
    5 when  kun3,
    6 when others;

  process(s_out)begin  report "s_out: "&to_string(s_out);end process;
end arch_e;

library ieee;
use ieee.std_logic_1164.all;
use ieee.numeric_std.all;
entity tb is
end entity;
architecture arch_tb of tb is
  signal p3: unsigned(3 downto 0);
begin
  dut_inst:entity work.e port map (p3);

```

```

process begin
    p3 <= "0000";wait for 5 ns;
    p3 <= "0001";wait for 5 ns;
    p3 <= "0010";wait for 5 ns;
    p3 <= "0011";wait for 5 ns;
    p3 <= "0100";wait for 5 ns;
    p3 <= "0101";wait for 5 ns;
    p3 <= "0110";wait for 5 ns;
    wait;
end process;
end architecture;
```

Output

```

Report NOTE at 0 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
    "s_out: 6"
Report NOTE at 0 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
    "s_out: 0"
5 NS
Report NOTE at 5 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
    "s_out: 1"
10 NS
Report NOTE at 10 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
    "s_out: 2"
15 NS
Report NOTE at 15 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
    "s_out: 3"
20 NS
Report NOTE at 20 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
    "s_out: 4"
25 NS
Report NOTE at 25 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
    "s_out: 5"
30 NS
Report NOTE at 30 NS in design unit E(ARCH_E) from
process /TB/DUT_INST/_P1:
    "s_out: 6"
(simv): Simulation complete, time is 35 NS.
```

The following are the commands to execute:

```
% vhdlan -vhdl08 test.vhd
% vcs tb
% simv
```

Limitations

The following are the limitations for the static composite expression feature:

1. Complete packages of `numeric_bit` and `numeric_bit_unsigned` are not supported.
2. The following function calls from `numeric_std` and `numeric_std_unsigned` packages are not supported:
 - minimum and maximum
 - `find_leftmost` and `find_rightmost`
 - `to_string` and similar string conversions
 - `resize` function with vector size is not supported but only scalar size is supported.
3. The string conversion function `to_string` and similar string conversions present in the `std_logic_1164` package are not supported.
4. Only '`ascending`', '`left`', '`right`', '`low`', '`high`', '`length`', and '`range`' attributes are supported.
5. Aggregated and index selects from multidimensional arrays are not supported.
6. The physical data types are not supported.
7. Using aliased objects to slices of vectors is not supported.

Support for VHDL 2008 Enhanced Generics in Components

VCS supports the usage of VHDL 2008 enhanced generics (types/subprograms/packages) in components and corresponding instantiations of components with generic maps as defined in the VHDL 2008 LRM.

With this support of enhanced generics in VHDL 2008, you can declare generic types, sub-programs, and packages in a component. This allows you to declare abstract and reusable entities with different types and design specialization. When you instantiate a component, you supply values for the generic constants for that instance.

Following is the use model to enable this feature:

```
% vhdlan -vhdl08 <other_options> file.vhd
```

Examples

The following example shows the usage of generic type and function in component interface:

Example 229 test.vhd

```

entity GenericCnt is
  generic (
    type T;
    Constant C : T;
    function GenFunc (signal arg : T) return T
  );
  port (
    clk : in bit;
    rst : in bit;
    Cnt : inout T
  );
end;

architecture arch of GenericCnt is
begin
  process(clk,rst)
  begin
    if(rst='1') then
      cnt <= C;
    elsif(clk'event and clk='1') then
      cnt <= GenFunc(cnt);
    end if;
  end process;
end architecture arch;

library ieee;
use ieee.std_logic_1164.all;
use ieee.std_logic_unsigned.all;
use ieee.std_logic_arith.all;

entity top is
end entity;

architecture arch of top is
component GenericCnt is
  generic (
    type T;
    Constant C : T;
    function GenFunc(signal arg : T) return T
  );
  port (
    clk : in bit;
    rst : in bit;
    Cnt : inout T
  );
end component;

```

```

end component;

signal clk , rst : bit;
signal cnt : std_logic_vector(2 downto 0):= (others => '0');
function IncrCnt ( signal arg : in std_logic_vector ) return
std_logic_vector is
begin
  return arg + 1;
end function;
begin
  inst : GenericCnt generic map(
    T => std_logic_vector(2 downto 0),
    C => (others => '0'),
    GenFunc => IncrCnt
  )
  port map ( clk,rst,cnt);
process
begin
  if(now = 80 ns) then
    wait;
  else
    clk <= '0';
    wait for 5 ns;
    clk <= '1';
    wait for 5 ns;
  end if;
end process;

process
begin
  rst <= '1';
  wait for 3 ns;
  rst <= '0';
  wait for 10 ns;
  wait;
end process;

process(cnt)
use std.textio.all;
use ieee.std_logic_textio.all;
variable l : line;
begin
  write(l,"time : " & time'image(now));
  write(l," CNT : " & integer'image(conv_integer(unsigned(cnt))));
  writeline(output,l);
end process;
end architecture;

```

Compile and run test.vhd as follows:

```
% vhdlan -vhdl08 -full64 test.vhd
% vcs top -nc -full64 -R
% simv
```

Following is the simulation output:

```
time : 0 ns CNT : 0
time : 0 ns CNT : 0
time : 5 ns CNT : 1
time : 15 ns CNT : 2
time : 25 ns CNT : 3
time : 35 ns CNT : 4
time : 45 ns CNT : 5
time : 55 ns CNT : 6
time : 65 ns CNT : 7
time : 75 ns CNT : 0
```

Note:

Components with enhanced generics can be fully configured with generics and port map using configuration specifications. For example, as follows:

```
entity l is
  generic(type t; n : integer := 0);
  port(i : t; o : out t);
end entity;

architecture a of l is
  signal s : t;
begin
  s <= i;
  o <= s;
end architecture;

library ieee;
use ieee.std_logic_1164.all;

entity e is
  generic(n : integer := 0);
  port(i : std_logic_vector(1 to n+2); o : out
  std_logic_vector(1 to n+2));
end entity;

architecture a of e is
  component l is
    generic(type t; m : integer := 0);
    port(i : t; o : out t);
  end component;
  for u1:l use entity l generic map (n=>m) port map( "1");
  for u2:l use entity l generic map (n=>m) port map("01");
  signal s : bit_vector(1 to n+1);
begin
  u1 : l generic map( bit_vector(1 to n+1), m=>n+1) port
  map(o=>s);
  u2 : l generic map(std_logic_vector(1 to n+2), m=>n+2) port
  map(o=>o);
end architecture;
```

```
configuration c of e is
for a
end for;
end configuration;
```

Limitations

Following are the limitations of the VHDL 2008 Enhanced Generics in Components feature:

- Component with enhanced generics can be configured in the configuration declaration with just entity aspect, which is the common case, like:

```
for u1:l use entity l; end for;
```

The limitation is that the incremental generic or port map is not supported.

Note:

This limitation is only for configuration declaration. Components with enhanced generics can be fully configured in configuration specifications, including incremental generic or port map.

Example 230 test1.vhd

```
entity l is
generic(type t; n : integer := 0);
port(i : t; o : out t);
end entity;

architecture a of l is
signal s : t;
begin
s <= i;
o <= s;
end architecture;

library ieee;
use ieee.std_logic_1164.all;

entity e is
generic(n : integer := 0);
port(i : std_logic_vector(1 to n+2); o : out
std_logic_vector(1 to n+2));
end entity;

architecture a of e is
component l is
generic(type t; m : integer := 0);
port(i : t; o : out t);
end component;
signal s : bit_vector(1 to n+1);
```

```

begin
  u1 : l generic map( bit_vector(1 to n+1), m=>n+1) port
  map(o=>s);
  u2 : l generic map(std_logic_vector(1 to n+2), m=>n+2) port
  map(i=>o);
end architecture;

configuration c of e is
for a
  for u1:l use entity l generic map (n=>m) port map(i=> "1");
  end for;
  for u2:l use entity l generic map (n=>m) port
  map(i=>"01"); end for;
end for;
end configuration;
```

Error Message:

```

Error-[VHDLENHCMPCFGMAP] Unsupported interface map in component
configuration.
test1.vhd, 21
C
for u1:l use entity l generic map (n=>m) port map(i=> "1"); end for;
^
Unsupported generic or port map in enhanced component (L) configuration.
Please use generic or port map at the instantiation statement and/or at
the configuration specification.

Error-[VHDLENHCMPCFGMAP] Unsupported interface map in component
configuration.
test1.vhd, 22
C
for u2:l use entity l generic map (n=>m) port map(i=>"01"); end for;
^
Unsupported generic or port map in enhanced component (L) configuration.
Please use generic or port map at the instantiation statement and/or at
the configuration specification.
```

- Components with enhanced generics are not supported for the Verilog modules instantiated in VHDL top. For example:

Example 231 test2.vhd

```

library ieee;
use ieee.std_logic_1164.all;
entity ent is
  generic(
    type T
  );
  port (
    in1 : in T;
    out1 : out T
  );
```

```

end entity ent;

architecture arch of ent is
    signal out2 : T;
    component VL is
        port(
            in1 : in std_logic_vector(3 downto 0);
            out1 : out std_logic_vector( 3 downto 0)
        );
    end component;
begin
    out1 <= in1;
    VL_inst : VL port map(in1,out2);
end;

library ieee;
use ieee.std_logic_1164.all;
entity top is
end;

architecture arch of top is
    component ent is
        generic(
            type T
        );
        port (
            in1 : in T;
            out1 : out T
        );
    end component ent;
    signal in1,out1 :std_logic_vector(3 downto 0);
begin
    inst : ent generic map(std_logic_vector(3 downto 0)) port map
    (in1,out1);
end;

```

Warning Message:

```

Warning-[ELW_UNBOUND] Unbound component
The component instantiation '/TOP/INST/VL_INST' (file:
test.vhd, line: 28) will have no effect because component 'VL' is
unbound. No entity definition for component
'VL' can be found in the following libraries ( WORK ) referenced by the
architecture 'ARCH' of entity 'TOP'.
Please bind the component explicitly to an entity (architecture) pair,
and verify that the pair was analyzed successfully.

```

Support for VHDL 2008 Local Packages

The front-end and back-end compilers of VCS support the VHDL 2008 Local Packages feature. You must use the `-vhdl08` option for the VHDL 2008 feature.

Use either of the following methods to enable the VHDL 2008 Local Packages feature:

- Use the `-xlocal_pack vhdlan` option.
- Specify `LOCAL_PACK=TRUE` in the `synopsys_sim.setup` file.

Example 232 Global uninstantiated package, multiple local package instantiation

```
package gp is
    generic(type t; n : t);
    function f return t;
end package;
package body gp is
    function f return t is
    begin
        return n;
    end function;
end package body;

entity e is
    generic(g : integer := 8);
end entity;

architecture a of e is
    package pi is new work.gp generic map(integer, g);
    package pv is new work.gp generic map(bit_vector,
    bit_vector'("101"));
    use pi.all;
    use pv.all;
    constant ci : integer := f;
    constant cv : bit_vector := f;
begin
end architecture;
```

Example 233 Local uninstantiated package, multiple local package instantiation

```
entity e is
    generic(g : integer := 8);
end entity;

architecture a of e is
    package lp is
        generic(type t; n : t);
        function f return t;
    end package;
    package body lp is
        function f return t is
        begin
            return n;
        end function;
    end package body;

    package pi is new lp generic map(integer, g);
```

```

package pv is new lp generic map(bit_vector, bit_vector'("101"));
use pi.all;
use pv.all;
constant ci : integer := f;
constant cv : bit_vector := f;
begin
end architecture;

```

Example 234 Local global uninstantiated package, multiple local package instantiation

```

package gp is
    package lp is
        generic(type t; n : t);
        function f return t;
    end package;
end package;
package body gp is
    package body lp is
        function f return t is
        begin
            return n;
        end function;
    end package body;
end package body;

entity e is
    generic(g : integer := 8);
end entity;

architecture a of e is
    use work_gp.lp;
    package pi is new lp generic map(integer, g);
    package pv is new lp generic map(bit_vector, bit_vector'("101"));
    use pi.all;
    use pv.all;
    constant ci : integer := f;
    constant cv : bit_vector := f;
begin
end architecture;

```

Limitation

The following is the limitation of this feature:

- The VHDL 2008 Local Packages feature is not supported for Coverage, Debug, Xprop, NLP, and Verdi.

Support for Unconstrained Elements in Generics and Generic Types

VCS supports unconstrained elements in the types associated with the formal generic types in VHDL 2008 using the `-vhdl08` option.

VCS supports the unconstrained elements for the formal generic types where the arbitrarily complex derived types can be created and used via implicit subtypes, allocators, and aggregates. You must maintain agreement between the element subtypes and the composite subtypes in which they appear, directly or indirectly.

Example 235

```

package gp is
    generic (type t);

    type lh;
    type alh is access lh;
    type lt is record
        l : string;
        v : t;
    end record;
    type vlt is array (natural range <>) of lt;

    type lh is record
        n : alh;
        v : lt;
    end record;
    procedure push(l : inout alh; v : t; lbl : string);
    function dup(v : t; n : positive) return vlt;
end gp;
package body gp is
    procedure push(l : inout alh; v : t; lbl : string) is
        variable e : alh;
    begin
        e := new lh'(l, (lbl, v));
        l := e;
    end push;

    function dup(v: t; n : positive) return vlt is
        constant k : lt := (to_string(n), v);
    begin
        if n = 1 then
            return (0 => k);
        else
            return dup(v, n - 1) & k;
        end if;
    end dup;
end gp;

package pt is

```

```

type bv is array (natural range <>) of bit_vector;
type r1 is record
    e1 : string;
    e2 : bv;
end record;
end pt;

use work.pt.all;

package sp is new gp generic map (r1);

```

Limitations

The Unconstrained Elements in Generics and Generic Types feature has the following limitations:

- The port associations at the VHDL/Verilog boundary is not supported.
- The formal sub element association and the type conversions are not supported.
- The Unconstrained Elements in Generics and Generic Types feature is not supported for Coverage, Debug, Xprop, NLP, and Verdi.
- The array aggregates where the element type is a generic type associated with the unconstrained elements are not supported.

VHDL 2019 Conditional Analysis Tool Directives

Conditional analysis tool directives are supported by VCS. For more information on conditional analysis, refer to section 24.2 in the VHDL 2019 LRM.

This feature enables you to control which part of the description to use during the Analysis phase. The `conditional_analysis_directive` are as follows:

```

`if conditional_analysis_expression then
`elsif conditional_analysis_expression then
`else
`end [ if ]
`warning string_literal
`error string_literal

```

For more information about using the directives, see the *IEEE Std. VHDL 2019 LRM*.

Use Model

Conditional analysis tool directives are supported with the following options:

`-vhdl87, -vhdl93, -vhdl02 or -vhdl08 (vhdlan)` options.

OR

VHDL_MODE = vhdl187, VHDL_MODE = vhdl193, VHDL_MODE = vhdl108 or VHDL_MODE = vhdl102 setup variable.

Usage Example

Consider the following example:

Example 236 t1.vhd

```
entity e is
    port (clk : bit; r : out integer := 0);
end e;
architecture a of e is
begin
    p : process (clk)
        variable v : integer := 0;
        variable n : integer := 10000000;
        variable d : integer := 0;
    begin
        if clk'event and clk = '1' then
            v := v + n;
            d := d + 1;
            n := n / d;
        `if vhdl_version >= "2008" then
            if n = 0 then
                std.env.finish;
            end if;
        `else
            assert n > 0 report "done" severity failure;
        `end if
            r <= v;
        end if;
    end process;
end a;
entity top is
end top;
architecture a of top is
component e
    port (clk : bit; r : out integer);
end component;
    signal clk : bit;
    signal r : integer;
begin
    clk <= not clk after 5 ns;
    i : e port map (clk, r);
    `if vhdl_version >= "2008" then
        p : process (r)
        begin
            report to_string(r);
        end process;
```

```

`else
  p : process (r)
    use std.textio.all;
    variable l : line;
begin
  write(l, now, field => 6);
  write(l, string'(" "));
  write(l, r, field => 8);
  writeline(output, l);
end process;
`end if
end a;

```

Run the above example using the following commands:

- vhdlan -vhdl108 t1.vhd
- vcs top
- simv

The following output is generated:

```

Report NOTE at 0 NS in design unit TOP(A) from process /TOP/P:
"0"
5 NS
Report NOTE at 5 NS in design unit TOP(A) from process /TOP/P:
"10000000"
15 NS
Report NOTE at 15 NS in design unit TOP(A) from process /TOP/P:
"20000000"
25 NS
Report NOTE at 25 NS in design unit TOP(A) from process /TOP/P:
"25000000"
35 NS
Report NOTE at 35 NS in design unit TOP(A) from process /TOP/P:
"26666666"
45 NS
Report NOTE at 45 NS in design unit TOP(A) from process /TOP/P:
"27083332"
55 NS
Report NOTE at 55 NS in design unit TOP(A) from process /TOP/P:
"27166665"
65 NS
Report NOTE at 65 NS in design unit TOP(A) from process /TOP/P:
"27180553"
75 NS
Report NOTE at 75 NS in design unit TOP(A) from process /TOP/P:
"27182537"
85 NS
Report NOTE at 85 NS in design unit TOP(A) from process /TOP/P:
"27182785"
95 NS

```

```
Report NOTE at 95 NS in design unit TOP(A) from process /TOP/P:  
"27182812"  
105 NS
```

In this example, the analysis phase includes the following directives:

```
`if vhdl_version >= "2008" then  
  if n = 0 then  
    std.env.finish;  
  end if;
```

Limitations

Inclusion or exclusion of parts of a VHDL description is determined during analysis phase.
You can change it by re-analyzing the description.

28

SAIF Support

The Synopsys Power Compiler enables you to perform power analysis and power optimization for your designs by entering the `power` command at the `vcs` prompt. This command outputs Switching Activity Interchange Format (SAIF) files for your design.

SAIF files support signals and ports for monitoring as well as constructs such as generates, enumerated types, and integers.

This chapter covers the following topics:

- [Using SAIF Files with VCS](#)
- [SAIF System Tasks for Verilog or Verilog-Top Designs](#)
- [The Flows to Generate a Backward SAIF File](#)
- [SAIF Calls That Can Be Used on VHDL or VHDL-Top Designs](#)
- [SAIF Support for Two-Dimensional Memories in v2k Designs](#)
- [UCLI SAIF Dumping](#)
- [Criteria for Choosing Signals for SAIF Dumping](#)
- [Improving Simulation Time by Reducing the Overhead due to SAIF File Dumping](#)

Using SAIF Files with VCS

VCS has native SAIF support so you no longer need to specify any compile-time options to use SAIF files. If you want to switch to the old flow of dumping SAIF files with the PLI, you can continue to give the option `-P $VPOWER_TAB $VPOWER_LIB` to VCS, and the flow will not use the native support.

Note the following when using VCS native support for SAIF files:

- VCS does not need any additional switches.
- VCS does not need a Power Compiler specific tab file (and the corresponding library)
- VCS does not need any additional settings.
- Functionality is built into VCS.

SAIF System Tasks for Verilog or Verilog-Top Designs

This section describes SAIF system tasks that you can use at the command line prompt.

Note that *mixedHdScope* in the following discussion can be one of the following:

- Verilog scope
- VHDL scope
- Mixed HDL scope

Note also that a *design_object* in the following discussion can be one of the following:

- Verilog scope or variable
- VHDL scope or variable
- Any mixed HDL scope or variable

`$set_toggle_region`

Specifies a module instance (or scope) for which VCS records switching activity in the generated SAIF file. Syntax:

```
$set_toggle_region(instance[, instance]);
```

`$toggle_start`

Instructs VCS to start monitoring switching activity.

Syntax:

```
$toggle_start();
```

`$toggle_stop`

Instructs VCS to stop monitoring switching activity.

Syntax

```
$toggle_stop();
```

`$toggle_reset`

Sets the toggle counter to 0 for all the nets in the current toggle region.

Syntax:

```
$toggle_reset();
```

`$toggle_report`

Reports switching activity to an output file.

Syntax:

```
$toggle_report("outputFile", synthesisTimeUnit,  
               mixedHdlScope);
```

This task has a slight change in native SAIF implementation compared to PLI-based implementation. VCS considers only the arguments specified here for processing. Other arguments have no meaning.

VCS does not report signals in modules defined under the `'celldefine` compiler directive.

```
$read_lib_saif
```

Allows you to read in a state dependent and path dependent (SDPD) library forward SAIF file. It registers the state and path dependent information on the scope. It also monitors the internal nets of the design.

Syntax:

```
$read_lib_saif("inputFile");
```

```
$set_gate_level_monitoring
```

Allows you to turn on or off the monitoring of nets in the design.

Syntax:

```
$set_gate_level_monitoring("on" | "rtl_on");
```

The `"on"` and `"rtl_on"` keyword arguments are called policies.

`"rtl_on"` Monitors all reg, tri, and trireg data objects for toggles. Monitors other types of nets for toggles if they are cell highconn (ports that connect toward the top of the design hierarchy in a module declared to be a cell).

`"on"` Monitors all net type of objects for toggles. Monitors reg data objects if they are cell highconn. This is the default monitoring policy.

Verilog memories, Multi-dimensional arrays, and SystemVerilog data objects are supported with an extended syntax:

```
$set_gate_level_monitoring("on" | "rtl_on",  
                           "mda" | "sv");
```

You include the `mda` argument for Verilog memories and multi-dimensional arrays, the `sv` argument for SystemVerilog data objects.

For more details on these task calls, refer to the *Power Compiler User Guide*.

Note:

The \$read_mpm_saif, \$toggle_set, and \$toggle_count tasks in the PLI-based vpower.tab file are obsolete and no longer supported.

on: Monitors both ports and signals.

off: Does not print ports or signals.

rtl_on: Monitors both ports and signals (same as on)

<region/signal> Arguments for specifying the following:

region: Mixed HDL/VHDL region and its children to consider for monitoring.

signal: (hierarchical path to) signal name.

Note:

VHDL variables are not dumped in SAIF SDPD (VHDL gate level).

Examples

```
# power -enable
# power -report
```

The Flows to Generate a Backward SAIF File

You can generate the following kinds of backward (or output) SAIF files:

- an SDPD backward SAIF file — using a library forward (or input) SAIF file
- a non-SDPD backward SAIF file — without using a library forward (or input) SAIF file.

Generating an SDPD Backward SAIF File

To generate an SDPD backward SAIF file, include the SAIF system tasks in the module definition containing the \$read_lib_saif("inputFile") system task.

For example:

```
initial begin
    $read_lib_saif("inputFile");
    $set_toggle_region(mixedHdlScope);
    // initialization of Verilog signals
        M
    $toggle_start;
    // testbench
        M
    $toggle_stop;
```

```
$toggle_report("outputFile", timeUnit,mixedHdlScope);
end
```

The `$set_toggle_region(mixedHdlScope)` system task's scope argument must be one level higher in the design hierarchy than the scope of the module in the library forward SAIF file, for which you intend VCS to generate the backward SAIF file.

For example, if VCS monitors instance `top.u_dut.u_saif_module`, the argument to the `$set_toggle_region` system task is `top.u_dut`, as follows:

```
$set_toggle_region(top.u_dut);
```

Enclose the modules listed in the library forward SAIF file, those from which you intend VCS to monitor and generate the backward SAIF file, between `'celldefine` and `'endcelldefine` compiler directives.

Generating a Non-SPDP Backward SAIF File

If you are not including a library forward (or input) SAIF file, include the `$set_gate_level_monitoring("on")` system task with the other SAIF system tasks.

For example:

```
initial begin
    $set_gate_level_monitoring("on");
    $set_toggle_region(mixedHdlScope);
    // initialization of Verilog signals, and then:
    $toggle_start;
    // testbench
        M
    $toggle_stop;
    $toggle_report("outputFile", timeUnit,mixedHdlScope);
end
```

SAIF Calls That Can Be Used on VHDL or VHDL-Top Designs

VHDL use model mainly consists of the `power` command and its options at the `simv` command-line.

The `power` command syntax is as follows:

```
power -enable -disable -reset -report <filename> <synthesisTimeUnit>
<mixedHdlScope> <filename> [<testbench_path_name>] -gate_level on|off|
rtl_on <region/signal/variable>
```

Here:

`-enable`: Enables monitoring of switching (`toggle_start`).

- disable: Disables monitoring of switching (toggle_stop).
- reset: Resets monitoring of switching (toggle_reset).
- report: Reports switching activity to an output file (toggle_report).
- gate_level: Turns on or off the monitoring based on the following:
 - on: Monitors both ports and signals.
 - off: Does not print ports or signals.
 - rtl_on: Monitors both ports and signals (same as on)
- <region/signal>: Arguments for specifying the following:
 - region: Mixed HDL/VHDL region and its children to consider for monitoring.
 - signal: (hierarchical path to) signal name.

Note:

VHDL variables are not dumped in SAIF SDPD (VHDL gate level).

Examples:

```
# power -enable
# power -report
```

SAIF Support for Two-Dimensional Memories in v2k Designs

SAIF supports monitoring of two-dimensional memories in v2k designs.

You must pass the `mda` keyword to the `$set_gate_level_monitoring` system task to monitor two-dimensional memories in v2k designs.

If you want to dump through the UCLI command, you must pass the `mda` string to the `power -gate_level` command, as shown in the below section.

UCLI SAIF Dumping

The following is the use model for UCLI SAIF dumping:

```
% simv -ucli
ucli% power -gate_level on mda
ucli% power <scope>
ucli% power -enable
ucli% run 100
ucli% power -disable
ucli% power -report <saif_filename> <timeUnit> <modulename>
ucli% quit
```

Criteria for Choosing Signals for SAIF Dumping

Verilog:

VCS supports only scalar wire and reg, as well as vector wire and reg, for monitoring. It does not consider wire/reg declared within functions, tasks and named blocks for dumping. Also, it does not support bit selects and part selects as arguments to `$set_toggle_region` or `$toggle_report`. In addition, it monitors cell highconns based on the policy.

VHDL:

Signals or ports are supported for monitoring. Variables are not supported, as it is difficult to infer latches/flops at RTL level.

Constructs like generates, enumerated types, records, array of arrays integers, and so on, are also supported over and above the basic VHDL types.

The following rules are followed regarding the monitoring policy for VHDL:

Port	Signals	Variables
on	Y	Y
off	N	N
rtl_on	Y	N

Mixed HDL:

The rules for mixed HDL are basically the same as that of VHDL if VHDL is on top, and Verilog if Verilog is on top.

Improving Simulation Time by Reducing the Overhead due to SAIF File Dumping

SAIF file dumping is enhanced to improve the simulation time by reducing the overhead due to SAIF.

Use Model

At runtime, you can use the `-saif_opt` option with appropriate arguments to reduce the overhead due to SAIF file dumping as shown in the following command line:

```
% simv [simv_options] -saif_opt+option1+option2+...
```

You can specify one or more options along with the `-saif_opt` option. The options available are as follows:

```
toggle_start_at_set_region
```

Use this option to implicitly call \$toggle_start with \$set_toggle_region.

```
toggle_stop_at_toggle_report
```

Use this option to implicitly call \$toggle_stop with \$toggle_report.

```
skip_celldefine_scopes
```

Use this option to skip monitoring activity for the modules that are defined under the `celldefine compiler directive or are resolved using -v and -y options.

Example

Consider the following example:

```
`timescale 1ns/1ns
module top;
    wire w;
    bot b(w);
    initial begin
        #5 $set_gate_level_monitoring("rtl_on");
        $set_toggle_region(b);
        #95 $toggle_stop;
        $toggle_report("1.saif", 1e-9, b);
        $finish;
    end
endmodule

module bot(output reg p);
    initial begin
        #20 p = 1'b1;
        #20 p = 1'b0;
        #20 p = 1'b1;
        #20 p = 1'b0;
    end
endmodule
```

To run the example, use the following commands:

```
% vcs -sverilog 1.v
% simv -saif_opt+toggle_start_at_set_region
```

It generates the following output:

```
/** The set_gate_level_monitoring command explicitly turns ON the
internal nets monitoring */
(SAIFFILE
(SAIFVERSION "2.0")
(DIRECTION "backward")
(DESIGN)
(VENDOR "Synopsys, Inc")
(VERSION "1.0")
```

```
(DIVIDER / )
(TIMESCALE 1 ns)
(DURATION 95.00)
(INSTANCE top
    (INSTANCE b
        (NET
            (p
                (T0 40) (T1 40) (TX 15)
                (TC 3) (IG 0)
            )
        )
    )
)
```

Limitations

The feature has the following limitations:

- The `skip_celldefine_scopes` option is supported only if library forward SAIF file is not read.
- The enhancements are not supported for UCLI `power` command.

29

Encrypting Source Files

There are different ways to encrypt your HDL source files to deliver your IP. Of these, this chapter describes the following methods to encrypt your Verilog and VHDL source files and exchange IPs.

- [IEEE Verilog Standard 1364-2005 Encryption](#)
 - [IEEE VHDL Standard 1076-2008 Encryption](#)
 - [128-bit Advanced Encryption Standard](#)
 - [Skipping Encrypted Source Code](#)
 - [gen_vcs_ip](#)
-

IEEE Verilog Standard 1364-2005 Encryption

VCS supports encryption of Verilog and SystemVerilog IP code in protected envelopes as defined by the IEEE Standard 1364-2005.

In addition, VCS supports the recommendations from the IEEE P1735 working group for encryption interoperability between different encryption and decryption tools, denoted as “version 1” by P1735.

Note:

SystemC encryption is not supported by this feature.

The following option tells VCS to encrypt the specified Verilog or SystemVerilog source files according to the “IEEE Std 1364-2005” standard for encryption envelopes.

`-ipprotect protection_header_file`

In this encryption mode, VCS does not compile the Verilog or SystemVerilog source files, but instead encrypts each source file into a separate encrypted Verilog or SystemVerilog file. Each encrypted file is saved under the same filename, but changes its filename extension to .vp. Using the `-ipprotect` option allows IP providers to specify a `protection_header_file` that contains various protection pragmas.

VCS encrypts:

- the source files on the vcs command line
- the source files specified in 'include compiler directives.

Note:

- By default, VCS encrypts complete input files. Use the `-ipopt=partialprotect` option and argument to enable partial protection with it, VCS encrypts only the regions specified by 'protect pragma begin-end expressions.
- All 'include directives in the encrypted source files are modified by changing the extension of the included filenames from .v to .vp. The modified 'include directives are left as unencrypted text. In addition, every file included by a 'include directive is also encrypted and saved under the modified filename (changing the extension to .vp). Use the `-ipopt=noincludeprotect` option and argument with the `-ipprotect` option to disable processing of 'include compiler directives and the source files included by it.

This section on the IEEE Std 1364-2005 encryption mode includes the following:

- [The Protection Header File](#)
- [Other Options for IEEE Std 1364-2005 Encryption Mode](#)
- [How Protection Envelopes Work](#)
- [VCS Public Encryption Key](#)
- [Creating Interoperable Digital Envelopes Using VCS - Example](#)
- [Discontinued -ipkey Option](#)

The Protection Header File

The `protection_header_file` may look like the following:

Example 237 Sample IEEE Encryption Header File

```
`pragma protect data_method = "x-abc"
`pragma protect encoding = (enctype = "base64")
`pragma protect key_keyowner="IPcorp"
`pragma protect key_method="rsa"
`pragma protect key_keyname="IPcorp-123"
`pragma protect key_public_key
<content_representing_the_public_encryption_key>
```

For the VCS `base64` encoded RSA public encryption key, contact Synopsys support (vcs_support@synopsys.com).

The following `pragma protect expressions are required inside the protection_header_file:

key_keyowner

Identifies the owner of the key encryption key.

key_method

Specifies the key encryption algorithm, the asymmetric method for encrypting or decrypting.

key_keyname

Specifies keyowner's key name.

key_public_key

Specifies the public key for key encryption

The optional `pragma protect expressions that can be included are as follows:

data_method

Identifies the data encryption algorithm. Supported methods are as follows:

aes256-cbc

aes192-cbc

aes128-cbc

des-cbc

3des-cbc

The default data_method, if none is specified, is aes256-cbc.

author

Identifies the author of an envelope.

author_info

Specifies additional author information.

encoding

Specifies the coding scheme for encrypted data, you can specify either of the following:

base64

uuencode

The default encoding scheme, if none is specified, is base64.

comment

Comment documentation string that is not encrypted.

Note:

These encryption pragmas are only supported inside the `protection_header_file`, which is specified by the `-ipprotect` option. If they are specified anywhere else (such as in the Verilog or SystemVerilog source files), VCS outputs a warning message and ignores the pragma.

The only `'pragma protect` expressions allowed in input Verilog and SystemVerilog files are `'pragma protect begin` and `'pragma protect end`, when enabled with the `-ipopt=partialprotect` option and argument to mark the regions to be protected.

Unsupported Protection Pragma Expressions

The `'pragma protect` expressions that are not currently supported include:

<code>data_keyowner</code>	<code>data_keyname</code>
<code>data_public_key</code>	<code>data_decrypt_key</code>
<code>decrypt_license</code>	<code>runtime_license</code>
<code>reset</code>	<code>viewpoint</code>

Also unsupported are any expressions beginning with `digest_`.

Other Options for IEEE Std 1364-2005 Encryption Mode

In addition to the `-ipprotect` option, there are other options that you can use in this mode. This section describes them.

`-ipopt=partialprotect`

VCS encrypts complete file by default. Use this option to encrypt only regions marked by the pragmas: `'pragma protect begin` and `'pragma protect end` in the Verilog or SystemVerilog source files.

`-ipopt=noincludeprotect`

VCS in encryption mode encrypt files which are included by the `'include` compiler directive. Use this option to disable the processing of the `'include` compiler directive and files included by it.

`-ipopt=ext=ext`

Use this option to specify the filename extension for encrypted files.

`-ipopt=outdir=dir`

Use this option to specify the target directory for encrypted files.

`+includer+directory+...`

Specifies the directories that VCS searches for source files specified with the `'include` compiler directive. By default VCS writes encrypted versions of these source files in the directory in which it finds the source files.

The encrypted copies have the same filename and extension of the original except that the `p` character is appended to the filename extension. So for example if it finds a SystemVerilog source file in a Verilog library with the name `dev1.sv`, the encrypted version in that library is `dev1.svp`.

You can specify multiple Verilog libraries with this option by using the plus (+) character as a delimiter, for example:

`+includer+INTRCTR+IOMTR+/DW/SIMENV`

`-f|-F|-file filename`

Specifies a file that contains a list of Verilog or SystemVerilog source files to be encrypted. The `-f`, `-F`, and `-file` options are interchangeable in this encryption mode.

`+define+MACRO=VALUE`

Defines the specified text macro to the specified value.

A text macro so defined at encryption time (when encrypting files instead of compiling files) cannot be overridden at a subsequent compile-time (when including the encrypted files in some later compilation and also entering the `+define` option). VCS ignores the attempted override without displaying any error, warning, or informational message.

`-ipout filename.ext`

This option tells VCS to write the encrypted file for the first Verilog or SystemVerilog source file on the command line with the specified filename and extension. You can enter a pathname for the protected file.

This option only works for the first Verilog or SystemVerilog source file on the `vcs` command line, and does not work for other source files on the command line or files included with the `'include` compiler directive or in Verilog libraries.

How Protection Envelopes Work

As specified in IEEE Std 1364-2005, annex H “Encryption/decryption flow,” section H.3 Digital envelopes:

“The sender encrypts the design using a symmetric key encryption algorithm and then encrypts the symmetric key using the recipient’s public key. The encrypted symmetric key

is recorded in a `key_block` in the protected envelope. The recipient is able to recover the symmetric key using the appropriate private key and then decrypts the design with the symmetric key."

Protection envelopes work as follows:

1. The encrypting tool generates a random key called "session key."
2. The encrypting tool then encrypts the design using this session key.
3. For each potential decrypting tool, information about that tool must be provided using `'pragma protect` expressions in the encryption envelope.

This information includes `key_keyowner`, `key_keyname`, the asymmetric `key_method`, and `key_public_key` for each tool.

4. The encrypting tool then encrypts the session key multiple times, once for each decrypting tool using information provided in the encryption envelope for that tool.
5. The encrypted session key is then recorded in `key_blocks` in the protected envelope.

Multiple `key_blocks` are generated, one for each decrypting tool.

6. The decrypting tool examines the `key_blocks` in the decryption envelope to find one encrypted using a key to which the tool has access.
7. The decrypting tool is able to recover the session key from its `key_block` using the appropriate private key.
8. The decrypting tool then decrypts the design with the session key.

VCS Public Encryption Key

For the VCS `base64` encoded RSA public encryption key, contact Synopsys support (vcs_support@synopsys.com). Synopsys also provides use cases and examples along with the key.

The following `'pragma protect` expression identifies this key:

```
'pragma protect key_keyowner="IPcorp"
'pragma protect key_method="rsa"
'pragma protect key_keyname="IPcorp-123"
```

VCS can decrypt and compile source files, which are encrypted by VCS or third-party tools.

To allow VCS to decrypt encrypted source files, the following snippet must be included while encrypting.

```
'pragma protect key_keyowner="IPcorp"
'pragma protect key_method="rsa"
```

```
'pragma protect key_keyname="IPcorp-123"
'pragma protect key_public_key
<content_representing_the_public_encryption_key>
```

The following example illustrates the protection envelope methodology for using this key in Verilog or SystemVerilog source code.

Creating Interoperable Digital Envelopes Using VCS - Example

VCS allows more than one key_block in a single protected envelope so it can be decrypted by tools from different vendors.

In the following example an IP provider created encrypted source files that can be decrypted by two different EDA tools, VCS and tools from VendorX.

An IP provider retrieves public keys for an EDA tool according to the EDA tool vendor's specific policies. For VCS, contact Synopsys support (vcs_support@synopsys.com).

The `protection_header_file` that this example specifies with the `-ipprotect` option is in [Figure 175](#).

Figure 175 Example Protection Header File for Source Encryption with VCS

```
'pragma protect author = "IPProvider"
  'pragma protect data_method = "x-abc"
  'pragma protect encoding = (enctype = "base64")

  'pragma protect key_keyowner="IPcorp"
  'pragma protect key_method="rsa"
  'pragma protect key_keyname="IPcorp-123"
  'pragma protect key_public_key
<content_representing_the_public_encryption_key>

  'pragma protect key_keyowner="VendorX"
  'pragma protect key_method="rsa"
  'pragma protect key_kevname="VFNDORX-123"
```



- (1) key block for VCS
- (2) key block for VendorX

Example 238 Verilog Source File to be Encrypted

```
// example.v
module secret (a, b);
    input a;
    output b;
    reg b;

    initial
        begin
            b = 0;
        end

    always
        begin
            #5 b = a;
        end
endmodule
```

The following `vcs` command line generate the encrypted file `example.vp` which can be decrypted by VCS and tools from VendorX.

```
vcs -ipprotect pragma_header_file example.v
```

Figure 176 *example.vp generated by VCS*

```
'pragma protect begin_protected
`pragma protect version=1
`pragma protect encrypt_agent="VCS"
`pragma protect encrypt_agent_info="G-2012.09-A[D] (ENG) Build
Date Feb 18 2012 00:14:12"
`pragma protect author="IPProvider"
`pragma protect key_keyowner="IPcorp"
`pragma protect key_keyname="IPcorp-123"
`pragma protect key_method="rsa"
`pragma protect encoding = (enctype = "base64", line_length =
76, bytes = 128 )
`pragma protect key_block
<content_representing_the_public_encryption_key>
```



```
'pragma protect key_keyowner="VendorX"
`pragma protect key_keyname="VENDORX-123"
`pragma protect key_method="rsa"
`pragma protect encoding = (enctype = "base64", line_length = 76,
bytes = 128 )
`pragma protect key_block
<content_representing_the_public_encryption_key>
```



```

`pragma protect data_method="x-abc"

`pragma protect encoding = (enctype = "base64", line_length = 76,
bytes = 176 )
`pragma protect data_block

<content_representing_the_public_encryption_key>

`pragma protect end_protected

```



- (1) Key block for VCS which contains the encrypted session key.
(encrypted using VCS public RSA key)
- (2) Key block for VendorX which contains the encrypted session key.
(encrypted using VendorX public RSA key)
- (3) Data block which contains the encrypted IP (encrypted using the session key)

To determine the session key that was used to encrypt the `data_block`:

- VCS retrieves the session key from `first key_block`
- VendorX uses the second `key_block`

Consequently, both implementations could successfully decrypt the data block which contains the encrypted IP.

Discontinued -ipkey Option

The `-ipkey key` option will be obsolete in future releases.

IP providers should use `-ipprotect` instead. It allows you to specify various protection pragmas (via a protection header file) which are needed while generating interoperably encrypted IPs.

VCS will no longer use the key you pass with the `-ipkey key` option. It will generate a secure key internally.

IEEE VHDL Standard 1076-2008 Encryption

VCS supports the encryption and decryption of the VHDL IP source code in protection envelopes, as defined by the IEEE VHDL Standard 1076-2008.

This section describes the following topics:

- [VHDL 1076-2008 Encryption Use Model](#)
 - [Protection Envelopes](#)
 - [The VCS Public Encryption Key](#)
 - [Usage Example](#)
 - [Debug Protection](#)
 - [Limitations](#)
-

VHDL 1076-2008 Encryption Use Model

You must specify the `-ipprotect` option on the `vhdlan` command line to use VHDL 1076-2008 encryption. The `-ipprotect` option tells VCS to encrypt the VHDL source files specified on the `vhdlan` command line according to the IEEE VHDL Standard 1076-2008 for encryption envelopes.

The syntax of the `-ipprotect` option is as follows:

```
-ipprotect <header_file>
```

Where, `header_file` is the protection header file that contains various protection directives. For more information on the protection header file, see [Using the Protection Header File](#).

In the encryption mode, VCS does not analyze VHDL source files, instead it encrypts each source file into a separate encrypted VHDL file. It saves each encrypted file with the same filename, but changes its filename extension by appending `p` at the end of it.

Example:

```
% vhdlan -ipprotect toolkeys /src/dir/ipmod1.vhd \ package5.vhdl
```

The above command line generates:

```
/src/dir/ipmod1.vhdp and package5.vhdlp.
```

The following sections describe the use models to encrypt the complete or parts of the VHDL source files:

- [Encrypting the Entire VHDL Source Files](#)
- [Encrypting the Parts of VHDL Source Files](#)

Encrypting the Entire VHDL Source Files

VCS encrypts the entire VHDL input files, by default. The following is the use model to encrypt entire VHDL source files:

```
% vhdlan -ipprotect <header_file> <VHDL_file_list>
```

During encryption, if a file contains a decryption envelope (`protect begin_protected), VCS generates an error message and stops the encryption.

Encrypting the Parts of VHDL Source Files

Use the `-ipopt=partialprotect` option to enable partial protection. The `-ipopt=partialprotect` option enables VCS to encrypt only the regions of the VHDL code specified between `protect begin and `protect end directives.

The use model to encrypt the parts of VHDL source files is as follows:

```
% vhdlan -ipprotect <header_file> -ipopt=partialprotect <VHDL_file_list>
```

`vhdlan` does not stop encryption if a file does not contain `protect begin or `protect end. It generates an error message and skips the encryption for that file.

Using the Protection Header File

The protection header file (`header_file`) specifies the protect directives as per the VHDL 1076-2008 LRM. It contains an additional directive, namely, `protect key_public_key, which is a base64 encoded public key.

The `protect key_public_key directive replaces the `protect key_block directive in the VHDL 2008 LRM. That is, the VHDL encryption requires that the key block (the sequence of key_keyowner, key_keyname, and key_method directives) ends with a `protect key_public_key directive.

The syntax of `header_file` is as follows:

```
`protect data_method = "data_method"
`protect key_keyowner="owner_name"
`protect key_keyname="owner_key_name"
`protect key_method="encryption_method_name"
`protect key_public_key
<content_representing_the_public_encryption_key>
```

where,

- The following `protect directives are required in header_file:
`key_keyowner, key_keyname, key_method, key_public_key`
- The following `protect directives are optional in header_file:
`data_method, encoding`
- The data block following the `protect key_public_key directive is an example of a base64 encoded version of a public key.

Table 61 describes the `protect directives in header_file.

Table 61 *The `protect Directives in header_file*

Directive	Description
data_method	Identifies the data encryption algorithm. The supported data encryption algorithms are as follows:3des-cbc, aes128-cbc, aes192-cbc, aes256-cbc, des-cbcThe default data_method is aes256-cbc.
encoding	Specifies the coding scheme for an encrypted data. VCS only supports the base64 encoding scheme.The default encoding scheme is base64.
key_keyowner	Identifies the owner of the encryption key.
key_keyname	Specifies the key name of keyowner.
key_method	Specifies the key encryption algorithm, that is, the asymmetric method for encrypting or decrypting. VCS supports the rsa encryption algorithm.
key_public_key	Specifies the public key for the key encryption.

Note:

- The `protect encryption directives are only supported in the header_file, which is specified by the -ipprotect option. If they are specified anywhere else, VCS generates a warning message and ignores the directives.
- The following directives are optional in the protection header file:

`author, author_info`

Options for VHDL 1076-2008 Encryption Mode

The options for the VHDL 1076-2008 encryption mode are as follows:

`-ipopt=partialprotect`

VCS encrypts an entire file by default. Use this option to encrypt only regions marked by the directives, namely ``protect begin` and ``protect end`, in VHDL source files.

`-ipopt=ext=<ext>`

Use this option to specify the filename extension for encrypted files.

Example:

```
% vhdlan -ipprotect toolkeys -ipopt=ext=de config6.vhd
```

The above command line generates:

`config6.vhdde`

`-ipopt=outdir=<dir>`

Use this option to specify the target directory for encrypted files.

Example:

```
% vhdlan -ipprotect toolkeys -ipopt=outdir=db6 /src/dir/ipmod1.vhd
package5.vhdl
```

The above command line generates:

`db6/ipmod1.vhdp` and `db6/package5.vhdlp`

`-ipout filename`

Use this option to allow VCS to write an encrypted file for the first VHDL source file on the command line with the specified file name and extension.

Example:

```
% vhdlan -ipprotect toolkeys -ipout encrypt.enc config6.vhd
```

The above command line generates an encrypted file, `encrypt.enc`.

`-ipopt=overwrite`

Use this option to allow VCS to replace any existing output files.

Note:

You must specify `-ipprotect` while using `-ipopt` or `-ipout` options; otherwise VCS generates an error message.

Protection Envelopes

The `protect encryption directives are used to form protection envelopes, which include specification of cryptographic methods and keys to be used by a tool. This section describes the functionality of the protection envelopes.

Protection envelopes work as follows:

1. The encrypting tool generates a random key called “session key”.
2. The encrypting tool encrypts a design using the session key.
3. Use `protect directives in the encryption envelope to provide the information about each potential decrypting tool.

This information includes key_keyowner, key_keyname, the asymmetric key_method, and key_public_key for each tool.

4. The encrypting tool encrypts the session key multiple times, once for each decrypting tool using its information provided in the encryption envelope.
5. The encrypting tool records the encrypted session in key blocks in the protected envelope. The tool then generates multiple key blocks, one for each decrypting tool.
6. The decrypting tool examines the key blocks in the decryption envelope to find the encrypted envelope using a key to which the tool has access.
7. The decrypting tool is able to recover the session key from its key block using an appropriate private key.
8. The decrypting tool decrypts the design with the session key.

The VCS Public Encryption Key

For the VCS base64 encoded RSA public encryption key, contact Synopsys support (vcs_support@synopsys.com). Synopsys also provides use cases and examples along with the key.

The following `protect directives identify this key:

```
`protect key_keyowner="IPcorp"
`protect key_method="rsa"
`protect key_keyname="IPcorp-123"
```

VCS can decrypt and analyze the source files, which are encrypted by VCS or third-party tools. To allow VCS to decrypt encrypted source files, include the following code snippet while encrypting:

```
`protect key_keyowner="IPcorp"
`protect key_keyname="IPcorp-123"
```

```
'protect key_method="rsa"
'protect key_public_key
<content_representing_the_public_encryption_key>
```

The following example illustrates the protection envelope methodology for using this key in Verilog or SystemVerilog source code.

Usage Example

VCS allows multiple key blocks in a single protected envelope, so you can decrypt it using the tools from different vendors.

Example for Full Encryption

[Figure 175](#) specifies `header_file` with the `-ippprotect` option.

Example 239 Example header_file for Source Encryption With VCS

```
'protect data_method = "x-abc"
'protect encoding = (enctype = "base64")
'protect key_keyowner="IPcorp"
'protect key_method="rsa"
'protect key_keyname="IPcorp-123"
'protect key_public_key
<content_representing_the_public_encryption_key>
```

Example 240 VHDL Source File to be Encrypted: `example.vhd`

```
library IEEE;
use IEEE.STD_LOGIC_1164.ALL;
use IEEE.STD_LOGIC_ARITH.ALL;
use IEEE.STD_LOGIC_UNSIGNED.ALL;

entity Counter is
Port(
clk : in STD_LOGIC;
Reset : in STD_LOGIC;
Count : out STD_LOGIC_VECTOR (3 downto 0);
Carry : out STD_LOGIC
);
end Counter;

architecture Behavioral of Counter is
    signal count_int : std_logic_vector(3 downto 0); -- define
    internal register
begin
    process (reset, clk)
    begin
        if reset = '1' then
            count_int <= "0000"; -- set counter, and
            carry <= '0';--carry to zero
        elsif clk'event and clk = '1' then
```

```

        if count_int <= "1000" then -- check count
            count_int <= count_int + "1"; --increment
            carry <= '0'; -- show still below 9
        else -- else we are at 9
            count_int <= "0000"; -- roll over count
            carry<= '1';-- flag roll over
            end if;
        end if;
        end process;
        count<= count_int;
-- send value to the outside
end Behavioral;

```

Perform the following commands:

The following vhdlan command line generates the encrypted file, example.vhdp, which can be decrypted by VCS:

```

% vhdlan -ipprotect header_file example.vhd
% vhdlan example.vhdp
% vcs Counter

```

Example for Partial Encryption

Refer [Figure 175](#) that specifies header_file.

Example 241 VHDL Source File to be Encrypted: example.vhd

```

library IEEE;
use IEEE.STD_LOGIC_1164.ALL;
use IEEE.STD_LOGIC_ARITH.ALL;
use IEEE.STD_LOGIC_UNSIGNED.ALL;

entity Counter is
Port(
clk : in STD_LOGIC;
Reset : in STD_LOGIC;
Count : out STD_LOGIC_VECTOR (3 downto 0);
Carry : out STD_LOGIC
);
end Counter;

`protect begin
architecture Behavioral of Counter is
    signal count_int : std_logic_vector(3 downto 0); -- define
    internal register
begin
    process (reset, clk)
    begin
        if reset = '1' then

```

```

count_int <= "0000"; -- set counter, and
carry <= '0';--carry to zero
elsif clk'event and clk = '1' then
  if count_int <= "1000" then -- check count
    count_int <= count_int + "1"; --increment
    carry <= '0'; -- show still below 9
  else -- else we are at 9
    count_int <= "0000"; -- roll over count
    carry<= '1';-- flag roll over
  end if;
end if;
end process;
count<= count_int;
-- send value to the outside
end Behavioral;
`protect end

```

Perform the following commands:

The following vhdlan command line generates the partially encrypted file, example.vhdp, which can be decrypted by VCS:

```
% vhdlan -ipprotect header_file -ipopt=partialprotect \ example.vhd
% vhdlan example.vhdp
% vcs Counter
```

Debug Protection

This feature only supports debug protection at the design-unit level. If any part of a design unit is encrypted, then all of it is protected against access from various debug tools.

This section discusses the following topics:

- [Combining Encrypted and Unencrypted Code](#)
- [Assertion and Report Statements](#)
- [Hierarchy Attributes](#)
- [Profiling](#)
- [VHPI](#)
- [VPD / VCD](#)
- [Coverage](#)
- [Error Messages](#)

Combining Encrypted and Unencrypted Code

A design can be unencrypted, entirely encrypted, or partially encrypted. This section consists of the following two sub-sections:

- [Entirely-Encrypted or Unencrypted Source Files](#)
- [Partially-Encrypted Source Files](#)

Entirely-Encrypted or Unencrypted Source Files

Generics, ports, declarations, concurrent statements in entity declarations, and architecture bodies are protected from debug as per [Table 62](#).

Table 62 Entirely-Encrypted or Unencrypted Source Files

Entity	Architecture	Generics	Ports	E. Decl	A. Decl	A. Conc
Plain	Plain	N	N	N	N	N
Plain	Crypt	N	N	N	Y	Y
Crypt	Plain	Y	N	Y	N	N
Crypt	Crypt	Y	N	Y	Y	Y

where,

E. Decl indicates declarations other than generics and ports

A. Decl indicates architecture declarations

A. Conc indicates architecture concurrent statements

N indicates not protected

Y indicates protected

Plain indicates plain text, not encrypted

Crypt indicates entire design unit encrypted

See [Table 63](#) to determine protection for packages and package bodies.

Table 63 Protection for Packages and Package Bodies

Package	Package Body	Package Declaration	Package Body Declaration
Plain	Plain	Not protected	Not protected

Table 63 Protection for Packages and Package Bodies (Continued)

Package	Package Body	Package Declaration	Package Body Declaration
Plain	Crypt	Not protected	Protected
Crypt	Plain	Protected	Not protected
Crypt	Crypt	Protected	Protected

Partially-Encrypted Source Files

Partially-encrypted code in architecture bodies, packages, and package bodies has the same consequences for protection as if the entire design unit is encrypted, except in the case of entity declarations, where protection is determined as per [Table 64](#).

Table 64 Partially-Encrypted Source Files

Encrypted Portion	Generic	Port	Declaration	Concurrent Statements
Context clause (all design units)	Not protected	Not protected	Not protected	Not protected
Entire entity except for context clause	Protected	Not protected	Protected	Protected
Entity generic clause	Protected	Not protected	Not protected	Not protected
Entity port clause	Not protected	Not protected	Protected	Protected

Assertion and Report Statements

As per IEEE VHDL Standard 1076-2008 LRM:

“If a decryption tool executes an assertion statement (see 10.3) that causes an assertion violation, or executes a report statement (see 10.4), the message shall not include the name of the design unit containing the statement, the rules of 10.3 and 10.4 notwithstanding.”

Messages from assertion and report statements exclude any hierarchy from an encrypted region.

Hierarchy Attributes

As per IEEE VHDL Standard 1076-2008 LRM:

“The value of any 'INSTANCE_NAME' or 'PATH_NAME' predefined attribute (see 16.2) formed by the decryption tool shall not include any element that is a name or label defined in a decrypted portion of a VHDL description, the rules of 16.2 notwithstanding.”

Profiling

Simulation time and memory profiles do not display information on encrypted design units.

VHPI

As per IEEE VHDL Standard 1076-2008 LRM:

“If a decrypted portion of a VHDL description includes an instantiation of a declaration that is declared in a portion of the VHDL description that is not encrypted, a decryption tool may provide access to a representation of the design sub-hierarchy whose root is the instance, provided the means of providing access does not contradict other requirements of this sub-clause. For example, a VHPI tool may return a handle to a VHPI object representing such an instance and allow navigation of associations from that reference object, provided the target objects represent parts of the design sub-hierarchy.”

VPD / VCD

Recursive VCD, VPD, or EVCD dumping does not dump objects or other information from encrypted design units.

Coverage

Coverage does not dump the information related to encrypted design units.

Error Messages

During analysis, if an encrypted code is encountered in the design unit, then error message is replaced with a generic message.

VCS displays error messages during elaboration and simulation, but hides the source code from the encrypted portions of a design.

Limitations

VCS does not support the following:

- Nested encryption envelopes in the source files
- Nested decryption envelopes

- The following protect directives:

```
data_keyname, data_keyowner, data_public_key, decrypt_license,
digest_block, digest_keyname, digest_keyowner, digest_key_method,
digest_method, runtime_license, viewport
```

- The following encrypt data methods:

```
blowfish-cbc, twofish128-cbc, twofish192-cbc, twofish256-cbc, serpent128-
cbc, serpent192-cbc, serpent256-cbc, cast128-cbc
```

- The following encoding types:

```
quoted-printable, raw
```

- The following asymmetric key methods:

```
elgamal, pgp-rsa
```

- VHDL 2008 encryption which specifies encryption envelopes to be present entirely in the source files.

128-bit Advanced Encryption Standard

VCS uses the 128-bit Advanced Encryption Standard (AES) to encrypt the Verilog files. The 128-bit key is generated internally by VCS. This 128-bit encryption methodology is exclusive to VCS, and can be decrypted only by VCS.

This section includes the following topics:

- [Compiler Directives for Source Protection](#)
- [Using Compiler Directives or Pragmas](#)
- [Automatic Protection Options](#)
- [Using Automatic Protection Options](#)
- [Protecting 'include File Directive](#)
- [Enabling Debug Access to Ports and Instance Hierarchy](#)
- [Debugging Partially Encrypted Source Code](#)

Compiler Directives for Source Protection

``protect`

Defines the start of protected code. Syntax: ``protect`

`'endprotect`

Defines the end of protected code. Syntax: `'endprotect`

`'protected`

Defines the start of protected code. Syntax: `'protected`

`'endprotected`

Defines the end of protected code. Syntax: `'endprotected`

`'protect128`

Defines the start of protected code. Syntax: `'protect128`

`'endprotect128`

Defines the end of protected code. Syntax: `'endprotect128`

Using Compiler Directives or Pragmas

You can use VCS to encrypt selected parts of your source files. In order to achieve this, complete the following steps:

-protect128

1. Enclose the Verilog code that you want to encrypt between the `'protect128` and the `'endprotect128` compiler directives.
2. Compile the files with the `-protect128` option. For example:

```
% vcs -protect128 foo.v
```

When you compile the design with the `-protect128` option, VCS creates a new file with the `.vp` extension for each Verilog file specified in the command line. For example, VCS creates `foo(vp` when you execute the command listed above.

In the `.vp` files, VCS replaces the `'protect128` and `'endprotect128` compiler directives with the `'protected128` and `'endprotected128` compiler directives and encrypts the code in between these directives.

Note:

- If you specify the `protect` and `protect128` compile options on the same `vcs` command line, VCS ignores the `protect128` option and uses the `protect` option. It also reports a warning message.
- The `protect128` and `genip` options are mutually exclusive, you cannot specify both of these options on the same `vcs` command line.

Example

The following Verilog file illustrates the use of `'protect128` and `'endprotect128` to mark the code that needs to be encrypted:

```
cat test.v
module counter( inp, outp);
  input [7:0] inp;
  output [7:0] outp;
  reg [7:0] count;
  always
begin:counter
  `protect128
  reg [7:0] int;
  count = 0;
  int = inp;
  while (int)
begin
  if (int [0]) count = count + 1;
  int = int >> 1;
end
  `endprotect128
end
  assign outp = count;
endmodule

module top;
parameter p1 = 3;
wire mux;
reg control,dataA,dataB;
dut #(p1(3)) d1(mux,control,dataA,dataB);
counter c1(inp,outp);
initial begin
  control=0;
  dataA=1;
  dataB=0;
#2; dataA=1;dataB=1;
#2;dataB=1'bx;
#2; dataA=0; dataB=0;
#2; dataB=1;
#2; dataB=1'bx;
#2 ;control=1; dataB=1;
#2; dataA=1;
#2;dataA=1'bx;
#2; dataA=0; dataB=0;
#2; dataA=1;
#2; dataA=1'bx;
#2; control=1'bx; dataA=0; dataB=0;
#2; dataA=1;dataB=1;
#2; $finish;
end
endmodule
```

```

primitive multiplexer(mux, control, dataA, dataB) ;
output mux ;
input control, dataA, dataB ;
table
// control dataA dataB mux
0 1 0 : 1 ;
0 1 1 : 1 ;
0 1 x : 1 ;
0 0 0 : 0 ;
0 0 1 : 0 ;
0 0 x : 0 ;
1 0 1 : 1 ;
1 1 1 : 1 ;
1 x 1 : 1 ;
1 0 0 : 0 ;
1 1 0 : 0 ;
1 x 0 : 0 ;
x 0 0 : 0 ;
x 1 1 : 1 ;
endtable
endprimitive

module dut #(parameter p1 = 1) (output mux, input control, dataA, dataB);
multiplexer m1(mux, control, dataA, dataB);
endmodule

```

The contents of the .vp file that are generated using the `-protect128` compile option are as follows:

```

always
begin:counter
`protected128
PWXH[Q[X&;D#.->0!
SIF<HI"D7X)2F-MZCTCK.R+U8;SAE3M.+ ,;N'/3.B=6%$ _5PHYD]E1G#<0,VW
A_>!1S/0%XYM98MW0'OA]?PNK:[T)*_]
IRSN+R.EE#]-I_JJRPA_#KZ+7$ \TIAY83B8L<U0!U.GK[V?\V,
=>JF:GK6"C8=\M5MB'!2+WY/7S5 &RONPGO!LK8#25
(CO>3N7N.YG%=FF'),"J90A8OS5$E2+ &4@T2Q!U?DOS;2 (O3G6G3T>
`endprotected128

```

-putprotect128 <Dir-name>

By default, the encrypted .vp file is saved in the same directory as the source files. You can change this location by using the `-putprotect128` compile option.

For example, the following command saves the `foo.vp` encrypted file in the `./out` directory:

```
% vcs -putprotect128 ./out -protect128 foo.vp
```

VCS creates a protected file in the specified directory. The '`./out/foo.vp`' protected file is created.

Automatic Protection Options

-autoprotect128

For Verilog and VHDL files, VCS encrypts the module port list (or UDP terminal list) along with the body of the module (or UDP).

-auto2protect128

For Verilog and VHDL files, VCS encrypts only the body of the module or UDP. It does not encrypt port lists or UDP terminal lists. This option produces a syntactically correct Verilog module or UDP header statement.

-auto3protect128

This option is similar to the `-auto2protect128` option except that VCS does not encrypt parameters preceding the ports declaration in a Verilog module.

+autoprotect[file_suffix]

Creates a protected source file; all modules are encrypted.

+auto2protect[file_suffix]

Creates a protected source file that does not encrypt the port connection list in the module header; all modules are encrypted.

+auto3protect[file_suffix]

Creates a protected source file that does not encrypt the port connection list in the module header or any parameter declarations that precede the first port declaration; all modules are encrypted.

+deleteprotected

Allows overwriting of existing files when doing source protection.

+pli_unprotected

Enables PLI and UCLI access to the modules in the protected source file being created (PLI and UCLI access is normally disabled for protected modules).

+protect[file_suffix]

Creates a protected source file by only encrypting `protect/`endprotect regions.

+object_protect <sourcefile>

Debugs the partially encrypted source code.

vcs +protect +object_protect <sourcefile.v>

`+putprotect+target_dir`

Specifies the target directory for protected files.

`+sdfprotect[file_suffix]`

Creates a protected SDF file.

`-Xmangle=number`

Produces a mangled version of input, changing variable names to words from list. Useful to get an entire Verilog design into a single file. Output is saved in the `tokens.v` file. You can substitute `-Xman` for `-Xmangle`.

The argument `number` can be 1, 4, 12, or 28:

`-Xman=1`

Randomly changes names and identifiers, and removes comments, to provide more secure code.

`-Xman=4`

Preserves variable names, but removes comments.

`-Xman=12`

Does the same thing as `-Xman=4`, but also enters, in comments, the original source file name and the line number of each module header.

`-Xman=28`

Does the same thing as `-Xman=12`, but also writes at the bottom of the file comprehensive statistics about the contents of the original source file.

`-Xnomangle=.first|module_identifier,...`

Specifies module definitions whose module and port identifiers VCS does not change. You use this option with the `-Xman` option. The `.first` argument specifies the module by location (first in file) rather than by identifier. You can substitute `-Xnoman` for `-Xnomangle`.

Using Automatic Protection Options

Note:

The `-auto3protect128` option takes precedence over `-auto2protect128` and `-autoprotect128` options, `-auto2protect128` takes precedence over `-autoprotect128`, and `-autoprotect128` takes precedence over `-protect128`.

-autoprotect128

For Verilog and VHDL files, VCS encrypts the module port list (or UDP terminal list) along with the body of the module (or UDP).

For example, the contents of the .vp file that are generated using the **-autoprotect128** option are as follows:

```
module counter
`protected128
P6O # ON'-, 5&.Y)AO )WH1MLZ6=M^=MG! HNZ [; ]%0^2CSHD; !"DA Y_*<7CQP.GB
P>NV,, 82,G9%HZBYEBWO@D^JP*HXZR8K\,) 1?'OI=-Q^T (@V7^I^@T&I1
[>.3GCO@ [ PWTN(F,CSX.ZH$37A3F/8IWXML[>/JUN8P\Q) Y=\FQ$J4M>
#. (31WZ' & (5&+%/L<RP0F+!$) E-U7!KA1Y!&5;S3>ID8RC) :@*V>X
YZ1NC:S"/F] !NX0NKG"K8X5&4D_#) %PV(Y%PFO?4*96PED9&SI:PGMM
(J?GOD$%XF8CV: ?#A_[^<QX3-;IC1) I3\ -8C%GIDPRR$%26.$L 'OZ5B4
6-_C10X,WOMU'Y'IM'CZ*;/CW=XYBE\L\,4.U =N<HY*O2I@
`endprotected128
endmodule

primitive multiplexer
`protected128
P"-9R8;C8?O\K)>&) $0%*8Q2_OQP5 (+NY%R&X+=G; QX@:=#$<CRS0A\&]
/IO&6+SFP+OK+-UK) $^ B*1NCC./ESFVG!_H2CYI3"+'T'*^-&*/#
P%<U:&I@] S=Y#2""]) I&P) .;YML_ #- [&7]>#5[9K@>9+L( Y8H$G\?TJ
&35W=*#-NKBM9] !HZ & (=B: ;$_] FUPE@T8Q+: 7*( Z+14ES2-^ZRJ(WX
#NV! 6;%>UM>VL0H(T0\+TRKYZG5) G'AK1)*F'$P=9LR \&;G#.
6D":CF71O@V/:&/;O3T491+,=A5((6LN"\U*J!,7>RQX2A1*DP,2J
PK_..KR$/((1C"+/^0"MHNPQ.,;D] [?NRD_X6W. XTPGP6-,0"<47*7>
$KYQS,-S<P84)%2K^O/:,>+0#4\]CJ) TA45&7H1$V@PJ$Q<=/PI9\5\
-3PSENY+K,) C-V.0E
`endprotected128
endprimitive
```

In this example all the module port lists and UDP terminal list along with UDP definition are encrypted.

-auto2protect128

For Verilog and VHDL files, VCS encrypts only the body of the module or UDP. It does not encrypt port lists or UDP terminal lists. This option produces a syntactically correct Verilog module or UDP header statement.

The contents of the .vp file that are generated using the **-auto2protect128** compile option are as follows:

```
module counter( inp, outp);
input [7:0] inp;
output [7:0] outp;
`protected128
P-62]23&H.F//I;K-+ [= WD$[*GB:L2U<9W,03Y<B_1=DRWLJV;'OM'P];^ [B5ZI
P" T)X^0Y.WRTK61I+D1_3=7\0D1C!%3+" (NR'K3$HKQA[FL@^).B. (P/
```

```

'''-X;, XUJP"# 4)M:.<4R2VAUA0TZM'61%_!;=,UY3/, P(=A$RA_/$
(EBK*>X>P8K,@'*LISO,PVGH"H+7,C4;@,.?X *HQLHHM3::F_E!
((8>BYMKVT8HA-4*N6EJU1OIU0T3]6@9!PAS43Y)QD_DI(J15%0KCEX[/+_
Y'7UC6<%D@;.0?I/-W$P[HNCB6A+X\9P</C6-$[.
`endprotected128
endmodule

primitive multiplexer(mux, control, dataA, dataB ) ;
output mux ;
input control, dataA, dataB ;
`protected128
P1T4VA5J%C(4VK!^U;R^"ND56SO3AG+*12MZ&7#<;&/_;Q1D4V >".4-
4Q#"(@T;P-P<'^#*2WG'8T/SNA(/:2Z*HK"$@L^D&AP@E;,P$O:9#PG3]
1X >DV?TZK/, S*:PMC+T1#65A@RYU+*=XFFMS^+C(8H9XL-Z-<J"E6%>
2N,%%:*U I>HQ2*F Z%D/QYPG32(5;P;D>X" ^^008])%)O&3/7/P)O"??
[B@\",E<Y,N'"(&R 05300;7X%3TV]QJP[H47-- DZ .]FAAJ^! V":T=
E0#PYJL\)\Y.:OGH%VW]D=R-.K_11S)4I-CU-P=&+
`endprotected128
endprimitive
endmodule

```

In this example it encrypts only the body of the module or UDP and does not encrypt port lists or UDP terminal lists.

-auto3protect128

This option is similar to the `-auto2protect128` option except that VCS does not encrypt parameters preceding the ports declaration in a Verilog module.

The contents of the `.vp` file that are generated using the `-auto3protect128` compile option are as follows:

```

module dut (mux,control, dataA, dataB);
parameter p1 = 1;
output mux;
input control;
input data;
input dataB;
`protected128
PR@#:B8A;TV_. 4184;Y,%!E@E-P8,WL)%D+%2C@JY0L3)_%J"P;8S*
ESYV_,38PAAX3?7V=/PD$@4E*9DK2U^R_0@2JUT:=#?D:0EX'+GLZ?8
S';N=FS!"S?D[I;E7
`endprotected128
endmodule

```

In this example it encrypts only the body of the module or UDP and does not encrypt port lists or UDP terminal lists.

```

module top;
parameter p1 = 3;

```

+protect option

1. Enclose the Verilog and VHDL code that you want to encrypt between the `'protect` and the `'endprotect` compiler directives.
2. Compile the files with the `+protect` option

For example:

```
% vcs +protect foo.v
```

When you compile the design with the `+protect` option, VCS creates a new file with the `.vp` extension for each Verilog file specified at the command line. For example, VCS creates `foo.vp` when you execute the command listed above.

In the `.vp` file, VCS replaces the `'protect` and `'endprotect` compiler directives with the `'protected` and `'endprotected` compiler directives, and encrypts the code in between these directives.

The contents of the `.vp` file that are generated using the `+protect` compile option are as follows:

```
always
begin:counter
reg [7:0] int;
count = 0;
int = inp;
`protected
Z370P(PNd1ZOKL9PH7?6a=LC8JB\Lf9dBES3T<#ZE58?b# [=#[ #_&) >
_3eL6_1aY7+c,@0BZF#U;</EHfdM&I1fI-@]#?U;Gef\PX2fJ?1.HQ
:M.X_>3CYc9_QUZ2R97VA^8IT3V/,Kf<9N^-MHS (=bBbN&BDPH\?$
`endprotected
```

+putprotect+<Dir-name>

By default, the encrypted `.vp` file is saved in the same directory as the source files. You can change this location by using the `+putprotect` option either at `vlogan` stage for three-step flow or at `VCS` stage for two-step flow.

For example, the following `vlogan` or `vcs` command saves the `testfile.vp` encrypted file in the `./out` directory:

```
% vlogan -sverilog +autoprotect +putprotect+$cwd/out testfile.v
```

or

```
% vcs -sverilog +autoprotect +putprotect+$cwd/out testfile.v
```

VCS creates a protected file under the specified directory. The protected file is '`./out/testfile.vp`'.

This option is not supported with `-protect128`, `-aotuprotect128`, `-auto2protect128`, and `-auto3protect128` options.

+autoprotect[file_suffix]

For Verilog and VHDL files, VCS encrypts the module port list (or UDP terminal list) along with the body of the module (or UDP).

For example, the contents of the `.vp` file that are generated using the `+autoprotect` compile option are as follows:

```
module counter
`protected
.423IPYX4.Z-JJ#MF2_EDM(7RN]634+76?=U?f-ZVLX(1?N<2UTZ())T4I2K)fXK
-@+EK?e=^Z\DLXU5XH0VQ19,>-9]6+gDJ7Rf431EgL=7#>Y1V,9+3-8&G>F[Q0C
4#B[FgQ#DUU<>_UR^I#D:eS(2+O15=^HMTY]f<XXU6=;4RP]f>?X,5d4B&X1T&UC
MAZQ[N=K6(>R>b2g/,HGEHMD/+W:38b[(6Lf4f@g)_Me#b\34E7ECQMDcHJKay?\cK:ZA]TbbMa]bAFX>fR&YC-MH[79#=CUUFG:>0RckOU\bI-&2I^_[K=LbUL9,GRF
U9)68:,CZ@Df[@(:PdEP2F)cWU7\K<[c)A?K,-9:C@c\F$`endprotected
endmodule

primitive multiplexer
`protected
T:=e^@R59Xg#P];gMBf9#>(d[ZD7J.Pa/8PSPY)=G1BaGT, //+QM5)T.a[/+e,D+
>g--ENRe-4GV(@7#UN0f_e/.dY.1Xbg-?9NZ0CTP-U^D@?Ja^8AF@&R=0CHd/VKV
--NN]RIS;Q2.A2RBE5_A,PJF@7</F6fH]AgM8N57RJ,.C>3KEWD4dN+V4B2a@<V:
5\QQJJ2O#/_=f/Ybf-\)/ERc_gM(Y,_3+?&?IGU_87ZLeYc;(SfcTePTRB]2LUR
4/,aMg?WIPS[A]+OUG]7,<]L4FP-8=_JPE]7O]&UbSdI+-F+_5gK[27NgXW4<0SD
Q2D>.d_;L89<Y[LFD0OME?fMA7b.5aa+^N/F#3[<[\N_<d5+>QKYQ>>+KZ,0/fbB
@ZTB7#P2RL=3Ud>e1CMa2<<7<\PIYR(S;$`endprotected
endprimitive
```

In this example all the module port lists, UDP terminal list along with UDP definition are encrypted.

+auto2protect[file_suffix]

For Verilog and VHDL files, VCS encrypts only the body of the module or UDP. It does not encrypt port lists or UDP terminal lists. This option produces a syntactically correct Verilog module or UDP header statement.

For example, the contents of the `.vp` file that are generated using the `+auto2protect` compile option are as follows:

```
module counter( inp, outp);
input [7:0] inp;
output [7:0] outp;
`protected
Y5S,#M;BJ&FL9:,U#/R;T;+)G:#XZD#NUZ58-U0RB;V?9JM?FcOI/)FE
:XM0I&#3LM.D]L2X0<:,89-DQ07GWM[Gc9LRc#7#IN:#1H@+CRBU-Z?G
```

```

O/c[9B;.Q9e@30IZM]7XR0LRXFI;FT4<&&M#+E6Z-].B(,ceZBDO4<fo
[Nd,O@#>a3\‐Df4EL[^SgXX^:#R+0‐d3MK^Wf(QY\WfLK?4IVdPXJFdHg
Ld(#./_NIYKaSOOMURg@00(C[1A\eo(<9WIT1,+Q8^e>fATb6\Y2K7@9f>
SK=>\H20&86;Y6;6KD$
`endprotected
endmodule

primitive multiplexer(mux, control, dataA, dataB) ;
output mux ;
input control, dataA, dataB ;
`protected
X-NKV3Ld>NGW@?2WZKeWBaZJ:[IUV+=H[?BKE&:#@+B;fSa^YL<,)dJ-
2HF.#A89K?K4+WT11Id9R<CJ^@=Q5KF(Y^S#\L5#bEdP:ag49F;=b15
CHfW</f?Sa.9=+^Id^OWN^‐IAX^.AU<^81T2AB=4+Ce) ]KFAYBRD>DT>L
#Z/7;;YC>KRBJ3GHLKT;_<V&6V?(WJa#W//.QRcW&OCG^R#A.+0HH(>=
(((SegQ]YC0,G2^4.03B9@Uf/@a_OG=-++Rc?A2/J^_;MdG-C>S_NaH\WM
f#BY6;VR_2451C2<7A].Nb^\\3BP$`endprotected
endprimitive

module dut #(parameter p1 = 1) (output mux, input control, dataA, dataB);
`protected
.5(K9-=#4,NGO2NY+&g+^bN,SN4f) )0]J2PC) & (?9+WKdaGS\C+) )
;KP>;I_@1B3?SFLc5U&B;,?,S==2_;4K-PD,-I=1E\`a8^YC=-/)I9f--`endprotected
GE24UBaD9CFGd;BHJKU$`endmodule

```

In this example it encrypts only the body of the module or UDP and does not encrypt port lists or UDP terminal lists.

+auto3protect[file_suffix]

This option is similar to the `+auto2protect` option except that VCS does not encrypt parameters preceding the ports declaration in a Verilog module.

The contents of the `.vp` file that are generated using the `+auto3protect` compile option are as follows:

```
module top;
parameter p1 = 3;
```

In this example it encrypts only the body of the module or UDP and does not encrypt port lists or UDP terminal lists.

+deleteprotected

Allows overwriting of existing files when doing source protection using the `+protect` option.

This option is not supported with `-protect128`, `-autoprotect128`, `-auto2protect128`, and `-auto3protect128` options.

+pli_unprotected

Enables PLI and UCLI access to the modules in the protected source file being created (PLI and UCLI access is normally disabled for protected modules).

This works with both `+protect` (all variants) and `-protect128` (all variants). To enable PLI capabilities, use the `+pli_unprotected` option as follows:

```
% vcs +protect +pli_unprotected <sourcefile.v>
```

or

```
% vcs -protect128 +pli_unprotected <sourcefile.v>
```

Protecting 'include File Directive

You can use VCS to automatically protect '`include` file directive while protecting the module.

+autoincludeprotect

For Verilog files, VCS encrypts the '`include` files using the switch `+autoincludeprotect`.

```
vlogan +autoincludeprotect test.v <auto-protect switches>
```

Or,

```
%vcs +autoincludeprotect test.v <auto-protect switches>
```

Consider that the source file `a.v` include `b.v` as shown below:

```
`include "b.v"
module a();
endmodule
```

After encryption, `b.v` is encrypted and is renamed to `b.vp`. The encrypted `b.vp` file along with the source file `a.vp` is saved in the same directory specified by `-putprotect128` compile option. The directive changes to '`include b.vp` as shown below:

```
`include "b.vp"
module a();
endmodule
```

Note:

This option is not supported for VHDL.

Enabling Debug Access to Ports and Instance Hierarchy

You can use VCS to enable debug access to port and instance hierarchy.

+autobodyprotect

For Verilog files, VCS enables debug access to port and instance hierarchy using the switch `+autobodyprotect`.

```
%vcs +autobodyprotect test.v
```

Hence, port list containing parameter and instance hierarchy of each module are accessible only to FSDB for Verdi users.

Note:

This option is not supported with `+autoprotect` and `-autoprotect128` options.
 This option is not supported for VHDL.

Debugging Partially Encrypted Source Code

The partial encrypted code is a code that has some of its part enclosed with the `'protect` and `'endprotect` macros. VCS allows you to debug the objects that are not enclosed within `'protect` and `'endprotect` while restricting access to the variables that are within `'protected` and `'endprotected` macros.

Note:

When you enclose a part of code using `'protect` and `'endprotect`, VCS converts it into `'protected` and `'endprotected` when you pass `+protect`.

To debug the partially encrypted source code, use the `+object_protect` option as follows:

```
vcs +protect +object_protect <sourcefile.v>
```

You can enable partial debug capability by adding the `+object_protect` option in the vcs encryption command line, so that partial encryption is applied and the encrypted file is also enabled with debug capability (`-debug_access+f+drivers -debug_region+encrypt`) for the unencrypted objects.

Skipping Encrypted Source Code

VCS allows you to skip some portion of the code unencrypted when the complete module is encrypted with `autoprotect` options. You can use `'unprotect` and `'endunprotect` pragmas to mark a block of source code to be excluded from encryption.

All autoprotect options ignore protection when you use '`unprotect`' and '`endunprotect`' pragmas.

Enclose the Verilog code that you want to decrypt between the '`unprotect`' and the '`endunprotect`' compiler directives. Enclose the VHDL code that you want to decrypt between the '`-unprotect`' and the '`-endunprotect`' compiler directives.

Note:

`unprotect` and '`endunprotect`' pragmas do not work with `-Xman=4` (for `tokens.v` file) and `-Xrad=0x2` (for `rad.v` file) options.

gen_vcs_ip

VCS allows you to protect a VHDL or a Verilog source file using the `gen_vcs_ip` utility as shown below:

```
% gen_vcs_ip -ipdir my_dir -e "vhdlan file1.vhd"
% gen_vcs_ip -ipdir my_dir -e "vlogan file1.v"
```

The protected IPs are platform and release independent. You share these protected IPs with your vendors.

The protected IP files are saved under the directory specified with the option `-ipdir dir_path`, and are named as `file1.vhd.e`, `file1.v.e` and so on. The `gen_vcs_ip` utility also writes the `analyze.genip` script, which can be later used to analyze all the protected files.

IPs protected using `gen_vcs_ip` are black box, and, therefore, are not in user readable format. Except for the ports of the protected design unit, none of the internal signals or variables can be accessed by any UI, GUI or PLIs. These black box IPs do not allow the following:

- Access by XMR paths to any object within or through the generated IP.
- PLI access (acc, tf, vpi, vhpi) to objects that reside in generated IP.
- Dumping (vcd or vpd files) any objects (signals or variables) that reside in generated IP.

You can use the `-debug` option to create the protected modules, whose ports are visible, and the internal signals and variables can be accessed using Synopsys UI, GUI or PLIs.

For example:

```
% gen_vcs_ip -ipdir my_dir -debug "vhdlan file1.vhd"
% gen_vcs_ip -ipdir my_dir -debug "vlogan file1.v"
```

The IP protected using the `-debug` option is a grey box and using VCS, UCLI, GUI, VHPI, VPI or MHPI, IP consumer can:

- View the ports at the boundary of the IP
- View the complete design hierarchy
- View all the internal signals or variables
- Query the value of signals or variables
- Set callbacks on value changes of the signal
- Use the `force` command to change the value of the signal
- Monitor the loads and drivers of the signal

Along with the specified design files, the `gen_vcs_ip` utility also protects the Verilog library files specified using '`include`', `-v` and `-y` options.

For example:

```
% gen_vcs_ip -ipdir VCSIP_DIR -e "vlogan top.v -v lib1/sub.v"
```

In the above example, the `gen_vcs_ip` utility protects both `top.v` and `sub.v`, and the protected files are saved under the `VCSIP_DIR` directory.

Syntax

```
% gen_vcs_ip -ipdir [ipdir_name] -debug \
-e "[analysis_command/script]" -parse -noencrypt
-top_name <IP Module/Entity Prefix>
```

Analysis Options

`-ipdir [ipdir_name]`

Physical directory where IP files are generated.

`-debug`

Generates binary IP files, whose ports are visible, and whose internal signals and the variables can be accessed using Synopsys UI, GUI or PLIs.

`-e`

Specify `vhdlan/vlogan` command line. You can also specify a make command or a run script.

`-parse`

Parses and does semantic checking of the design files.

`-noencrypt`

Used to package design files without encryption.

`-top_name <IP Module/Entity Prefix>`

It is possible that different IPs can use the same module name and can create module name conflicts. To resolve these conflicts, you need to make sure that all module/entity names inside an IP are unique. The `-top_name<IP Module/Entity Prefix>` option can be used to provide a prefix for all the IP modules being encrypted.

For example,

```
% gen_vcs_ip -top_name IP1 .....
```

All the modules in this IP will be prefixed with “IP1”.

```
% gen_vcs_ip -top_name IP2 .....
```

All the modules in this IP will be prefixed with “IP2.”

Thus, there is a lesser possibility that the module names from different IPs conflict.

Note that this switch is not intended to specify the top level module.

Note:

- VCS protects the library files specified with the `-y` and `-v` options and places in the directory where the IP model is generated.
- If you specify multiple `-y [lib_dir]` options, and if multiple files with the same file name exist in different library directories, the file that exists in the last directory you specify overwrites the others. In this case, VCS issues a warning message indicating from which library the module is picked up.

Exporting The IP

After protecting the IP, you can tar the generated IP directory and ship it to the IP consumer. To use the IP, the IP consumer should extract the IP directory and execute the `analyze.genip` script to analyze the protected files.

Use Model

IP Vendor

Synopsys recommends you analyze, elaborate and simulate the design before you protect them. This ensures that you are protecting the right set of source files.

Analysis

Always analyze Verilog before VHDL.

```
% vlogan [vlogan_options] file1.v file2.v
% vhdlan [vhdlan_options] file3.vhd file4.vhd
```

(The VHDL bottom-most entity first, then move up in order)

Elaboration

```
% vcs [elab_options] top_module/entity/config
```

Simulation

```
% simv [run_options]
```

IP Generation

```
% gen_vcs_ip -ipdir ip_dir -e "analyze.csh"
```

Note:

analyze.csh contains vlogan, and vhdlan command lines to analyze the Verilog and VHDL design files.

IP User

The usage model to use the protected IP is shown below:

Analysis

```
% ip_dir/analyze.genip
```

Elaboration

```
% vcs [elab_options] top_module/entity/config
```

Simulation

```
% simv [run_options]
```

Licensing

You require a “DW-Developer” license to protect an IP. However, a license is not required to use the protected IPs.

30

VCS Fine-Grained Parallelism Technology

This chapter explains how to use the new performance features in this release, in the following sections:

- [Introduction](#)
 - [Use Model](#)
 - [Profiling to Detect Design Suitability for Parallelism](#)
 - [Limitations](#)
-

Introduction

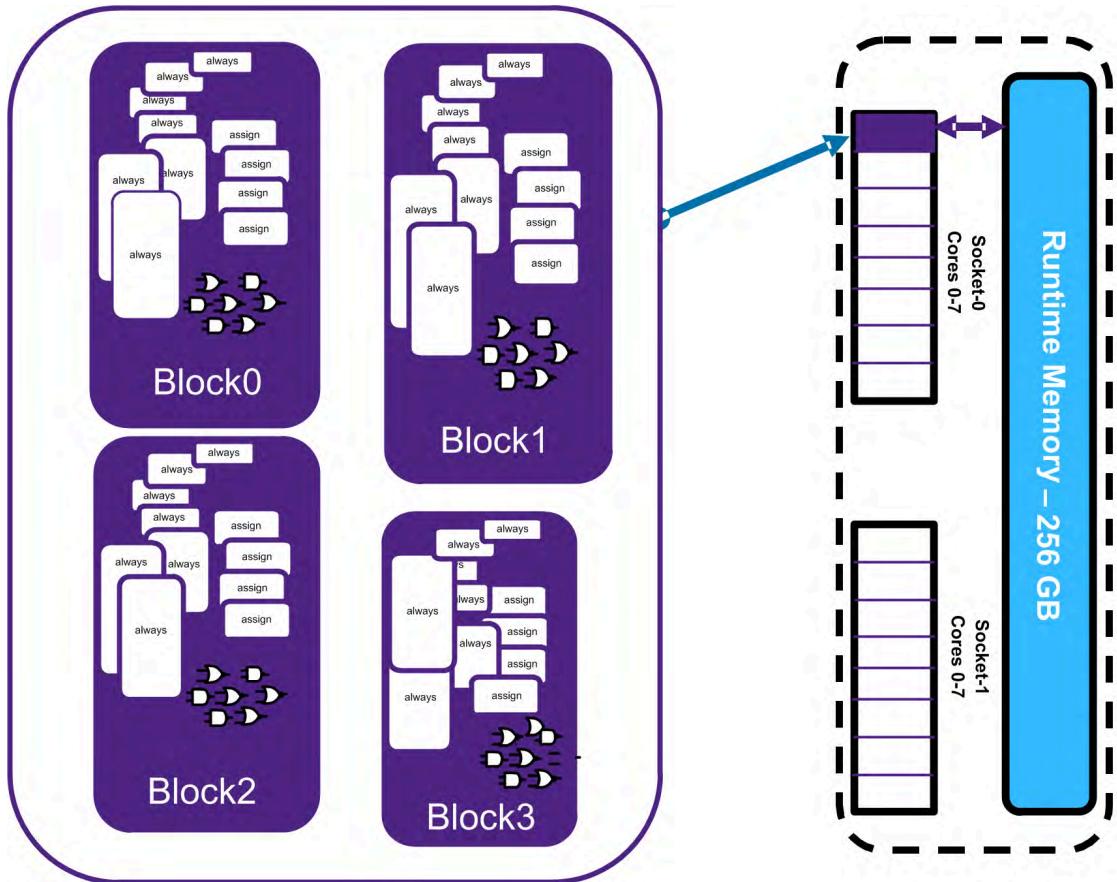
Traditionally, simulation performance has been a function of CPU performance, memory efficiency, and serial algorithm. However, advancements in processor hardware and instruction set architectures are providing the opportunity for order-of-magnitude simulation performance gains.

The rapid growth in the number of available cores per processor, in addition to the fast evolving vectorization support, enables next-generation simulation technologies, such as VCS' new Fine-Grained Parallelism (FGP). With this technology, you can gain the complete advantage of the newly available compute power to reduce the total turnaround time for simulation.

VCS FGP technology is designed to take advantage of the heterogeneous environments that consists of any combination of processors and vector cores. It improves the performance by adapting its algorithms dynamically to completely utilize all the available cores. This optimizes the simulation for latency sensitive tests.

Consider the following scenario, shown in [Figure 177](#).

Figure 177 Serial Simulation

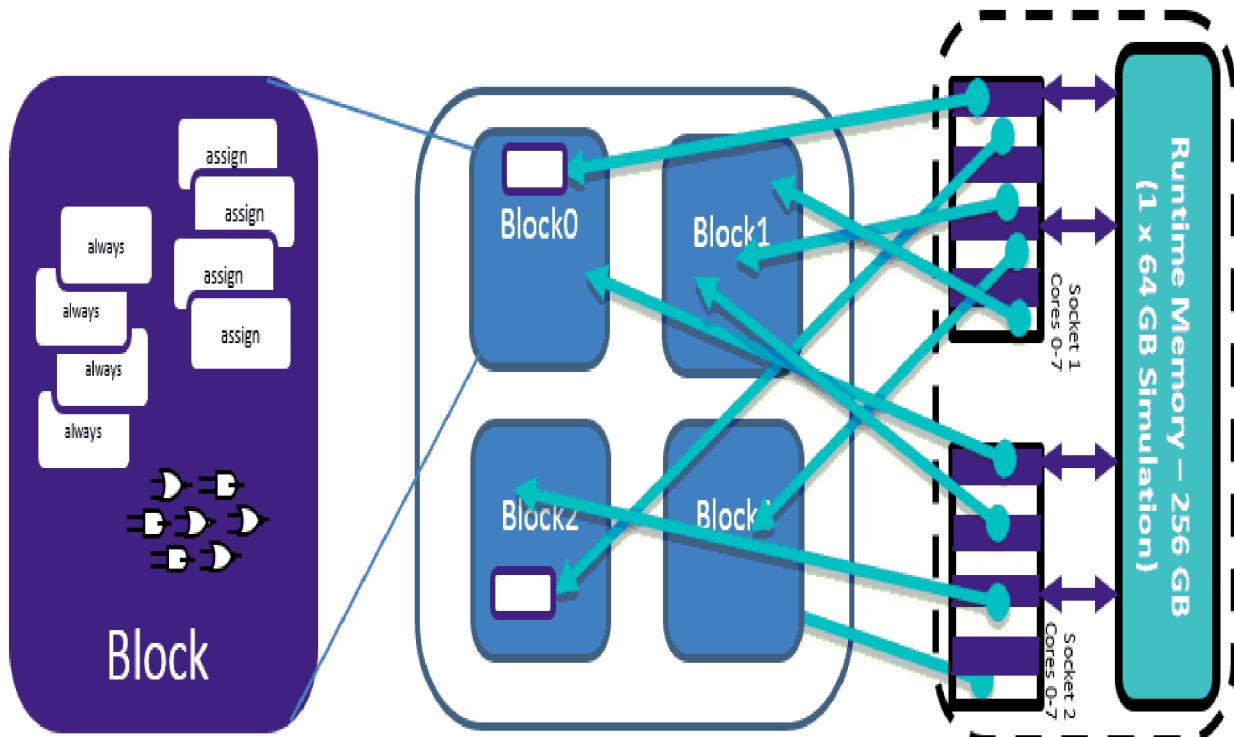


In Figure 177, entire block is executed on a single core in a single thread. Depending on the available memory, this might lead to available cores remaining unused, which might result in decreased utilization and efficiency.

Now, a HDL block is comprised of many events, such as `always` blocks, gates and assignments (shown in Figure 178). This enables you to parallelize events within a block on many CPU cores. While this can achieve significant performance gains, sometimes the gains are limited, if the processes are too large. To break these limitations, large processes are broken down further into parallel micro tasks to ensure well-balanced activity on each CPU core.

This method of partitioning a design into parallel micro tasks and events and scheduling them on multiple threads is the underlying principle of VCS FGP. In addition to this, VCS FGP further optimizes the simulation performance by adapting its algorithms for task scheduling, load balancing, and cache efficiency for target processor architectures. This enables you to take full advantage of the provisioned compute power.

Figure 178 Block Composition and Fine-Grained Parallelism



VCS FGP technology is inherently part of VCS M-2017.03 simulation engines. Therefore, it natively and transparently supports the crucial simulation features that you can rely on, such as Native Low Power (NLP), X-Propagation simulation, SDF, coverage (code coverage and functional coverage), and fully parallelized FSDB dumping for Verdi Debug (fully parallelized FSDB dumping is essential for parallel simulation technology to enable full performance speedup while maintaining the debug accuracy).

These simulation features continue to work as before with no changes needed to the design, or testbench and no disruption to the existing VCS simulation environment. Furthermore, VCS FGP delivers these performance gains with minimal impact to the memory usage and compile time.

In summary, VCS FGP technology provides breakthrough simulation performance improvements using the existing x86 hardware.

Use Model

VCS FGP use model is simple and is easy to deploy with no changes needed to the design and minimal changes required to the VCS verification environment.

Designs that get the most benefit from using VCS FGP are those that have a high-level of design activity. VCS provides a full range of profiling, analysis, and reporting tools to help determine the design suitability for FGP.

FGP is supported in both VCS two-step and three-step flows.

For two-step flow and three-step flow, add the option `-fgp` in VCS command line. At runtime, you need to specify the number of cores that you want to utilize for the parallel engines. So, if you specify 12 threads for the parallel engines and run the top design, you can see 13 cores being utilized for your simulation as one extra core is used by the master for running serial and parallel simulation.

You do not need to recompile to change the number of cores used at runtime.

Two-step flow:

```
% vcs -fgp[=compile_time_sub_options] -full64 <otherOptions>
% simv -fgp=num_threads:<value>[,runtime_options] Or simv
-fgp=num_threads:<value> -fgp=[runtime_options]
```

Three-step flow:

```
% vlogan -full64 <otherOptions>
% vhdlan -full64 <otherOptions> (for Mixed-HDL design only)
% vcs -fgp[=compile_time_sub_options] -full64 <otherOptions>
% simv -fgp=num_threads:<value>[,runtime_options] Or simv
-fgp=num_threads:<value> -fgp=[runtime_options]
```

Where, the `-fgp` option is used to enable FGP at compile-time or at runtime.

If you specify the `-fgp` option at compile time, and do not specify the option at runtime, `simv` runs on a single core.

Note:

The `-fgp` option works only in 64-bit mode.

You can use the `num_threads:<value>` runtime option to specify the number of cores. `<value>+1` cores are picked that includes one master core and `<value>` child cores.

`simv` collects a list of free cores. A core is determined to be loaded or free based on the load threshold. If the number of free cores is less than the requested number of threads (`<value>+1`), then VCS generates an error. Otherwise, VCS uses `<value>+1` cores out of the list (first use the cores on one socket) and bind threads to them. If enough free cores are not available, VCS generates an error message.

For example, for `auto_affinity` and `num_threads=15`, VCS generates the following error message, where the assumption is that 15 cores are available in the system whereas only 14 cores are available:

```
Error-[FGP_AFFINITY_FAILED] cpu_affinity/auto_affinity failed
Failed to allocate CPU cores: Simulation threads requested 15 cores but
only 14 core(s) are available in system.
For more details on status of CPU cores, please use runtime option
-fgp=diag:ruse.
```

Several sub-options are available under the `-fgp` option that allows you to control various functionality.

Compile Option for -fgp

The following sub-option is supported under the `-fgp` option during compilation:

`multisocket`

When you use this compile option, the memory binding for the thread takes into consideration the socket that is running the thread. This helps in further optimizing the performance.

Runtime Options for -fgp

The following are the additional runtime arguments that are optional:

`sync:<scheme_value>`

Use this option to specify thread synchronization scheme. The default synchronization scheme used for optimal performance is `-fgp=sync:busywait`, which is best for optimizing simulation performance. You can also select any of the following synchronization schemes to override the default scheme:

- `mutex` - Specify this scheme to use `pthread_mutex` based synchronization.
- `serial` - Specify this scheme to run the threads serially. This is used to debug any potential thread race issue.

`cpu_affinity`

Allows you to specify fixed list of cores that can be used. For example, consider the following configuration:

```
...
node0 CPU(s): 0-4,10-14
node1 CPU(s): 5-9,15-19
```

When you use the following runtime command:

```
% simv -fgp= num_threads:8,cpu_affinity:\(6-9,16-19\)
```

9 cores are selected from the 10 available node1 CPUs.

`single_socket_mode`

Use this option to use all the available cores on the given socket. If you enable dumping, some cores are shared with the FSDB dumping threads.

When simulation starts, VCS detects the number of cores that are available on the current socket. All cores are treated as available irrespective of the current CPU load. When the simulation threads are created, VCS distributes these threads to the available cores. If the simulation threads do not use all the assigned cores, the simulation continues, but VCS generates a warning message.

The following sub-options can be used with `single_socket_mode`:

`use_least_locked_socket`

This option enforces the selection of the socket with the least number of cores that are locked by other processes.

`num_cores:<value>`

Use this option to specify the number of cores. The number of cores, specified as `<value>`, are picked that includes one master core and `<value>-1` child cores.

`min_num_cores:<value>`

Use this option to set the minimum number of cores required by simulation. If there are less vacant cores than the `min_num_cores`, the simulation terminates. You can use the following command at runtime:

```
% simv -fgp=min_num_cores:<value>
```

Where `<value>` sets the lower limit for the number of cores for simulation. For example, when you use the following runtime command:

```
% simv -fgp=min_num_cores:9
```

9 is the minimum number of cores and the simulation terminates if there are less than 9 cores.

`allow_less_cores`

Use this option to change the hard limit of “`<value>` number of cores are required by the simulation” to the soft limit as “Maximum `<value>` number of cores are required by the simulation”. Therefore, if there are less than the specified number of cores available, simulation does not quit with an error but assigns the available number of cores to the

simulation and continue to run. To enable this feature, use the following command at runtime:

```
% simv -fgp=allow_less_cores
```

If there are 0 cores available, the simulation quits with an affinity error indicating the lack of resources.

```
num_fsdb_threads:<value>
```

Use this option to indicate VCS that dumping is involved. This option is mandatory when FSDB dumping is enabled. VCS allocates as many cores as specified by <value> for dumping.

When <value>=0, FSDB dumping runs on the master core (core that runs all the non-parallel portion of the design) and VCS generates a warning message. The remaining cores of the socket are used for FGP.

When <value>>0, additional cores are allocated for dumping.

When the num_fsdb_threads:<value> option is not specified, VCS generates the following error:

```
Error-[FGP_NO_FSDB_DUMPING_THREADS] No cores specified for FSDB dumping
Exclusive cores for FSDB dumping must be specified in non-mutex
synchronization mode.
```

Specify exclusive cores for FSDB dumping with -fgp=num_fsdb_threads option.

```
fsdb_adjust_cores
```

The number of cores are specified using the -fgp=fsdb_num_threads:<value> sub-option. Consider <value> as M. Adding the fsdb_adjust_cores switch converts the M number of cores from a hard limit of M number of cores to a soft limit of maximum of M number of cores required for FSDB dumping.

If all the cores are assigned to simulation, then the least loaded core is used for FSDB dumping. During simulation, the core switches dynamically depending upon the load on the simulation cores.

Following is an example use model command line:

```
%simv -fgp=num_cores:N -fgp=num_fsdb_threads:M -fgp=fsdb_adjust_cores
```

Under single socket mode, if you request N cores for simulation and M cores for FSDB dumping, the affinity behavior is as follows:

If the total number of cores = 0, the simulation shows an error.

If $0 < \text{available cores} \leq N$, VCS assigns all available cores to simulation. Assuming simulation to be heavier and higher priority process than dumping, dumping switches dynamically between cores depending upon the load.

If $N < \text{available cores} \leq N+M$, VCS assigns N cores for simulations and rest of the cores for dumping.

If $M+N < \text{available cores}$, VCS assigns N cores for simulation and M cores for dumping.

For example,

Number of cores (N)	Number of FSDB Threads (M)	Available Cores	Actual Simulation Cores Allocated	Actual Dumping Cores Allocated
6	3	0	Error	Error
6	3	5	5	0
6	3	6	6	0
6	3	8	6	2
6	3	9+	6	3

`diag:ruse`

Use this option to print the thread resource usage statistics at the end of the simulation. It generates the `fgp_diag_profile.txt` file with the details for each thread. You can find the data with thread divergence, imbalance, master time, and total time in this file.

`schedpli`

This option improves FGP runtime performance when the design has large number of PLI call-backs at runtime.

`auto_affinity:allowHyperThreadCpu`

In a hyper-threading enabled machine, each physical CPU core can be divided into virtual cores. By default, FGP simulation uses only physical cores. The `-fgp=auto_affinity:allowHyperThreadCpu` option allows FGP to run simulation on virtual threads and physical cores interchangeably.

For example, consider a hyper-threading enabled machine with four physical CPU cores. To run FGP simulation, you can use the following option: `%simv -fgp=auto_affinity:allowHyperThreadCpu -fgp=num_threads:<value>`

Where `<value>` = 2 to 7.

When the `auto_affinity:allowHyperThreadCpu` option is not specified for FGP simulation, then `<value>` = 2 or 3.

`auto_affinity:maxLoadForAvailCpu+<load_threshold_value>`

Use this option to modify the load threshold above which a core is not considered for FGP simulation. The default load threshold value is 0.1.

Running FGP simulation on a highly loaded core slows down the other job(s) running on the core and degrades the FGP runtime performance. It is recommended to use only lightly loaded cores for FGP.

Profiling to Detect Design Suitability for Parallelism

Designs that get the most benefit from using VCS FGP are those that have a high-level of design activity.

Designs that do not get benefit include designs that spend a significant portion of the overall time in the procedural testbench code, or have a high PLI or DPI content. Other examples include designs that have low activity. The following are the profiling tools to detect the design suitability:

- Simulation Profile: Tells details on the time spent in DUT, shown in [Figure 179](#).

Figure 179 Simulation Profile

Time Summary View		
Component	Time	Percentage
VERILOG	45199.29 s	70.98 %
Module	44942.50 s	70.57 %
Interface	147.36 s	0.23 %
Package	109.33 s	0.17 %
Function Coverage Kernel	80.12 ms	0.00 %
Functional Coverage	20.03 ms	0.00 %
HSIM	10036.67 s	15.76 %
KERNEL	8289.98 s	13.02 %
Garbage Collection	2.69 s	0.00 %
PLI/DPI/DirectC	138.07 s	0.22 %
CONSTRAINT	13.67 s	0.02 %
ASSERTION_KERNEL	2.98 s	0.00 %
PLI/DPI	20.03 ms	0.00 %
total	63680.69 s	100%

This profile report shows that this is a good design for parallelism because less time is spent on the testbench code and on the PLI/ DPI content.

For more information on simulation profile, see “The Unified Simulation Profiler” section in *VCS User Guide*.

- Timeline Profile: Tells event activity of the simulation.

The `-Xdprof=timeline` runtime option in the `simv` command line generates the DProf (event-counter based profiler) report under the current simulation working directory, shown in [Figure 180](#).

Figure 180 Timeline Profile

	Events	Events%	Time(s)	Time%	SimCycles	AvgTimePerCycle	(micro-seconds)
EPC>=100K	5,189,573,399	20.6	111.83	6.0	37,265	3000.83	
EPC[10K,100K]	17,981,521,541	71.3	1,428.68	76.7	464,480	3075.86	
EPC[1K,10K]	451,249,881	1.8	50.26	2.7	207,012	242.80	
EPC[100,1K]	232,112,673	0.9	44.42	2.4	636,677	69.76	
EPC[10,100]	772,581,317	3.1	72.29	3.9	17,736,179	4.08	
EPC<10	603,898,949	2.4	156.29	8.4	240,325,678	0.65	
Time-0	107,977,656	NA	434.41	NA	1	434407374.94	

Note:

Note:

If the top three rows of EPC (events per cycle) contribute to 80% and above, then it is better for parallelism. If the EPC of the last two rows dominate over other rows, this implies that it is a very low activity design. Such designs can actually slowdown with FGP when run with higher number of threads. The log file name where the EPC is generated is `dprof.txt`. Using the `diag:ruse` option at runtime, the switching activity is dumped in the report file.

Limitations

The feature has the following limitations:

- Analog (Spice and Verilog-AMS) is not supported.
- For multiple socket optimization, the `cpu_affinity` runtime option is not supported. Only the `auto_affinity` runtime option is supported.

31

Integrating VCS With Certitude

This chapter provides a brief description on the Certitude tool and how VCS works with Certitude.

Introduction to Certitude

Certitude is a functional qualification tool. This tool enhances simulation-based functional verification by providing measures and feedback to assist in improving the quality of the verification environment (VE).

To detect errors, the functional verification environment must ensure that each error is activated, propagated, and then detected.

Mutation-based techniques used by Certitude helps improve the testbench by identifying the flaws in the testbench. Generating the coverage metrics using simulation and performing the testbench qualification are sequential activities. This necessitates the need to perform redundant steps to setup the tool, one time for simulation and secondly for functional qualification of the testbench, which involves generation of database and fault-aware simulation executable.

VCS and Certitude Integration

With VCS and Certitude integration, you can generate VCS and Certitude databases with the same compilation. By eliminating the dependency on different parsers, you can generate the fault-aware simulation executable without having to prepare the design for testbench qualification separately. This seamless integration:

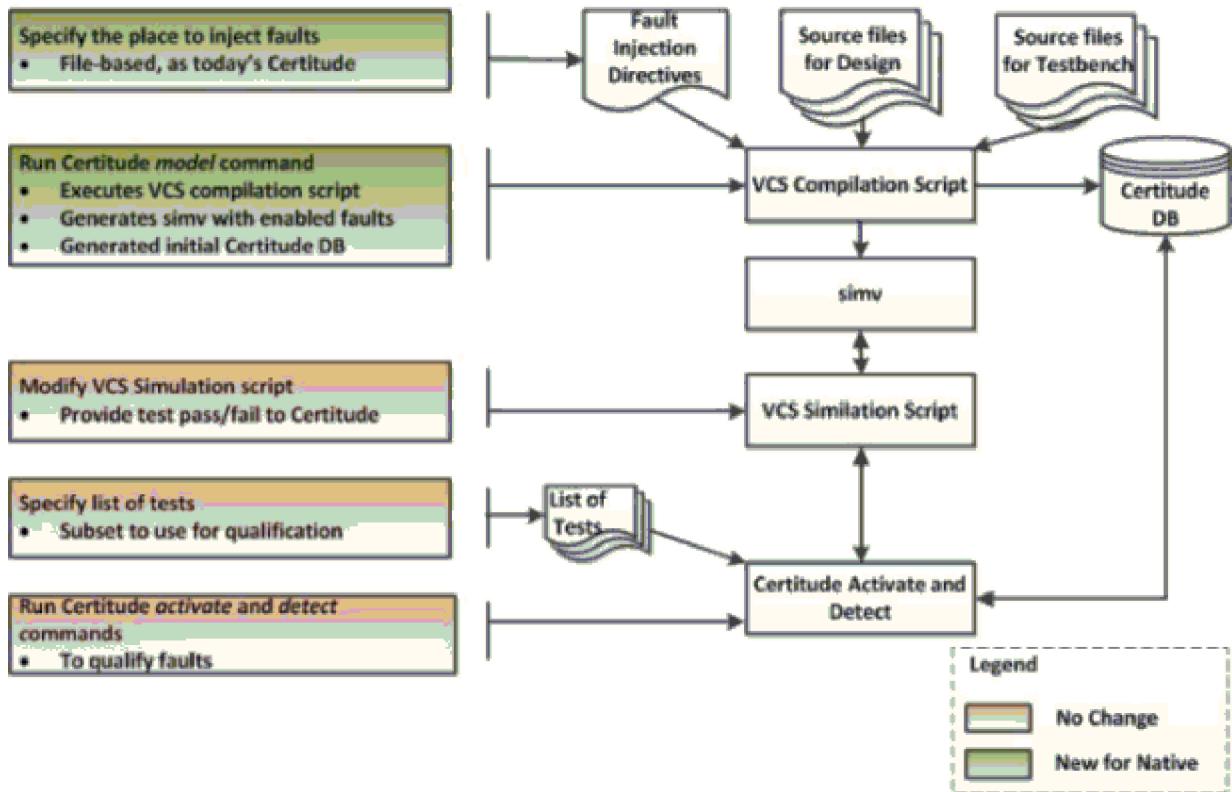
- Simplifies the setup requirements of Certitude environment with VCS
- Generates one or more simulation executables that contain all the instrumentation necessary to activate and detect faults.

With this integration, VCS generates the following database and executables:

- A Certitude database that corresponds to the Certitude model command.
- One or more simulation executables that contain all the instrumentation necessary to run activate, detect, regress and testscript commands.

Figure 181 shows the Certitude integration with VCS.

Figure 181 Certitude Integration with VCS



The Certitude Functional Qualification System requires a specific set of configuration files for a qualification run. These configuration files describe the verification environment. You must create these files before running a qualification phase. For more information about configuration files and use model, see the *Certitude User Manual*.

Note:

Certitude communicates with VCS internally through a combination of environment variables and the Certitude database. This process is transparent.

Loading Designs Automatically in Verdi with Native Certitude

With the integration of Certitude, VCS, and Verdi, you can load designs automatically in the Verdi system without setting the Certitude `VerdiInitCommand` configuration option.

This section consists of the following subsections:

- [Use Model](#)
 - [Points to Note](#)
-

Use Model

To use this feature, perform the following steps:

- ▶ Specify the Native mode using the following setting in the `certitude_config.cer` configuration file:

```
setconfig -Simulator=native
```

- ▶ Specify the `-kdb` option in the `certitude_compile` configuration file.

In the two-step flow, specify the `-kdb` option in the command line as follows:

```
#!/bin/sh -e
# VCS compile script
% vcs -kdb -sverilog tb_top.sv dut_top.sv dut_bot.sv -
debug_access+r
```

In the UUM flow, specify the `-kdb` option in all the `vcs/vlogan/vhdlan` command lines as follows:

```
#!/bin/sh -e
# VCS compile script
% vlogan -kdb -sverilog tb_top.sv dut_top.sv dut_bot.sv
% vcs -kdb -debug_access+r top
```

- ▶ Leave the `VerdiInitCommand` configuration option as empty (default value).
-

Points to Note

The following points must be noted for using this feature:

- During the model phase, the `certitude_compile` file is executed once and information of the KDB design is collected. The KDB design is then automatically loaded when the Verdi system is launched by Certitude. The KDB design cannot be loaded automatically if the model phase has not been executed. The feature is effective only after the model phase.

- If the design is not loaded automatically in the Verdi system, it may be due to one of the following reasons:
 - The `-kdb` option is not applied correctly in the `certitude_compile` file.
 - The `-kdb` option is applied but the KDB design is not compiled and generated correctly.
 - The `VerdiInitCommand` configuration option is set by the user, and Certitude applies the user setting.

Dumping and Comparing Waveforms in Verdi for SystemC Designs

With the integration of Certitude, VCS, Verdi, and CBug, the following benefits are available with this seamless integration:

- Dump the waveform for SystemC designs run on a specific testcase with or without an injected fault.
- Compare the reference waveform with the faulty waveform for SystemC designs.
- Generate Runtime Information Database (RIDB) for loading SystemC designs in Verdi.

This section consists of the following subsections:

- [Use Model](#)
- [Point to Note](#)

Use Model

To use this feature, perform the following steps:

- Specify VCS as the simulator using the following setting in the `certitude_config.cer` configuration file:

```
setconfig -Simulator=vcs
```

In the `certitude_compile` configuration file, compile the design using VCS. For example,

```
#!/bin/sh -e
# VCS compile script
syscan ${CER_SCAN_OPTIONS} $SRC/top.cpp
vcs -sysc sc_main ${CER_VCS_SC_OPTIONS}
```

- ▶ Set the `WaveUseEmbeddedDumper` configuration option to `true` in the `certitude_config.cer` configuration file to use the embedded dumper for dumping waveforms:

```
setconfig -WaveUseEmbeddedDumper=true
```

- ▶ Invoke Certitude and execute commands for the model, activation, and detection phases.

```
>certitude
cer> model
cer> activate
cer> detect
```

- ▶ Execute the `dumpwaves` and `verdiwavedebug` commands accordingly to dump and compare waveforms.

For example,

```
cer> dumpwaves -fault=10 -testcaselist=fir_rtl
cer> verdiwavedebug -fault=10 - testcase=fir_rtl
```

Note:

For more details on dump and compare waveforms with Certitude, see the *Certitude User Manual*.

- ▶ Generate an RIDB file with the original source code and load the design automatically in Verdi with the `verdistart` or the `verdisourcedebug` command.

For example,

```
verdidumpridb -testcase=fir_rtl
```

Point to Note

Simulation executed by the `dumpwaves` command is killed if simulation CPU timeout is reached. However, simulation is not killed if the `dumpwaves` command is executed immediately after executing the `model` command.

Reducing Compilation Time in Native Certitude With VCS Partition Compile Flow

This feature is Limited Customer Availability (LCA). Limited Customer Availability (LCA) features are features available with select functionality. These features will be ready for a general release based on customer feedback and meeting the required feature completion criteria. LCA features do not need any additional license keys.

With the integration of Native Certitude with VCS Partition Compile flow, you can improve the turnaround time for successive compilations in Native Certitude.

Partition Compile is used primarily to improve the turnaround time in successive compilations. In Native Certitude, you have various phases of compilation. Therefore, when you integrate Native Certitude along with the Partition Compile flow, you are able to reduce the compilation time at each step. This results in improved performance during compilation time.

Use Model

The Native Certitude use model remains the same as that of the previous use model. For information about the use model, see the *Native Integration of Certitude and VCS* section in the *Certitude User Manual*.

Native Certitude consists of three phases in the following order:

- Model
- Activation
- Detection

The three different phases always run in the same order.

The model phase and the activation phase occur in a single compilation. The model phase analyzes the DUT and creates a list of faults. The activation phase generates the `simv` executable, which is used to determine the activated faults. The detection phase performs the new compilation that generates the `simv` executable to determine the propagation and the detection status of the faults. All the phases target the DUT part of the design and they do not affect the testbench.

If you want to fine-tune the testbench and recompile the design at activation or detection phase, you must recompile the entire design including the DUT.

Similarly, after you are completed with the model and activation phases compilation and you want to perform compilation in the detection phase, you must recompile the entire design including the testbench that has not changed.

By integrating with Partition Compile, you can avoid the recompilation of unchanged testbench part of the design. The compilation of testbench partition takes place only when the testbench is changed.

Example

Consider the following test cases:

```
//topcfg.v
config topcfg;
    design top;
    partition instance top.dut use dut;
endconfig

//top.v
`noinline
module top;
    dut dut();
    Test test();
endmodule

module dut;
    reg clk;
    reg data;
    initial begin
        clk=1'b0;
        #25
        $finish();
    end

    always
    #7 clk = ~clk;
    Core core1(clk,data);
endmodule

module Core(input clk,output data);
    reg data;
    always @ (posedge clk)
        data = ~data;
    initial begin
        $display("%m");
    end
    sub sub_core(data);
endmodule

module Test;
    initial begin
        #4 $display("%m");
    end
endmodule

module sub(input data);
    initial begin
        $display("Data: %d",data);
    end
endmodule
```

The following is the VCS compilation script that is located in the `certitude_compile` configuration file:

```
% vlogan -sverilog ./topcfg.v ./top.v  
% vcs topcfg -partcomp -partcomp_dir=".//partitionlib"
```

Invoke Certitude and execute commands for the model, activation and detection phases as follows:

- ▶ Run model using the following command:

```
cer> model
```

This command compiles all partitions.

- ▶ Run activate using the following command:

```
cer> activate -compile
```

This command compiles only the DUT partition.

- ▶ Run detect using the following command:

```
cer> detect -compile
```

This command recompiles only the DUT partition.

Note:

You can use all default partition compile options along with the mentioned limitation.

Limitation

The feature has the following limitation:

- The DUT must be present entirely in a single partition.

32

Integrating VCS with Vera

Vera® is a comprehensive testbench automation solution for module, block and full system verification. The Vera testbench automation system is based on the OpenVera™ language. This is an intuitive, high-level, object-oriented programming language developed specifically to meet the unique requirements of functional verification.

You can use Vera with VCS to simulate your testbench and design. This chapter describes the required environment settings and usage model to integrate Vera with VCS.

Setting Up Vera and VCS

To use Vera, you must set the Vera environment as shown below:

```
% setenv VERA_HOME Vera_Installation
% setenv PATH $VERA_HOME/bin:$PATH
% setenv LM_LICENSE_FILE license_path:$LM_LICENSE_FILE
or
% setenv SNPSLMD_LICENSE_FILE license_path:$SNPSLMD_LICENSE_FILE
```

Note:

If you set the `SNPSLMD_LICENSE_FILE` environment variable, then VCS ignores the `LM_LICENSE_FILE` environment variable.

Set the VCS environment as shown below:

```
% setenv VCS_HOME VCS_Installation
% setenv PATH $VCS_HOME/bin:$PATH
% setenv LM_LICENSE_FILE license_path:$LM_LICENSE_FILE
or
% setenv SNPSLMD_LICENSE_FILE license_path:$SNPSLMD_LICENSE_FILE
```

Note:

If you set the `SNPSLMD_LICENSE_FILE` environment variable, then VCS ignores the `LM_LICENSE_FILE` environment variable.

For more information on VCS installation, see “Setting Up the Simulator” .

Using Vera with VCS

The usage model to use Vera with VCS includes the following steps:

- Compile your OpenVera code using Vera

This will generate a `.vro` file and a `filename_vshell.v` file. The `filename_vshell.v` is a Verilog file.

The following table lists the Vera options to generate a shell file based on your design topology:

Table 65 Vera Options

Option	Description
<code>-vlog</code>	Generates a Verilog shell file, <code>filename_vshell.v</code> . Use this option if your design is a Verilog-only design.
<code>-sro</code>	Generates a VHDL shell file, <code>filename_vshell.vhd</code> . Use this if your design is a VHDL-only design.
<code>-sro_mx</code>	Generates a VHDL shell file, <code>filename_vshell.vhd</code> . Use this if your design top is in VHDL.
<code>-vcs_mx</code>	Generates a Verilog shell file, <code>filename.vshell</code> . Use this if your design top is in Verilog.

- Analyze all Verilog files including the vshell file generated in the above step.
- Analyze all VHDL files.
- Compile/elaborate your design and the `filename_vshell.v` file using the `-vera` option. This option is required to use Vera with VCS.
- Simulate the design by specifying the `.vro` file created in the first step using the `+vera_load` runtime option. You can also specify this `.vro` file in the `vera.ini` file in your working directory as shown in the following example:

```
vera_load = tb_top.vro
```

See the *Vera User Guide* for more information.

Usage Model

Use the following usage model to compile OpenVera code using Vera:

```
% vera -cmp [Vera_options] OpenVera_files
```

See the *Vera User Guide* for a list of Vera compilation options.

Two-Step Flow

Compilation

```
% vcs [compile_options] -vera verilog_filelist filename_vshell.v
```

Simulation

```
% simv [simv_options] +vera_load=file.vro
```

Three-Step Flow

Analysis

```
% vlogan [vlogan_options] Verilog_files filename.vshell  
% vhdlan [vhdlan_options] VHDL_files
```

Elaboration

```
% vcs [elab_options] -vera top_entity/module/config filename_vshell.v
```

Simulation

```
% simv [simv_options] +vera_load=file.vro
```

33

VCS Mixed-Signal Simulation

This chapter describes Synopsys CustomSim™ simulator and VCS mixed-signal mixed-HDL environment setup and usage model for better understanding. Supported analog simulators also include HSIM and FineSim. You can use any one of the simulators to do mixed-signal simulations.

Before reading the subsequent topics in these sections, you must be familiar with the:

- SPICE, Verilog, and VHDL languages
- CustomSim and VCS usage

This section consists of the following topics:

- [Introduction to VCS and CustomSim](#)
- [Setting up the Environment](#)
- [Scheduling Analog-to-Digital Events in the NBA Region](#)
- [Support of Verilog Force and Release Assignments on Wreal Nets](#)
- [Support for Wreal Nets in Verilog-AMS Flow](#)
- [Support for SystemC Designs in Verilog-AMS](#)
- [Support for Wildcard Character and the -exclude Option in the Mixed Signal Control Command](#)
- [Enhancement to the force Command to Force SPICE Bus Ports](#)
- [VCS with AMS Integration](#)
- [Support for Predefined Nettypes in SystemVerilog-SPICE Flow](#)
- [Support for SystemVerilog Nettypes in Mixed-Signal Simulation](#)
- [Using SystemVerilog Nettypes in Mixed-Signal Simulation](#)
- [Support for SystemVerilog Packed Array at Mixed Design Boundary](#)
- [Support for Unmapped User-Defined Nettypes in SystemVerilog-SPICE Flow](#)
- [Support for User-Defined Nettypes in Mixed-Signal Simulation](#)

- Enhancements in Type Coercion and Converter Insertions
- Enhancements to Event Functions in SV-SPICE Flow
- Support for UPF Power-Aware Interface Elements in Mixed Signal Simulation Environment
- Real Number Modeling With VIZ Nettypes
- Connectivity Technologies in VCS
- Support for Wreal Nets
- Support for Converting Wreal to Nettype
- Support for Detail Discipline Resolution of Mixed Signals

For more information about CustomSim, see the Discovery AMS: *Mixed-Signal Simulation User Guide*. For more information about CustomSim HSIM, see the CustomSim HSIM documentation. For more information about CutomSim FineSim, see the *FineSim User Guide: Pro and SPICE Reference*.

Introduction to VCS and CustomSim

The VCS and CustomSim cosimulation feature provides mixed-signal mixed-HDL verification solution. This feature enables you to simulate a design, which is described in SPICE (or other transistor-level description language that FineSim supports), Verilog-HDL (Verilog), and VHDL.

VCS and CustomSim cosimulation supports:

- Verilog-top, VHDL-top, and SPICE-top netlist configurations.
- Verilog and VHDL as digital modeling languages.
- Donut design configuration, which is the interleaved instantiations of SPICE subcircuits and Verilog or VHDL digital cells in the design hierarchy.
- The use of cell-based partitioning.

In the VCS and CustomSim cosimulation flow, if a SPICE cell is instantiated under a VHDL block, a dummy Verilog wrapper is required for the instantiated SPICE cell. For successful SPICE instantiation, this wrapper file must be analyzed like any other Verilog file.

Note:

CustomSim and VCS cosimulation uses Direct Kernel Interface to exchange data between CustomSim and VCS.

The VCS and CustomSim mixed-signal simulation process involves the following three phases:

1. [Analyzing a Design](#)
 2. [Elaborating a Design](#)
 3. [Running the Simulation](#)
-

Analyzing a Design

During design analysis, the syntax of Verilog and VHDL files is verified and intermediate files are generated. The generated intermediary files are later used during the elaboration phase. Any syntax errors in Verilog or VHDL netlists are flagged at this phase.

Elaborating a Design

During elaboration, the design hierarchy is built based on the information obtained from the analysis phase. In this phase, incorrect port connectivity or missing definitions for instantiated blocks in Verilog, VHDL, or SPICE are identified and flagged, if they exist. If no error is encountered, at the end of the Elaboration phase, the binary executable is generated.

Running the Simulation

To start the mixed-signal simulation, run the executable generated during the elaboration phase.

A working installation of VCS and a matching version of CustomSim are required to run VCS and CustomSim mixed-signal mixed-HDL simulation that can be found at:<https://solvnetplus.synopsys.com/retrieve/1463626.html>

Setting up the Environment

You must set the following environment variables, before running the VCS and CustomSim simulation:

Licenses

Either `LM_LICENSE_FILE` or `SNPSLMD_LICENSE_FILE` can be used to specify the license file location:

```
% setenv LM_LICENSE_FILE license_file_path
```

or

```
% setenv SNPSLMD_LICENSE_FILE license_file_path
```

Required UNIX Paths and Variable Settings

To set the paths for CustomSim and VCS, do the following:

For CustomSim

```
% source XA_install_directory/CSHRC_xa
```

For CustomSim HSIM

```
% setenv VCS_HOME VCS_Installation
% setenv HSIM_HOME HSIM_Installation
% setenv HSIM_64 1
```

Unset the variable `HSIM_64`, if you are using in 32-bit mode.

Note:

If you set the `SNPSLMD_LICENSE_FILE` environment variable, then VCS ignores the `LM_LICENSE_FILE` environment variable.

For CustomSim FineSim

```
% source FineSim_install_directory/finesim.cshrc
```

For VCS

```
% setenv VCS_HOME VCS_install_directory
% set path = ($VCS_HOME/bin $path)
```

Use Model

Using VCS and CustomSim involves the following three phases:

1. Analysis

Execute the following during analysis:

```
% vlogan [vlogan_options] Verilog_files
% vhdlan [vhdlan_options] VHDL_files
```

2. Elaboration

Execute the following during elaboration:

```
% vcs -ad=initFile [elab_options] top_entity/module/config
```

3. Simulation

To enable mixed-signal simulation, use the `-ad=initFile` elaboration option. If you use `-ad` without specifying the `initFile`, VCS assumes the mixed-signal setup filename as `vcsAD.init`.

```
% simv [simv_options]
```

Example

The following example shows a sample compilation script that contain commands to analyze and elaborate a design with VHDL, Verilog, and SPICE components. In this example, the files `tb.vhd` and `blk_1.vhd` contain the VHDL netlist, files `blk_2.v` and `blk_3.v` contain the Verilog netlist, and the file `all_spice.spi` contains the SPICE netlist.

Example:

```
% vlogan blk_2.v blk_3.v
% vhdlan tb.vhd blk1.vhd
% vcs -ad=setup.init testbench
% simv
```

In this example, `testbench` is the name of the top-level entity. The mixed-signal setup file `setup.init` is shown in the following code:

For CustomSim

```
choose xa -n all_spice.spi;
use_spice -cell counter ddr_flop;
set bus_format <%d>;
```

Limitation

If a testbench contains an array of instances, SPICE substitution is not supported for the elements of the array.

For example, in the following command, if `top.dut5[1]` is an instance of the array, `addr2`, SPICE substitution does not occur for the `top.dut5[1]` instance.

```
use_spice -cell addr2 -inst top.dut5[1];
```

For CustomSim HSIM

```
choose hsim all_spice.spi;
use_spice -cell counter ddr_flop;
set bus_format <%d>;
```

For CustomSim FineSim

```
choose finesim -n all_spice.spi;
use_spice -cell counter ddr_flop;
set bus_format <%d>;
```

Where, `counter` and `ddr_flop` are the names of the multi-view cells. The SPICE view of these cells are used in this design.

Scheduling Analog-to-Digital Events in the NBA Region

VCS enables you to schedule a2d events in the nonblocking assignment (NBA) region. With this feature you will have more flexibility on data latching and more control to handle data race conditions thereby bringing in more predictability in mixed signal designs.

Use Model

To schedule all a2d events on a given SPICE node in the NBA region, add the `queue=blocking|nonblocking` option to the a2d command in the mixed signal setup file (`vcsAD.init`).

```
a2d node=<spice_node> queue=nonblocking;
```

The default value is `blocking` and all a2d events on `<spice_node>` to the digital field are scheduled in the active region. When the value is `nonblocking`, the a2d events on the given node are scheduled in the NBA region.

Note:

If a SPICE node for which `queue=nonblocking` is assigned turns to be a digital-to-digital (d2d) node, then this feature will have no effect on the SPICE node.

Support of Verilog Force and Release Assignments on Wreal Nets

The force and release on wreal nets allows you to force and release wreal nets from the testbench code written in UCLI, MHPI, or VPI and allows you to use `$hdl_xmr_force/$hdl_xmr_release` system tasks. It is useful when there are large number of designs and instances.

It does not support deposit functionality. It generates an error message if you try to deposit values on wreal nets. This applies to all sources of deposit including:

- `$deposit`
- `UCLI force -deposit`

- \$hdl_xmr
 - vpi_put_value **without** vpiForceFlag
-

Usage Example

The following example supports usage of force and release assignments on wreal nets:

Example 242 top.v

```
module top;
  mid m();
endmodule

module mid;
  wreal wr[1:2];
  bot b();
  always @(wr[1]) $display ("wr[1]=%g at %0t", wr[1], $time);
endmodule

module bot;
  real r;
  assign top.m.wr[1] = r;

  initial begin
    r = 0.1;
    repeat (5) #10 r = r * 2.0;
    $finish;
  end
endmodule
```

Example 243 force.tcl

```
catch {force {top.m.wr[1]} 1.0 -deposit} err
puts $err
force {top.m.wr[1]} 8.0
run 1
get {top.m.wr[1]}
release {top.m.wr[1]}
run 3
get {top.m.wr[1]}
quit
```

Command line:

```
% vcs top.v -sverilog -wreal res_sum -debug_access+all
% ./simv -ucli -i force.tcl
```

The following output is generated:

```
ucli% catch {force {top.m.wr[1]} 1.0 -deposit} err
1
```

```

ucli% puts $err
Error-[UCLI-FORCE-ERR-WREAL] NYI force on WREAL
  Force command failed on object 'top.m.wr[1]' with type 'vpiRealWire'.
  Command force deposit on wreal net is not yet supported.
  Please refer to UCLI user guide for more documentation on force
  commands.
ucli% force {top.m.wr[1]} 8.0
ucli% run 1
ucli% get {top.m.wr[1]}
8.000000
ucli% release {top.m.wr[1]}
ucli% run 3
ucli% get {top.m.wr[1]}
0.100000
ucli% quit

```

Limitations

The feature has the following limitations:

- The `$deposit` system task on the wreal net is not supported.
- The force/release on wreal arrays that are present in modules instantiated in VHDL is not supported.
- The force/release on wreal arrays are not supported when wreal scalarization is disabled using `-wreal noscalarize` option.

Support for Wreal Nets in Verilog-AMS Flow

Connect modules are inserted based on the specified connect rules. Connect rules are created to control which connect modules must be used and where they must be inserted. The connect modules are also searched as per the connect rules defined and first matched connect module is selected.

To select the connect module, the discipline of the `wreal` net or port should also match along with the `wreal` type.

Usage Example

The following example explains how to select the connect module that matches the discipline of the `wreal` net or port along with the `wreal` type:

```

module bot(output wreal p1);
  myLogical p1;
  assign p1 = 1.0;
  anaMod ANAMOD (p1);
endmodule

```

```
// Analog module
module anaMod (e1);
    output e1;
    electrical e1;
endmodule

// Following connect module has output port with myLogical discipline
// with wreal port type.
connectmodule elect_to_wreal (ain, dout);
    input ain;
    electrical ain;

    output dout;
    wreal dout;
    myLogical dout;
.....
endmodule

// Following connect module has output port with myLogical discipline
// with wire type.
connectmodule elect_to_logic (ain, dout);
    input ain;
    electrical ain;

    output dout;
    myLogical dout;
.....
endmodule

connectrules mixedsignal;
connect elect_to_logic input electrical, output myLogical;
connect elect_to_wreal input electrical, output myLogical;
.....
endconnectrules
```

In the above example, a connect module insertion is required inside the `bot` module because `p1` `wreal` signal is connected to the `e1` `electrical` port of the `anaMod` module. To select the connect module, there are two connect rules that refer to two different connect modules, that is, `elect_to_logic` and `elect_to_wreal` respectively.

The `elect_to_wreal` connect module has digital output port of type `wreal` with `myLogical` discipline. Therefore, the `elect_to_wreal` connect module is selected as per the connect module selection algorithm.

Support for SystemC Designs in Verilog-AMS

Verilog-AMS is supported with SystemC in the design topologies using the same option – `sysc=ams`.

Note:

The `-ams` option must be provided along with the `-sysc=ams` option.

Use Model

This section provides the use model to support Verilog-AMS and SPICE with SystemC designs.

The other options of VCS elaboration for mixed signal simulations remain the same.

If the `-sysc=ams` option is not used for Verilog-AMS elaboration, then an error message is generated that mentions Verilog-AMS is not supported.

Usage Example

The following example illustrates the usage of SystemC designs in Verilog-AMS and SPICE:

```
test.v
`include "connectmodules.vams"
`timescale 1ns/1ps

// Function to check values of 1, 0, x or z
function check_10xz (input logic orig_logic, input logic ref_logic, input
integer tag);
    if(orig_logic === ref_logic) $display("Passed - %-d", tag);
    else $display("Failed - %-d : Orig_value = %-d : Ref_value =
%-d", tag, orig_logic, ref_logic);
    return 0;
endfunction

module amp (out, in);
input in; output out;

electrical out, in;
parameter real Gain = 2;

analog
    V(out) <+ Gain * V(in);
endmodule

module testbench();
electrical out,in;
bit sysc_out, clk;

amp a(out,in);

initial clk = 1'b0; initial #1000 $finish();
```

```
always #10 clk = ~clk;
analog begin
    V(in) <+ transition((clk == 1) ? 3.3 : 0.0, 0.1n, 0.1n ,0.1n);
end
```

```
    sysc_inv SYS_INV (out, sysc_out);
```

```
initial begin
    #6 check_10xz(sysc_out, 1'b1, 1);
    #6 check_10xz(sysc_out, 1'b0, 2);
end
endmodule
```

disciplines.vams

```
discipline logical
domain discrete;
enddiscipline

nature Current
    units = "A";
    access = I;
    idt_nature = Charge;
    abstol = 1e-12;
endnature

nature Charge
    units = "coul";
    access = Q;
    ddt_nature = Current;
    abstol = 1e-14;
endnature

nature Voltage
    units = "V";
    access = V;
    idt_nature = Flux;
    abstol = 1e-6;
endnature

nature Flux
    units = "Wb";
    access = Phi;
    ddt_nature = Voltage;
    abstol = 1e-9;
endnature

discipline electrical
    potential Voltage;
    flow Current;
enddiscipline
```

connectmodules.vams

```

`include "disciplines.vams"
`timescale 1ps/1ps

connectmodule elect_to_logic(el, cm);
    input el; electrical el;
    output cm; logical cm;
    reg cm_1;
    real vel;
    assign cm = cm_1;

    always @(above(V(el) - 1.65)) begin
        cm_1 = 1;
    end
    always @(above(1.65 - V(el))) begin
        cm_1 = 0;
    end
endmodule

connectmodule logic_to_elect(l2e_cm,l2e_el);
    input l2e_cm; logical l2e_cm;
    output l2e_el; electrical l2e_el;

    assign l2e_cm = l2e_cm;
    analog begin
        V(l2e_el) <+ transition((l2e_cm == 1) ? 3.3 : 0.0, 0.1n,
        0.1n ,0.1n);
    end
endmodule

connectrules mixedsignal;
    connect elect_to_logic input electrical, output logical;
    connect logic_to_elect input logical, output electrical;
endconnectrules

```

sysc_inv.cpp

```
#include "sysc_inv.h"
```

sysc_inv.h

```

#ifndef _sysc_inv_h_
#define _sysc_inv_h_

#include "systemc.h"

SC_MODULE(sysc_inv) {
    sc_in<bool> ina;
    sc_out<bool> outx;

    SC_CTOR(sysc_inv) : ina("ina"), outx("outx") {
        SC_METHOD(action);
        sensitive << ina;
    }
}

```

```
void action() {
    outx.write(!ina.read());
}
};

#endif

test.init

choose xa
```

To run the example, use the following commands:

```
%> vlogan -sverilog test.v -ams
%> syscan sysc_inv.cpp:sysc_inv
%> vcs -timescale=1ns/1ps -sysc=ams testbench -
ams_discipline logical -ams -ams_iereport -full164 -sverilog
-ad=test.init
%> simv
```

This plugs-in the preverified SystemC models into Verilog-AMS simulation.

Support for Wildcard Character and the -exclude Option in the Mixed Signal Control Command

The `-exclude` option is supported in the `use_verilog` mixed signal control command, which can be used to provide the instances that must not be partitioned. Also, the "*" wildcard character is supported in the hierarchical path of the `-inst` and `-exclude` options in the `use_verilog` mixed signal control command. This feature is useful when there are large number of instances in the design.

Use Model

The syntax of the `-exclude` option with the wildcard character is as follows:

```
use_verilog -module inverter -inst top.*.* -exclude top.a0.*;
```

Note:

The "*" wildcard character in the instance name is only supported if the `-adopt wildcard` option is added to the VCS command line as follows:

```
% vcs -adopt wildcard ...
```

For example:

```
% vcs -sverilog example.v -ad=example.init -adopt wildcard -full164
```

Note:

The use of the `-adopt wildcard` option and the "*" wildcard character in the `use_verilog` command increases (sometimes almost doubles) the compilation time.

Usage Example

The following example illustrates usage of `-exclude` option in the `use_verilog` command.

*Example 244 Example of usage of -exclude option in the use_verilog command**Example.spi*

```
.subckt inv i zn
xi0 i zn inverter
.ends

.subckt inverter in out
mn1 out in gnd gnd n w=1u l=1u
mp1 out in vdd vdd p w=4u l=1u
.ends
```

Example.init

```
choose xa -n example.spi;

use_spice -cell inv;
use_verilog -module inverter -inst top.*.* -exclude top.a0.i0;
resolve_x_inst_prefix enable;
```

Limitation

The feature has the following limitation:

- Wildcard character is restricted to only one layer of design hierarchy. For example if `top.*.b1` design is searched in the following design hierarchies `top.a1.b1` `top.p0.a1.b1` then the wildcard character is applied only to the design `top.a1.b1`.

Enhancement to the force Command to Force SPICE Bus Ports

As mixed signal designs are becoming more prevalent, the requirement is for a single flow that works independently if the individual blocks used in the RTL designs are analog blocks or digital blocks.

The `force` command is extended to force SPICE bus ports. It allows you to force SPICE buses from UCLI/\$hdl_xmr_force/HDL_XMR_FORCE using the same command as the digital vectors. You can only force vectors (bus) of SPICE ports.

This feature allows you to use the same flow for both analog and digital blocks, and these blocks are interchangeable. You can force SPICE circuit ports to a given value using the UCLI/\$hdl_xmr_force/HDL_XMR_FORCE force command.

Following are the various sets of force/release pairs that can be used:

- [UCLI force/release](#)
- [Verilog force/release Procedural Statements](#)
- [\\$hdl_xmr_force/\\$hdl_xmr_release Verilog System Tasks](#)
- [HDL_XMR_FORCE/HDL_XMR_RELEASE VHDL Sub-Programs](#)

Note:

The requirements that are applicable to the `force` command are applicable to its corresponding `release` command.

Terminology

When showing examples of hierarchical paths, the following conventions are used:

() – Indicates a digital block.

<> – Indicates an analog block.

For example,

`(top).<sp1>.<sp2>.a` is an analog net 'a' in sub-circuit sp2.

A bus refers to a set of logic data types (reg, logic, and wire in Verilog; bit and std_logic in VHDL). A bus might have different formats in SPICE. This format must be defined in the CustomSim initialization file (using the `bus_format` command). The <> format is used in this document to indicate an analog block. However, it is implicit that any other format specified by the `bus_format` command is recognized.

Inclusions in the MSV Control File

To enable this feature, specify `form_spice_bus enable` in the Mixed Signal Verification (MSV) control file. Use the `-ad=<initFile>` elaboration option on the `vcs` command line, to enable the MSV control file. If you use the `-ad` option without specifying the control file, VCS assumes the mixed-signal setup file name as `vcsAD.init`.

All the ports are treated as scalar if the bus format is not defined. To avoid this ambiguity, the bus format must be defined in the control file. The `bus_format` must contain a special character, such as `<%d>`. This format is applicable for all sub-circuits in a SPICE hierarchy apart from the boundary.

For example, include the following highlighted lines in the MSV control file:

```
choose xa -n sptop.spi -c sptop.cfg;
....
spice_top;
form_spice_bus enable;
bus_format <%d>;
resolve_x_inst_prefix enable;
```

Note:

An assignment is always made from right to left and is independent of the direction of the array declaration. The direction of the bus is based on the declaration order in SPICE.

Examples

As a prerequisite to run the following examples, include the parameter (.mod) file and the MSV configuration (.cfg) file.

UCLI force/release

sptop_1.init

```
choose xa -n sptop_1.spi -c sptop_1.cfg;
use_spice -cell sp1;
form_spice_bus enable;
bus_format <%d>;
```

sptop_1.spi

```
* SUBCKT inverter
.inc 'sptop_1.mod'

.subckt sp1 sp_in<2> sp_in<1> sp_out<1> sp_out<2>
mn1 sp_out<1> sp_in<1> gnd gnd n w=1.00u l=0.35u
mp1 sp_out<1> sp_in<1> vdd vdd p w=2.00u l=0.35u

mn2 sp_out<2> sp_in<2> gnd gnd n w=1.00u l=0.35u
mp2 sp_out<2> sp_in<2> vdd vdd p w=2.00u l=0.35u
xvmon sp_in<1> sp_in<2> sp_out<1> sp_out<2> vmon
.ends

.global vdd gnd
vsu vdd gnd 3.3
.end
```

sptop_1.v

```
`timescale 1ns/10ps
`define TOLERANCE 0.0001
```

```

function cmpr_logic_nums (input logic orig_logic, input logic
    ref_logic_num, input integer tag);
    logic diff;
    diff = orig_logic - ref_logic_num;
    if(diff <= 0) diff = -diff;
    if(diff < `TOLERANCE) $display("      Passed - %-d", tag);
    else $display("      Failed - %-d orig_val = %b, ref_val=%b",
    tag, orig_logic, ref_logic_num);
    return 0;
endfunction

module sv_top ();
    logic [1:2] sp_in, sp_out;

    initial begin
        sp_in[1]=1'b1;
        sp_in[2]=1'b1;
    end

    sp1 SP1 (sp_in, sp_out);

endmodule

module vmon (input logic [1:2] in, input logic [1:2] out);
    initial begin
        #1 cmpr_logic_nums(out[2], 0, 1);
        #10 cmpr_logic_nums(out[2], 0, 2);
        #10 cmpr_logic_nums(out[2], 1, 3);
        #10 cmpr_logic_nums(out[2], 1, 4);
        #10 cmpr_logic_nums(out[2], 0, 5);
    end
endmodule

module sp1 (input logic [1:2] sp_in, output logic [1:2] sp_out);
endmodule

sstop_1.ucli
run 2000
force {sv_top.SP1.sp_in[2:1]} 2'b00
run 2000
release {sv_top.SP1.sp_in[1:2]}
run
quit

```

Command Lines

```
% vcs sstop_1.v -sverilog -full64 -q -ad=sstop_1.init -debug_access+f
% simv -ucli -i sstop_1.ucli
```

Verilog force/release Procedural Statements

sstop_2.init

```
choose xa -n sstop_2.spi -c sstop_2.cfg;
use_spice -cell sp1;
form_spice_bus enable;
bus_format <%d>;
```

sstop_2.spi

```
* SUBCKT inverter
.inc 'sstop_2.mod'

.subckt sp1  sp_in<1> sp_in<2> sp_out<1> sp_out<2>
mn1 sp_out<1> sp_in<1> gnd gnd n w=1.00u l=0.35u
mp1 sp_out<1> sp_in<1> vdd vdd p w=2.00u l=0.35u

mn2 sp_out<2> sp_in<2> gnd gnd n w=1.00u l=0.35u
mp2 sp_out<2> sp_in<2> vdd vdd p w=2.00u l=0.35u
xvmon sp_in<1> sp_in<2> sp_out<1> sp_out<2> vmon
.ends

.global vdd gnd
vsu vdd gnd 3.3
.end
```

sstop_2.v

```
`timescale 1ns/10ps

`define TOLERANCE 0.0001

function cmpr_logic_nums (input logic orig_logic, input logic
  ref_logic_num, input integer tag);
  logic diff;
  diff = orig_logic - ref_logic_num;
  if(diff <= 0) diff = -diff;
  if(diff < `TOLERANCE) $display("    Passed - %-d", tag);
  else $display("    Failed - %-d orig_val = %b, ref_val=%b",
    tag, orig_logic, ref_logic_num);
  return 0;
endfunction

module sv_top ();
  logic [1:2] sp_in, sp_out;

  initial begin
    sp_in[1]=1'b1;
    sp_in[2]=1'b1;
  end

  sp1 SP1 (sp_in, sp_out);
```

```

    initial begin
#20 force sv_top.SP1.sp_in[1:2]=2'b00;
#20 release sv_top.SP1.sp_in[1:2];
    end

endmodule

module vmon (input logic [1:2] in, input logic [1:2] out);

initial begin
#1 cmpr_logic_nums(out[2], 0, 1);
#10 cmpr_logic_nums(out[2], 0, 2);
#10 cmpr_logic_nums(out[2], 1, 3);
#10 cmpr_logic_nums(out[2], 1, 4);
#10 cmpr_logic_nums(out[2], 0, 5);
    end

endmodule

module sp1 (input logic [1:2] sp_in, output logic [1:2] sp_out);
endmodule

```

Command Lines

```
% vcs sptop_2.v -sverilog -full64 -q -ad=sptop_2.init -debug_access+all \
+vcs+vcpluson
% simv
```

\$hdl_xmr_force/\$hdl_xmr_release Verilog System Tasks

sptop_3.init

```

choose xa -n sptop_3.spi -c sptop_3.cfg;
use_spice -cell sp1;
use_spice -cell sp2;
use_spice -cell sp3;
use_spice -cell sp4;
use_spice -cell sp5;
spice_top;
form_spice_bus enable;
bus_format <%d>;
resolve_x_inst_prefix enable ;

```

sptop_3.spi

```

* SUBCKT inverter
.inc 'sptop_3.mod'

xsp1 sp_in1<2> sp_in1<1> sp_out1<2> sp_out1<1> sp1
xsp2 sp_in2<2> sp_in2<1> sp_out2<2> sp_out2<1> sp2

vsu1 sp_in1<2> gnd 3.3

```

```

vsu2 sp_in1<1> gnd 3.3
vsu3 sp_in2<2> gnd 3.3
vsu4 sp_in2<1> gnd 3.3

.subckt sp1 sp_in<2> sp_in<1> sp_out<2> sp_out<1>
xvhdl sp_in<2> sp_in<1> sp_out<2> sp_out<1> vh_child1
.ends

.subckt sp2 sp_in<2> sp_in<1> sp_out<2> sp_out<1>
xvlog2 sp_in<2> sp_in<1> sp_out<2> sp_out<1> vlog2
.ends

.subckt sp3 sp_in<2> sp_in<1> sp_out<2> sp_out<1>
mn1 sp_out<2> sp_in<2> gnd gnd n w=2.00u l=0.35u
mp1 sp_out<2> sp_in<2> vdd vdd p w=1.00u l=0.35u

mn2 sp_out<1> sp_in<1> gnd gnd n w=2.00u l=0.35u
mp2 sp_out<1> sp_in<1> vdd vdd p w=1.00u l=0.35u
.ends

.subckt sp4 sp_in<2> sp_in<1> sp_out<2> sp_out<1>
xvhdl sp_in<2> sp_in<1> sp_out<2> sp_out<1> vh_child2
.ends

.subckt sp5 sp_in<2> sp_in<1> sp_out<2> sp_out<1>
mn1 sp_out<2> sp_in<2> gnd gnd n w=2.00u l=0.35u
mp1 sp_out<2> sp_in<2> vdd vdd p w=1.00u l=0.35u

mn2 sp_out<1> sp_in<1> gnd gnd n w=2.00u l=0.35u
mp2 sp_out<1> sp_in<1> vdd vdd p w=1.00u l=0.35u
xvmon sp_in<2> sp_in<1> sp_out<2> sp_out<1> vmon
.ends

.subckt inv_spi_2nd in out
mn1 out in gnd gnd n w=2.00u l=0.35u
mp1 out in vdd vdd p w=1.00u l=0.35u
.ends

.global vdd gnd
vsu vdd gnd 3.3
.end

sstop_3.v

`timescale 1ns/10ps

`define TOLERANCE 0.0001

function cmpr_logic_nums (input logic orig_logic, input logic
    ref_logic_num, input integer tag);
    logic diff;
    diff = orig_logic - ref_logic_num;
    if(diff <= 0) diff = -diff;

```

Chapter 33: VCS Mixed-Signal Simulation
 Enhancement to the force Command to Force SPICE Bus Ports

```

        if(diff < `TOLERANCE) $display("      Passed - %-d", tag);
        else $display("      Failed - %-d  orig_val = %b, ref_val=%b",
                      tag, orig_logic, ref_logic_num);
        return 0;
    endfunction

module vlog1 (input logic [2:1] vlog_in, output logic [2:1] vlog_out);

    sp3 SP3 (vlog_in, vlog_out);

    initial begin
#20
$hdl_xmr_force("snps_sptop.sp2.vlog2.SP4.vhd2.vh_inst_of_sp5.sp_in[2:1]"
,"00","0 ns","freeze","",1);
#20
$hdl_xmr_release("snps_sptop.sp2.vlog2.SP4.vhd2.vh_inst_of_sp5.sp_in",
1);
    end
endmodule

module vlog2 (input logic [2:1] vlog_in, output logic [2:1] vlog_out);

    sp4 SP4 (vlog_in, vlog_out);

endmodule

module vmon (input logic [2:1] in, input logic [2:1] out);

    initial begin
#1  cmpr_logic_nums(out[2], 0, 1);
#10 cmpr_logic_nums(out[2], 0, 2);
#10 cmpr_logic_nums(out[2], 1, 3);
#10 cmpr_logic_nums(out[2], 1, 4);
#10 cmpr_logic_nums(out[2], 0, 5);
    end
endmodule

module sp1 (input logic [2:1] sp_in, output logic [2:1] sp_out);
endmodule

module sp2 (input logic [2:1] sp_in, output logic [2:1] sp_out);
endmodule

module sp3 (input logic [2:1] sp_in, output logic [2:1] sp_out);
endmodule

module sp4 (input logic [2:1] sp_in, output logic [2:1] sp_out);
endmodule

module sp5 (input logic [2:1] sp_in, output logic [2:1] sp_out);
endmodule

```

sstop_01_3.vhd

```

library ieee;
use ieee.std_logic_1164.all;
library synopsys;
use synopsys.hdl_xmr_pkg.all;

use std.textio.all;
use ieee.std_logic_textio.all;
entity vh_child1 is
    port ( vlog_in1 : in    std_logic_vector (1 to 2);
           vhd_out  : out   std_logic_vector (1 to 2) );
end;

architecture a of vh_child1 is

component vlog1 is
port ( vlog_in : in std_logic_vector (1 to 2);
       vlog_out : out std_logic_vector (1 to 2));
end component vlog1;

    signal vlog_out1 : std_logic_vector (1 to 2);
begin

    vh_inst_of_vlog1 : vlog1 port map (vlog_in1, vlog_out1);

    vhd_out  <= vlog_out1;

end;

```

sstop_02_3.vhd

```

library ieee;
use ieee.std_logic_1164.all;
use std.textio.all;
use ieee.std_logic_textio.all;
entity vh_child2 is
    port ( sp_in  : in    std_logic_vector (1 to 2);
           vhd_out : out   std_logic_vector (1 to 2) );
end;

architecture a of vh_child2 is

component sp5 is
port ( sp_in : in std_logic_vector (1 to 2);
       sp_out : out std_logic_vector (1 to 2));
end component sp5;

    signal sp_out1 : std_logic_vector (1 to 2) ;
begin

    sp5

```

```

vh_inst_of_sp5 : sp5 port map (sp_in, sp_out1);
    vh_out <= sp_out1;
end;

```

Command Lines

```

% vlogan -q sptop_3.v -sverilog
% vhdlan -q sptop_01_3.vhd sptop_02_3.vhd -nc
% vcs -full64 -q snps_sptop -ad=sptop_3.init -debug_access+f
+vcsvcdpluson
% simv

```

HDL_XMR_FORCE/HDL_XMR_RELEASE VHDL Sub-Programs

sptop_4.init

```

choose xa -n sptop_4.spi -c sptop_4.cfg;
use_spice -cell sp1;
use_spice -cell sp2;
use_spice -cell sp3;
use_spice -cell sp4;
use_spice -cell sp5;
spice_top;
form_spice_bus enable;
bus_format <%d>;
resolve_x_inst_prefix enable ;

```

sptop_4.spi

```

* SUBCKT inverter
.inc 'sptop_4.mod'

xsp1 sp_in1<1> sp_in1<2> sp_out1<1> sp_out1<2> sp1
xsp2 sp_in2<1> sp_in2<2> sp_out2<1> sp_out2<2> sp2
*xsp2 sp_in2 sp_out2 sp2

vsu1 sp_in1<1> gnd 3.3
vsu2 sp_in1<2> gnd 3.3
vsu3 sp_in2<1> gnd 3.3
vsu4 sp_in2<2> gnd 3.3

.subckt sp1 sp_in<1> sp_in<2> sp_out<1> sp_out<2>
xvhdl sp_in<1> sp_in<2> sp_out<1> sp_out<2> vh_child1
.ends

.subckt sp2 sp_in<1> sp_in<2> sp_out<1> sp_out<2>
xvlog2 sp_in<1> sp_in<2> sp_out<1> sp_out<2> vlog2
.ends

.subckt sp3 sp_in<1> sp_in<2> sp_out<1> sp_out<2>
mn1 sp_out<1> sp_in<1> gnd gnd n w=1.00u l=0.35u

```

Chapter 33: VCS Mixed-Signal Simulation
 Enhancement to the force Command to Force SPICE Bus Ports

```

mp1 sp_out<1> sp_in<1> vdd vdd p w=2.00u l=0.35u
mn2 sp_out<2> sp_in<2> gnd gnd n w=1.00u l=0.35u
mp2 sp_out<2> sp_in<2> vdd vdd p w=2.00u l=0.35u
.ends

.subckt sp4 sp_in<1> sp_in<2> sp_out<1> sp_out<2>
xvhd2 sp_in<1> sp_in<2> sp_out<1> sp_out<2> vh_child2
.ends

.subckt sp5 sp_in<1> sp_in<2> sp_out<1> sp_out<2>
mn1 sp_out<1> sp_in<1> gnd gnd n w=1.00u l=0.35u
mp1 sp_out<1> sp_in<1> vdd vdd p w=2.00u l=0.35u

mn2 sp_out<2> sp_in<2> gnd gnd n w=1.00u l=0.35u
mp2 sp_out<2> sp_in<2> vdd vdd p w=2.00u l=0.35u
xvmon sp_in<1> sp_in<2> sp_out<1> sp_out<2> vmon
.ends

.subckt inv_spi_2nd in out
mn1 out in gnd gnd n w=1.00u l=0.35u
mp1 out in vdd vdd p w=2.00u l=0.35u
.ends

.global vdd gnd
vsu vdd gnd 3.3
.end

```

sstop_4.v

```

`timescale 1ns/10ps

`define TOLERANCE 0.0001

function cmpr_logic_nums (input logic orig_logic, input logic
    ref_logic_num, input integer tag);
    logic diff;
    diff = orig_logic - ref_logic_num;
    if(diff <= 0) diff = -diff;
    if(diff < `TOLERANCE) $display("      Passed - %-d", tag);
    else $display("      Failed - %-d orig_val = %b, ref_val=%b",
        tag, orig_logic, ref_logic_num);
    return 0;
endfunction

module vlog1 (input logic [1:2] vlog_in, output logic [1:2] vlog_out);

    sp3 SP3 (vlog_in, vlog_out);

endmodule

module vlog2 (input logic [1:2] vlog_in, output logic [1:2] vlog_out);

```

```

        sp4 SP4 (vlog_in, vlog_out);

endmodule

module vmon (input logic [1:2] in, input logic [1:2] out);

initial begin
    #1 cmpr_logic_nums(out[2], 0, 1);
    #10 cmpr_logic_nums(out[2], 0, 2);
    #10 cmpr_logic_nums(out[2], 1, 3);
    #10 cmpr_logic_nums(out[2], 1, 4);
    #10 cmpr_logic_nums(out[2], 0, 5);
end

endmodule

module sp1 (input logic [1:2] sp_in, output logic [1:2] sp_out);
endmodule

module sp2 (input logic [1:2] sp_in, output logic [1:2] sp_out);
endmodule

module sp3 (input logic [1:2] sp_in, output logic [1:2] sp_out);
endmodule

module sp4 (input logic [1:2] sp_in, output logic [1:2] sp_out);
endmodule

module sp5 (input logic [1:2] sp_in, output logic [1:2] sp_out);
endmodule

sstop_01_4.vhd

library ieee;
use ieee.std_logic_1164.all;
library synopsys;
use synopsys.HDL_XMR_PKG.all;

use std.textio.all;
use ieee.std_logic_textio.all;
entity vh_child1 is
    port ( vlog_in1 : in    std_logic_vector (1 to 2);
           vhd_out  : out   std_logic_vector (1 to 2) );
end;

architecture a of vh_child1 is

component vlog1 is
port ( vlog_in : in std_logic_vector (1 to 2);
       vlog_out : out std_logic_vector (1 to 2));
end component vlog1;

```

```

        signal vlog_out1 : std_logic_vector (1 to 2);
begin

    vh_inst_of_vlog1 : vlog1 port map (vlog_in1, vlog_out1);

    vhd_out <= vlog_out1;

    xmr: process      begin
        wait for 20 ns;
    HDL_XMR_FORCE("snps_sptop.sp2.vlog2.SP4.vhd2.vh_inst_of_sp5.sp_in[1:2]",
    "00","0 ns","freeze","",1);
        wait for 20 ns;
    HDL_XMR_RELEASE("snps_sptop.sp2.vlog2.SP4.vhd2.vh_inst_of_sp5.sp_in", 1);
        wait;
    end process;

end;

sptop_02_4.vhd

library ieee;
use ieee.std_logic_1164.all;
use std.textio.all;
use ieee.std_logic_textio.all;
entity vh_child2 is
    port ( sp_in : in    std_logic_vector (1 to 2);
           vhd_out : out   std_logic_vector (1 to 2) );
end;

architecture a of vh_child2 is

component sp5 is
port ( sp_in : in std_logic_vector (1 to 2);
       sp_out : out std_logic_vector (1 to 2));
end component sp5;

    signal sp_out1 : std_logic_vector (1 to 2) ;
begin

    vh_inst_of_sp5 : sp5 port map (sp_in, sp_out1);

    vhd_out <= sp_out1;

end;

```

Command Lines:

```

% vlogan -q sptop_4.v -sverilog
% vhdlan -q sptop_01_4.vhd sptop_02_4.vhd -nc
% vcs -full64 -q snps_sptop -ad=sptop_4.init -debug_access+all
+vcsvcdpluson
% simv

```

Bit-Select/Part-Select

Partial forcing of buses is available. You can force a bit-select or a part-select of a bus.

For example,

```
// Ex: The bus dimension for top.dut.g1.a is [3:0]
// Following bit-select/part-select can be forced.
force { top.dut.g1.a[0] } 1'b0
force { top.dut.g1.a[2:1] } 1'b10
```

The assignment is always made from right to left and is independent of the direction of the array declaration.

For example,

```
force { top.dut.g1.a[0:1] } 2'b01 => a[0]=0, a[1]=1
force { top.dut.g1.a[1:0] } 2'b01 => a[0]=1, a[1]=0
```

Limitations

This feature does not support the following:

- Multidimensional arrays (MDAs).
- Array of real and nettype. Only logic types are supported.
- Unpacked array.
- Forcing “bundles” of analog nodes - that is, nodes with different names grouped together to form a bus.
- Forcing analog bus that is not a sub-circuit port.

VCS with AMS Integration

This section explains the integration of VCS with Analog Mixed Signal (AMS), how to setup the environment to achieve the integration, and the usage model for VCS-AMS mixed-signal simulations. It contains the following sections:

- [Integrating VCS With AMS](#)
- [Setting up the Environment](#)
- [Use Model](#)
- [Limitations in VCS-AMS Flow](#)

Integrating VCS With AMS

VCS-AMS allows a mixed-signal solution, which enables simulating a design that is modeled in both analog and digital.

Before starting a mixed-signal simulation, it is recommended that both analog and digital parts of the design are individually tested and should be error free. For more information, see the *Discovery AMS: Mixed-Signal Simulation User Guide*.

VCS-AMS mixed-signal simulation supports SPICE or Verilog-AMS top with VHDL down.

Setting up the Environment

For more information about how to set up the environment, see the VCS® > *Integrating VCS Mixed-Signal Simulation* chapter.

Use Model

The use model for the VCS AMS flow conforms to the VCS three-step flow. The steps in this use model are described in the following sections.

Analyzing all VHDL, Verilog, Verilog-AMS Files

Use the following commands to analyze all VHDL and Verilog or Verilog-AMS files into logical libraries:

```
% vlogan -ams <amsFiles> [-work <AMS_LIB>] \
[<otherParseOptions>]
% vlogan -sverilog <svFiles> [-work <VLOG_LIB>] \
[<otherParseOptions>]
% vhdlan <vhdlFiles> [-work <VHDL_LIB>] \
[<otherParseOptions>]
```

If you analyze two different Verilog-AMS disciplines with the same name into the same library, then the last one overwrites the previous definition. This is also applicable for natures, connectrules, and connectmodules.

Elaborating the Design

Use the following command to elaborate the design using the top module, entity, or configuration:

```
% vcs -ad=<setupFile> <topDesignUnit> -ams -ams_crude \
[library.]<ruleID> [-ams_discipline <discreteDiscipline>] \
[-ams_iereport] [<otherElabOptions>]
```

The command options and descriptions are listed in the following :

Option	Description
-ad=< <i>mixed_signal_control_file</i> >	VCS compiles the design file(s). If the file is not mentioned, then it reads <i>vcsAD.init</i> file as the default mixed-signal control file.
-ams	Invokes the Verilog-AMS compiler.
-ams_crule [<i>library</i> .]< <i>ruleID</i> >	Picks up a specific connectrule from a particular library at elaboration time when this option is used in the VCS command line. Library name is optional. If the library names are not mentioned, then libraries are searched in the Synopsys library search order to find the requested connectrule.
-ams_discipline < <i>discreteDiscipline</i> >	Defines a default discrete discipline (disciplines explicitly defined on nets or pins override this option).
-ams_iereport	Displays all the connectmodule instances in the design.

Simulating the Design

You can use the following command to simulate the design:

```
% simv [<runOption>]
```

Limitations in VCS-AMS Flow

The following design configurations are not supported in the VCS-AMS flow:

- VHDL-top design, where a VHDL net is directly or hierarchically connected to an analog port.
- VHDL real or integer port connected to an analog high-conn.

Support for Predefined Nettypes in SystemVerilog-SPICE Flow

As defined in the IEEE Std SystemVerilog LRM 2012, SystemVerilog supports nettypes. This allows you to describe general abstract values for a net with an optional resolution function.

VCS supports the predefined nettypes in the SystemVerilog-SPICE flow. The nettype objects may be directly or hierarchically connected to SPICE components in a design. This enhancement enables you to develop and connect multiple design blocks that

communicate with each other using complex data types and sophisticated net resolution mechanisms, such as:

- The interface element is defined at the analog or the digital boundary that transfers values from a nettype to the analog solver or vice-versa.
- As analog tunneling through nettype is supported, the analog tunneling takes place between two distant SPICE ports if they are hierarchically connected through nettype.

Note:

If multiple SPICE ports are connected through the same nettype net, then those SPICE ports are collapsed to a single SPICE node.

Predefined Nettypes

Using the predefined nettypes, you can precisely communicate electrical voltage, current or Thevenin information among multiple analog and digital blocks. The following predefined nettypes are available in the \$VCS_HOME/etc/snps_msv/snps_msv_nettype_pkg.svp package:

- `r_wire`

`r_wire` is a structure that is considered as a nettype without any resolution function. It consists of one member, which is of `real` type.

- `v_wire_sum`

`v_wire_sum` is a structure that is considered as a nettype by resolving multiple driver voltage values. The resolved voltage value on this nettype is the sum of the voltages of the active drivers. This has two members, one real type named as `v` and one logic type named as `active`.

For example,

```
v_wire_sum vs;
```

`vs.v` —holds the driver voltage

`vs.active`—the driver voltage is counted when `vs.active` is “1”.

- `v_wire_avg`

`v_wire_avg` is a structure that is considered as a nettype by resolving multiple driver voltage values. The resolved voltage value on this nettype is the average of the voltages of the active drivers. This has two members, one real type named as `v` and one logic type named as `active`.

For example,

`v_wire_avg va;`

`va.v`— holds the driver value

`va.active`—the driver voltage is counted when `vs.active` is “1”.

- `v_wire_one`

`v_wire_one` is structure that is considered as a nettype. Only one driver’s “active” value should be ‘1’ and this driver’s value is driven to left-hand side. If there is more than one active driver, the “active” field of the resolved value is “0”.

This nettype has two members, one real type named as `v` and one logic type named as `active`.

For example,

`v_wire_one vo;`

`vo.v`— holds the driver value

`vo.active`—the driver value is counted when `vo.active` is “1”

- `i_wire`

`i_wire` is a structure that is considered as a nettype by resolving multiple driver current values. The resolved current value on this nettype is the sum of the active drivers. This nettype has two members, one real type named as `i` and one logic type named as `active`.

For example,

`i_wire iw;`

`iw.i`—holds the driver current value

`iw.active`—the driver current is counted when `iw.active` is “1”.

- `th_wire`

`th_wire` is a structure that is considered as a nettype to model Thevenin resistance and voltage. The `th_wire` nettype consists of the following three members:

- “real” type member that holds the voltage value named as `vth`
- “real” type member that holds the resistance value named as `rth`
- one logic type named as `active`

For example,

```
th_wire tw;
```

`tw.vth`— holds the voltage value.

`tw.rth`— holds the resistance value.

`tw.active`— the voltage and resistance values are counted to resolve the final Thevenin voltage and resistance when `tw.active` is “1”.

Resolution function for this nettype resolves all active drivers (for which “active” is 1) to drive the final Thevenin voltage and resistance on left-hand side.

Use Model

During the elaboration step, enable mixed-signal simulation using the `-ad=initfile` elaboration option using the following command:

```
% vcs -sverilog -ad=initfile [elab_options] \
top_entity/module/config
```

To enable the SystemVerilog nettype support in the SystemVerilog-SPICE flow, include the `sv_nettpe enable;` option in the mixed-signal setup file (`init` file).

Example

The following example illustrates the use of predefined nettypes in a SystemVerilog-SPICE boundary.

```
01_test.v
`timescale 1ns/10ps

import snps_msv_nettpe_pkg::*;

`define TOLERANCE 0.0001

module sv_top;
  v_wirer nt_wire;
  v_wire_sum sp_in1, vhd_out;

  initial begin
#1;
    nt_wire.v = 0.0; // #1
    nt_wire.active = 1;
#10;
    nt_wire.v = 2.0; // #11
    nt_wire.active = 1;
#10;
    nt_wire.v = 1.0; // #21
```

```

        nt_wire.active = 1;
        #10;
        nt_wire.v = 2.0; // #31
        nt_wire.active = 1;
        #10;
        nt_wire.v = 1.0; // #41
        nt_wire.active = 1;
        #100 $finish;
    end

    assign sp_in1 = nt_wire;

    sp1 SP1 (sp_in1, vhd_out);
    sp2 SP2 (sp_in1, vhd_out);

endmodule

module sp1 (input v_wire_sum sp_in, output v_wire_sum sp_out);
endmodule

module sp2 (input v_wire_sum sp_in, output v_wire_sum sp_out);
endmodule

```

01_test.spi

```

* SUBCKT inverter
.inc '01_test.mod'
.subckt sp1 sp_in sp_out
mn1 sp_out sp_in gnd gnd n w=1.00u l=0.35u
mp1 sp_out sp_in vdd vdd p w=2.00u l=0.35u
.ends

.subckt sp2 sp_in sp_out
mn1 sp_out sp_in gnd gnd n w=1.00u l=0.35u
mp1 sp_out sp_in vdd vdd p w=2.00u l=0.35u
.ends

.global vdd gnd
vsu vdd gnd 3.3
.end

```

01_test.init

```

sv_nettypes enable;
choose xa -n 01_test.spi;
use_spice -cell sp1;
use_spice -cell sp2;

```

Command Line

```

% vlogan 01_test.v -sverilog -full64
% vcs sv_top -ad=01_test.init -full64
% simv

```

User-Defined Nettypes

User-defined nettypes are nettypes other than the predefined nettypes that are defined in the Synopsys package. These nettypes are not allowed to connect to any SPICE sub-circuit unless the following method is used.

Consider the following user-defined nettypes, data structures and resolution functions in the HDL models and flow:

```
volt_u, volt: Unresolved and resolved "voltage" values.  
current_u, current: Unresolved and resolved "current" values.  
iv_u, iv: Unresolved and resolved "voltage" and "current" values.
```

All these nettypes are defined by the user in a SystemVerilog package. The user-defined nettype should be mapped to a predefined nettype so that it can connect to SPICE. The mapping to the nettype ID is made as follows:

```
vcsAD.init:  
nettype_map <library>.<package>.  
<nettype information> => <nettype alias name>;
```

For example, the `nettype_map` command in the mixed-signal setup file (`vcsAD.init` file) can be as follows:

```
nettype_map msv_lib.msv_types.volt_u => msv_ie_v_wire_av_r;  
nettype_map msv_lib.msv_types.volt => msv_ie_v_wire_av_r;  
nettype_map msv_lib.msv_types.current_u => msv_ie_i_wire_r  
nettype_map msv_lib.msv_types.current => msv_ie_i_wire_r  
nettype_map msv_lib.msv_types.iv_u => msv_ie_th_wire_r  
nettype_map msv_lib.msv_types.iv => msv_ie_th_wire_r
```

If you map a nettype as shown in the following example, then MSV identifies that usage of `current_u` is same as `i_wire`.

```
nettype_map msv_lib.msv_types.current_u => msv_ie_i_wire_r;
```

You can use the following interface element names for all predefined types:

VCS_NETTYPE_R_WIRE,	msv_ie_r_wire_ur	r_wire
VCS_NETTYPE_V_WIRE_AVG,	msv_ie_v_wire_av_r	v_wire_avg
VCS_NETTYPE_V_WIRE_SUM,	msv_ie_v_wire_sum_r	v_wire_sum
VCS_NETTYPE_V_WIRE_ONE,	msv_ie_v_wire_one_r	v_wire_one
VCS_NETTYPE_I_WIRE,	msv_ie_i_wire_r	i_wire
VCS_NETTYPE_TH_WIRE,	msv_ie_th_wire_r	th_wire

Example

The following example illustrates how the `nettype_map` command is added in the mixed-signal setup file (`vcsAD.init` file):

01_test.v

```

`timescale 1ns/10ps
package MP;
  typedef struct {
    real v;
    logic active;
  } nt_v;

  function automatic nt_v nt_v_rf_s(input nt_v driver[]);
    nt_v_rf_s.v = 0.0;
    nt_v_rf_s.active = 1'b0;
    foreach(driver[i]) begin
      if (driver[i].active) begin
        nt_v_rf_s.active = 1'b1;
        nt_v_rf_s.v += driver[i].v;
      end
    end
  endfunction

  nettype nt_v nt_v_s with nt_v_rf_s;
endpackage : MP;

import snps_msv_nettype_pkg::*;
import MP::*;

module sv_top;
  nt_v nt_wire;
  nt_v_s sp_in1;

  initial begin
    #1;
    nt_wire.v = 0.0; // #1
    nt_wire.active = 1;
    #10;
    nt_wire.v = 2.0; // #11
    nt_wire.active = 1;
    #10;
    nt_wire.v = 1.0; // #21
    nt_wire.active = 1;
    #10;
    nt_wire.v = 2.0; // #31
    nt_wire.active = 1;
    #10;
    nt_wire.v = 1.0; // #41
    nt_wire.active = 1;
  end
endmodule

```

```

        #100 $finish;
end

assign sp_in1 = nt_wire;
nt_v_s nt_tmp;

sp1 SP1 (sp_in1, nt_tmp );
sp2 SP2 (sp_in1, nt_tmp );

endmodule

module sp1 (input nt_v_s sp_in, output nt_v_s sp_out);
endmodule

module sp2 (input nt_v_s sp_in, output nt_v_s sp_out);
endmodule

```

01_test.spi

```

* SUBCKT inverter
.inc '01_test.mod'
.subckt sp1 sp_in sp_out
mn1 sp_out sp_in gnd gnd n w=1.00u l=0.35u
mp1 sp_out sp_in vdd vdd p w=2.00u l=0.35u
.ends

.subckt sp2 sp_in sp_out
mn1 sp_out sp_in gnd gnd n w=1.00u l=0.35u
mp1 sp_out sp_in vdd vdd p w=2.00u l=0.35u
.ends

.global vdd gnd
vsu vdd gnd 3.3
.end

```

01_test.init

```

sv_nettypes enable;
choose xa -n 01_test.spi;
use_spice -cell sp1;
use_spice -cell sp2;
nettype_map work.MP.nt_v_s => msv_ie_v_wire_sum_r;

```

Command line

```

% vlogan 01_test.v -sverilog -full164
% vcs sv_top -ad=01_test.init -full164

```

Limitations

This feature has the following limitations:

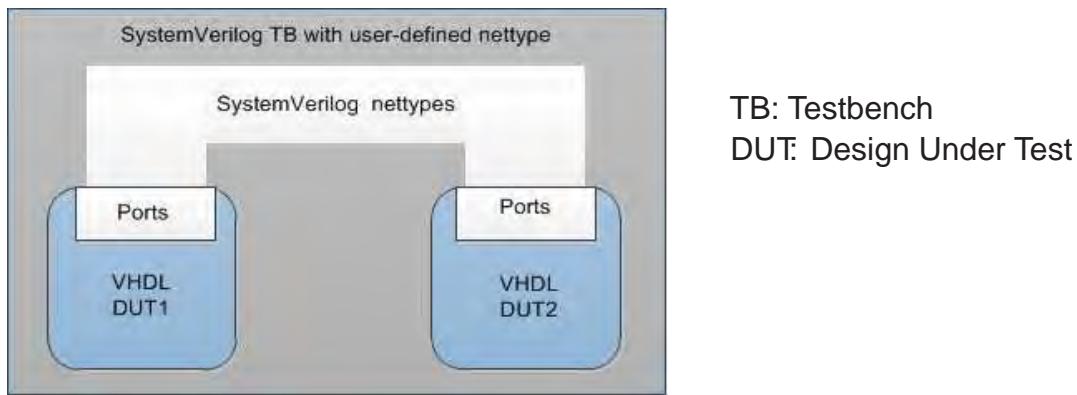
- Verilog-AMS flow is not supported.
- SystemVerilog array-nettype is not supported at SPICE boundary.
- User-defined nettype that is not mapped to predefined nettype ID is not supported at SPICE boundary.
- The INOUT nettype is not supported at SPICE boundary.
- Only three-step flow is supported in SystemVerilog-SPICE flow and two-step flow is not supported.

Support for SystemVerilog Nettypes in Mixed-Signal Simulation

VCS supports SystemVerilog nettypes in mixed-signal simulation. This enhancement enables you to perform real number modeling, where the SystemVerilog nettypes and VHDL user-defined data types are used to develop a model, such as an analog behavioral model.

In [Figure 182](#), the user-defined record type ports of DUT1 and DUT2 are connected through the SystemVerilog parent using nettypes.

Figure 182 Mixed-Language Design With User-Defined Nettypes



In general, the record data type can contain one or more user-defined real type fields. With this feature, VCS supports the following data types in a mixed-language simulation:

- SystemVerilog nettype with compatible VHDL type/subtype
- The underlying type of the nettype and the corresponding VHDL type/subtype can be a scalar or complex user-defined types such as a record.

Type Compatibility Between Nettype and VHDL Type/Subtype

To simulate mixed-language port connections, the connected nettype and VHDL type/subtype should be type-compatible. This is ensured by VCS with appropriate semantic checks.

You can provide this type-compatibility information at VCS compile-time using the `synopsys_sim.setup` file. For VHDL top design style, VCS also checks the type-compatibility information at “vhdlan” time for direct instantiation and when binding is explicitly provided.

Editing the Setup File

Include the `mx_type_mapfile = <type-mapping-file-name>` command in the `synopsys_sim.setup` file, where `<type-mapping-file-name>` is the name of the file that contains all the mixed-signal boundary nettype mapping information.

The type mapping file can be mentioned with an absolute path, relative path or by using an environment variable.

Syntax of the entries in the type-mapping-file is:

```
mx_nettype_map <lib_name>.<VHDL_package_name>.<VHDL_recordtype_name>
<lib-name>.<SV_package_name>.<nettype_name>;
```

In this mapping, the left-hand side should contain VHDL type/subtype and the right-hand side should contain SystemVerilog nettype name.

Note:

In the mapping, VHDL type/subtype must be stated before the SystemVerilog nettype.

The rules in map file must be mentioned in a single line. An error message is displayed when unexpected mapping rule is specified.

Rules for Type Compatibility

Ensure that the mapped VHDL types/subtypes and SystemVerilog nettypes are compatible. VCS checks for type compatibility only for the types that are used in the

design source code (these types should also be specified in the type map file) and issues an error message if the types are not compatible.

- If built-in or predefined types and subtypes such as std_logic, bit, Boolean, std_ulogic and so on are used as VHDL type in the type-map file, VCS displays an error message. For the list of built-in or predefined types, see the standard and 1164 IEEE package.
- If the VHDL type/subtype is a record, the corresponding type of nettype must be a SystemVerilog structure.
 - The number of fields in the VHDL record type must match the number of fields in the SystemVerilog structure type.
 - Each field of the VHDL record is type-compatible with the corresponding field in the SystemVerilog structure.

The VHDL user-defined types and SystemVerilog nettypes that are mapped through the type mapping file must always have 1-1 mapping. However, one-many or many-one mapping in the type-mapping file is not allowed.

Example 1

The following example illustrates the use of VHDL user-defined types in mixed-signal simulation.

VHDL Leaf—This package is parsed into the logical library, VH_LIB:

```
ex_pkg.vhd

package vh_pack is

  type vh_netT is record
    current_val : real;
    is_active   : bit;
  end record vh_netT;

  type vh_netvectorT is array (natural range <>) of vh_netT;
  function sum_resol(driver : in vh_netvectorT) return           vh_netT;
  subtype vh_current is sum_resol vh_netT;

end package vh_pack;
...

library ieee,vh_lib;
use vh_lib.vh_pack.all;
entity vh_dut is
  port (signal current_mfr : in vh_current) ;
end entity;

architecture arch of vh_dut is
begin
end arch;
```

SystemVerilog top—nets_package:

ex_pkg.sv

```
package nets_package ; // This package is parsed into logical
                      //library VL_LIB
typedef struct {
  real i;
  logic active;
}vl_netT;

function automatic vl_netT vl_net_rf_sum(input vl_netT driver[]);
  vl_net_rf_sum.i = 0.0;
  vl_net_rf_sum.active = 1'b0;
  ...
endfunction

nettype vl_netT vl_current with vl_net_rf_sum;
endpackage : nets_package

import nets_package::*;

module top;
  vl_current n1;
  vh_dut DUT (n1);
endmodule
```

synopsys_sim.setup file:

```
WORK > DEFAULT
DEFAULT : work
vh_lib : ./vh_lib
vl_lib : ./vl_lib

mx_type_mapfile = ./type_mapfile
```

Entries in type_mapfile:

```
mx_nettype_map vh_lib.vh_pack.vh_current vl_lib.nets_package.vl_current;
```

Command lines:

```
% vhdlan ex_pkg.vhd -work vh_lib
% vlogan ex_pkg.sv -sverilog -work vl_lib
% vcs vl_lib.top
% ./simv
```

Example 2

The following example illustrates the scalar real type support in mixed-signal simulation.

VHDL File: test.vhd

```
package vhdl_package is
  type real_ut_vector is array (natural range <>) of real;
  function real_ut_sum_rf(driver : in real_ut_vector) return real;
```

```

  subtype real_t is real_ut_sum_rf real;
end package;

...

use work.vhdl_package.all;
entity vhtop is
end entity;

architecture arc of vhtop is
  signal s : real_t := 0.0;

begin
  VL_CHILD: entity work.vl_child port map (s);
end architecture;

```

Verilog file: test.v

```

package vlog_package;
  function automatic real real_sum_rf(input real driver[]);
    real_sum_rf = 0.0;
    foreach (driver[i]) begin
      real_sum_rf += driver[i];
    end
  endfunction
  nettype real real_t with real_sum_rf;
endpackage : vlog_package

import vlog_package::*;

module vl_child (p);
  input p;
  real_t p;
endmodule

```

type_mapfile:

```
mx_nettype_map WORK.vhdl_package.real_t WORK.vlog_package.real_t;
```

synopsys_sim.setup file:

```
WORK > DEFAULT
DEFAULT : work
mx_nettype_mapfile = ./type_mapfile
```

Command Lines:

```
% vlogan -sverilog test.v
% vhdlan test.vhd
% vcs vhtop
% simv
```

Limitations

This feature has the following limitations:

- Array of SystemVerilog nettype is not supported.
- Nettype with nested structures is not supported.
- Nettype with array of structure is not supported.

Using SystemVerilog Nettypes in Mixed-Signal Simulation

VCS supports SystemVerilog nettypes in mixed-signal simulation. This enhancement helps you to use real number modeling, where the SystemVerilog nettypes and VHDL real types are useful to develop analog behavioral model. This improves the performance significantly over SPICE designs.

To use this feature, you must know about the usage of SystemVerilog nettypes in mixed-language simulation. For more information, see section [Support for SystemVerilog Nettypes in Mixed-Signal Simulation](#).

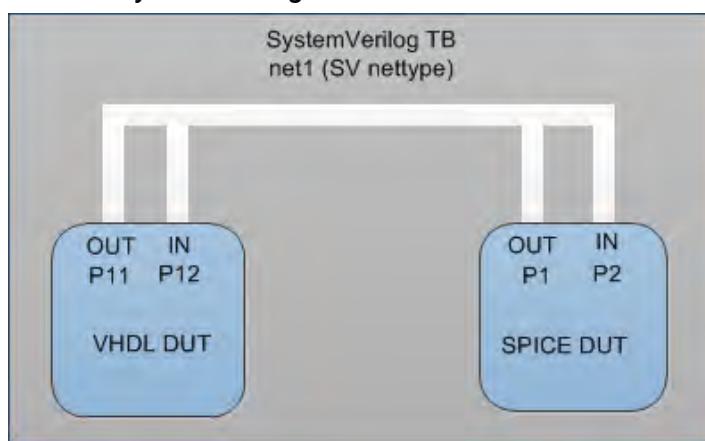
Supported Topologies

VCS supports the following mixed-signal topologies:

Case 1 - SystemVerilog Testbench With SPICE and VHDL Children

In this topology, the SPICE and VHDL ports are connected through SystemVerilog nettype nets in a SystemVerilog testbench as shown in [Figure 183](#).

Figure 183 SystemVerilog Testbench With SPICE and VHDL DUT



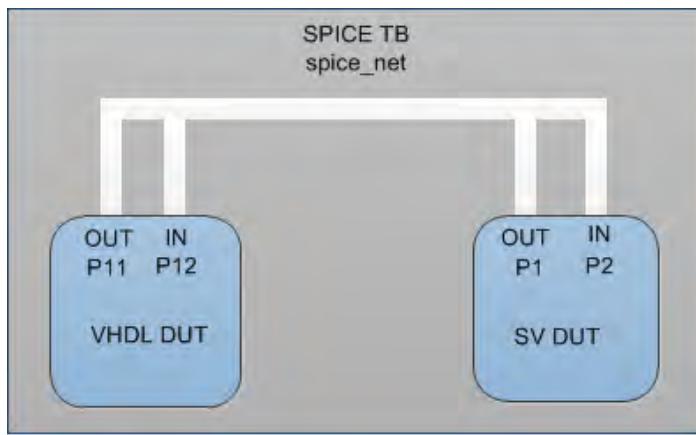
Verilog multiple view module for the SPICE child is required in this scenario. The ports of this multiple view module should have appropriate nettype information.

For more information about multiple view module, see *Discovery AMS: Mixed-Signal Simulation User Guide*.

Case 2 - SPICE Testbench With VHDL and SystemVerilog Children

In this topology, SystemVerilog and VHDL children are connected through the SPICE net in a SPICE testbench as shown in [Figure 184](#).

Figure 184 SPICE Testbench With SystemVerilog and VHDL DUT



Use Model

This section provides the use model to simulate mixed-signal designs as described in the following section:

- If a mixed-signal design contains a VHDL component, VCS should be invoked with the `synopsys_sim.setup` file that contains the type-mapping information between the VHDL type/subtype and the connected SystemVerilog nettype.

For more information about type mapping, see section [Type Compatibility Between Nettype and VHDL Type/Subtype](#).

- At the SPICE boundary only predefined nettypes are supported. The following is the list of predefined nettypes that are available in the `$VCS_HOME/etc/snps_msv/snps_msv_nettpe_pkg.svp` package:
 - `r_wire`
 - `v_wire_avg`
 - `v_wire_sum`

- v_wire_one
- i_wire
- th_wire

For more information about the predefined nettypes, see section [Support for Predefined Nettypes in SystemVerilog-SPICE Flow](#).

Example

The following example illustrates the use of VHDL record type at SPICE boundary.

01_test.v

```
`timescale 1ns/10ps

import snps_msv_nettype_pkg::*; //predefined package

`define TOLERANCE 0.0001

module sv_top;
    v_wiret nt_wire;
    v_wire_sum sp_in1, vhd_out;

    initial begin
        #1;
        nt_wire.v = 0.0; // #1
        nt_wire.active = 1;
        #10;
        nt_wire.v = 2.0; // #11
        nt_wire.active = 1;
        #10;
        nt_wire.v = 1.0; // #21
        nt_wire.active = 1;
        #10;
        nt_wire.v = 2.0; // #31
        nt_wire.active = 1;
        #10;
        nt_wire.v = 1.0; // #41
        nt_wire.active = 1;
        #100 $finish;
    end

    assign sp_in1 = nt_wire;

    vh_child1 VHDL1 (sp_in1, vhd_out);
    vh_child2 VHDL2 (sp_in1, vhd_out);

endmodule
```

```
module vlog (input v_wire_sum vlog_in, output v_wire_sum vlog_out);
    sp2 SP2 (vlog_in, vlog_out);
endmodule

module sp1 (input v_wire_sum sp_in, output v_wire_sum sp_out);
endmodule

module sp2 (input v_wire_sum sp_in, output v_wire_sum sp_out);
endmodule
```

Note:

The `snps_msv_nettype_pkg` package is predefined and pre-analyzed package included with the VCS installation. You need to add proper import clause to enable the definitions of this package that are visible in design code or module.

01_test.spi

```
* SUBCKT inverter
.inc '01_test.mod' //This is a parameter file
.subckt sp1 sp_in sp_out
mn1 sp_out sp_in gnd gnd n w=1.00u l=0.35u
mp1 sp_out sp_in vdd vdd p w=2.00u l=0.35u
.ends

.subckt sp2 sp_in sp_out
mn1 sp_out sp_in gnd gnd n w=1.00u l=0.35u
mp1 sp_out sp_in vdd vdd p w=2.00u l=0.35u
.ends

.global vdd gnd
vsu vdd gnd 3.3
.end
```

01_test_01.vhd

```
library ieee;
use ieee.std_logic_1164.all;
use work.snps_vhdl_package.all;
-- snps_vhdl_package is a user-defined package
use std.textio.all;
use ieee.std_logic_textio.all;
entity vh_child1 is
    port ( sp_in1 : in    v_wire_sum := (0.0, '0');
           vhd_out  : out   v_wire_sum := (0.0, '0') );
end;

architecture a of vh_child1 is

component sp1 is
port ( sp_in : in v_wire_sum;
```

```

        sp_out : out v_wire_sum);
end component sp1;

signal sp_out1 : v_wire_sum := (0.0, '0') ;
begin

vh_inst_of_sp1 : sp1 port map (sp_in1, sp_out1);

vhd_out <= sp_out1;

end;

```

Note:

The `snps_vhdl_package` is a user-defined VHDL package, compiled into the work library. This package contains VHDL types/subtypes that are used in this example.

01_test_02.vhd

```

library ieee;
use ieee.std_logic_1164.all;
use work.snps_vhdl_package.all;
use std.textio.all;
use ieee.std_logic_textio.all;
entity vh_child2 is
    port ( vlog_in : in    v_wire_sum := (0.0, '0');
           vhd_out : out   v_wire_sum := (0.0, '0') );
end;

architecture a of vh_child2 is

component vlog is
port ( vlog_in : in v_wire_sum;
       vlog_out : out v_wire_sum);
end component vlog;

signal vlog_out1 : v_wire_sum := (0.0, '0') ;
begin

vh_inst_of_vlog : vlog port map (vlog_in, vlog_out1);

vhd_out <= vlog_out1;

end;

```

01_test.init

```

sv_nettypes enable;
choose xa -n 01_test.spi;

```

```
use_spice -cell sp1;
use_spice -cell sp2;
```

Type_mapfile

The following entries should be included in the type_mapfile.

```
mx_nettype_map WORK.snps_vhdl_package.v_wire_sum
  SNPS_MSV.snps_msv_nettype_pkg.v_wire_sum ;
```

synopsys_sim.setup file

```
WORK > DEFAULT
DEFAULT : work
mx_type_mapfile = ./type_mapfile
```

Command line

```
% vlogan 01_test.v -sverilog -full164
% vhdlan snps_vhdl_package.vhd 01_test_01.vhd \
01_test_02.vhd -full164
% vcs sv_top -ad=01_test.init -full164
```

Limitations

This feature has the following limitations at SPICE boundary:

- SystemVerilog array-nettype is not supported.
- Nettype with nested structures is not supported.
- Nettype with array of structure is not supported.
- The INOUT nettype is not supported.
- Only predefined nettypes are allowed. However, if the user-defined nettype is mapped to a predefined nettype ID, these nettypes are supported.

Support for SystemVerilog Packed Array at Mixed Design Boundary

VCS supports SystemVerilog array types for the bit and logic base element types on the boundary with VHDL. The arrays can be packed, unpacked, a combination of packed and unpacked, and can have more than two dimensions.

The supported Verilog base types are as follows:

- bit
- logic

The supported VHDL base types are as follows:

- bit / bit_vector
- std_logic / std_logic_vector
- std_ulogic / std_ulogic_vector

In Verilog, the arrays can be fully packed, fully unpacked, or a mix of packed and unpacked as shown in the following examples:

```
typedef logic [DIM1:0] [DIM2:0] array_packed;
typedef logic array_unpackd [DIM1:0] [DIM2:0];
typedef logic [DIM2:0] array_mix [DIM1:0];
```

In VHDL, you can use the following array types:

```
type array1 is array (integer range <>) of std_logic;
type array2 is array (integer range <>, integer range <>) of std_logic;
type array3 is array (integer range <>) of std_logic_vector;
type array4 is array (integer range <>) of std_logic_vector (7 downto 0);
type array5a is array (integer range <>) of std_logic;
type array5b is array (integer range <>) of array5a;
type array5c is array(integer range <>, integer range <>) of array5b;
```

Editing the Setup File

To enable the behavior of the array dimensions cross boundary, apply the following setting in the `synopsys_sim.setup` file:

```
----- Elaboration Settings
-----
---- Relaxes the use of SystemVerilog packed arrays -- on the language
boundary
MX_RELAX_PACKED_ARRAYS = TRUE
```

Key Points to Note

- You can specify the size of the dimensions with constants and with generics or parameters.
- You can use arrays at input, output, and inout ports.
- The elaboration / runtime options are not required.

Verilog-on-Top Scenario

Consider instantiating the following VHDL entity in Verilog:

```
library ieee;
use ieee.std_logic_1164.all;
package pack is
  type array2dvec is array (integer range <>, integer range <>) of
    std_logic_vector(7 downto 0);
end package pack;

library ieee;
use ieee.std_logic_1164.all;
use work.pack.all;
entity dut is
  port (
    p1 : in array2dvec(1 downto 0, 3 downto 0);
    p2 : out array2dvec(1 downto 0, 3 downto 0));
end entity dut;
architecture dut_arch of dut is
begin
end architecture dut_arch;
```

The VHDL entity can be instantiated in the following Verilog module:

```
module top;
  logic [1:0][3:0][7:0] data1;
  wire [1:0][3:0][7:0] data2;
  dut H(data1, data2);
  // rest of module
endmodule
```

VHDL-on-Top Scenario

Consider instantiating the following Verilog module in VHDL:

```
module vlog_dut(input [1:0][3:0][7:0] p1,
                  output [1:0][3:0][7:0] p2);
endmodule
```

The Verilog module can be instantiated in the following VHDL module:

```
library ieee;
use ieee.std_logic_1164.all;
package pack is
  type array2dvec is array (integer range <>, integer range <>) of
    std_logic_vector(7 downto 0);
end package pack;

library ieee;
use ieee.std_logic_1164.all;
```

```

use work.pack.all;
entity top is
end entity top;
architecture top_arch of top is
  signal data1 : array2dvec(1 downto 0, 3 downto 0);
  signal data2 : array2dvec(1 downto 0, 3 downto 0);
begin
  V: entity work.vlog_dut port map (
    p1 => data1,
    p2 => data2);
end architecture top_arch;

```

Limitations

- Connecting arrays on the mixed design boundary must have the same number of base elements. When the number of base elements are not same, VCS generates an elaboration error.
 - Total number of dimensions must match, that is, the number of packed and unpacked dimensions in Verilog must be the same as the total number of array dimensions in VHDL.
 - Base elements, like integer, real, records, are not supported and when used, VCS generates an elaboration error.
-

Support for Unmapped User-Defined Nettypes in SystemVerilog-SPICE Flow

VCS supports user-defined nettypes that are not mapped to any predefined nettypes at the SystemVerilog -SPICE boundary when following conditions are satisfied:

- Unmapped user-defined nettypes do not have either analog load or analog driver.
- For unmapped user-defined nettypes, both a2d and d2a interface must be optimized in the presence of either or both of the following runtime switches:
 - -ad_runopt=a2dopt
 - -ad_runopt=d2aopt

With this enhancement, when both a2d and d2a interfaces are optimized and removed, the unmapped user-defined nettypes are supported through the SPICE hierarchy as SPICE blocks neither read nor contribute to the value of the nettype signal. This passthrough net optimization provides better flexibility to use SPICE netlists to stitch multiple SystemVerilog blocks in a design.

Use Model

The VCS runtime switches, `-ad_runopt=a2dopt` and `-ad_runopt=d2aopt` can be used for a2d and d2a optimization.

Example

`test.v`

```
'timescale 1ns/10ps
package MP;                      // Verilog Package for user defined nettype
  typedef struct {
    real v;
    logic active;
  } nt_v;

  function automatic nt_v nt_v_rf_s(input nt_v driver[]);
    nt_v_rf_s.v = 0.0;
    nt_v_rf_s.active = 1'b0;
    foreach (driver[i]) begin
      if (driver[i].active) begin
        nt_v_rf_s.active = 1'b1;
        nt_v_rf_s.v += driver[i].v;
      end
    end
  endfunction

  nettype nt_v nt_v_s with nt_v_rf_s; // User defined nettype
endpackage : MP;

import MP::*;

module sv_top;
  nt_v nt_wire1, nt_wire2;
  nt_v_s sp_in1, sp_in2;
  int i;
  real r;

  initial begin
    $display("start simulation");
    i = 1;
    r = 1.0;
    for (i = 1; i <= 6; i++) begin
      nt_wire1.v = r;
      nt_wire1.active = 1;
      nt_wire2.v = r;
      nt_wire2.active = 1;
      r = r + 1.0;
      #10;
    end
    #10 $finish;
  end
endmodule
```

```

        end

        always @(sp_in1.v) begin
            #1;
            $display("%g: in1: %1.4f | in2: %1.4f", $time, sp_in1.v,
sp_in2.v);
        end

        assign sp_in1 = nt_wire1;
        assign sp_in2 = nt_wire1;
        assign sp_in2 = nt_wire2;

        sp1      SP1(sp_in1, sp_in2); // User defined nettype at SV-SPICE
boundary

endmodule

module vbot (output nt_v_s in1, output nt_v_s in2);
    nt_v nt_wire1, nt_wire2;
    int k;
    real r;

    initial begin
        k = 1;
        r = 1.0;
        for (k = 1; k <= 6; k++) begin
            nt_wire1.v = r;
            nt_wire1.active = 1;
            nt_wire2.v = r;
            nt_wire2.active = 1;
            r = r + 1.0;
            #10;
        end
    end
    assign in1 = nt_wire1;
    assign in2 = nt_wire1;
    assign in2 = nt_wire2;
    assign in2 = in1;

endmodule

module sp1 (output nt_v_s sp1_in1, output nt_v_s sp1_in2);
endmodule

module sp2 (output nt_v_s sp2_in1, output nt_v_s sp2_in2);
endmodule

test.spi

//Pass through User defined nettype in SPICE

.subckt sp1 sp1_in1 sp1_in2

```

```
x1 sp1_in1 sp1_in2 sp2
.ends

.subckt sp2 sp2_in1 sp2_in2
x2 sp2_in1 sp2_in2 vbot
.ends
```

test.init

```
sv_netttype enable;
choose xa -n test.spi;
use_spice -cell sp1;
use_spice -cell sp2;
```

Command Lines

```
% vlogan <test.v> -sverilog
% vcs -q sv_top -ad=test.init
% ./simv -ad_runopt=a2dopt+d2aopt
```

Limitation

If the `optimize_shadowfile` command is included in the mixed signal setup file (`vcsAD.init`), the analog contribution information is set by CustomSim for SPICE signals in the shadow module and a2d or d2a optimization does not occur. Therefore, both a2d and d2a interfaces cannot be removed and unmapped user-defined nettypes are not supported at the SV-SPICE boundary.

Support for User-Defined Nettypes in Mixed-Signal Simulation

Custom User-Defined Nettypes (UDNs) are supported at the mixed signal boundaries. These UDNs can contain several fields for example, voltage field, current field, and so on.

Use Model

This section provides examples to explain the use model for setting up the custom UDNs at the mixed signal boundaries as follows:

Mixed-Signal Control Command

In the mixed-signal control file such as, `vcsAD.init`, additional commands must be added to map a UDN to its interface elements as follows:

```
udn_n2e type=<UDN_name> module=<va_module>;
udn_e2n type=<UDN_name> module=<va_module>;
```

For example,

```
udn_n2e type=mynettype module=my_n2e_va;
udn_e2n type=mynettype module=my_e2n_va;
```

Where,

`mynettype` is the name of the UDN that appears at mixed signal boundary.

The Verilog-A module `my_n2e_va` is used to send the nettype value to Spice. The last port of this module connects to Spice and the previous ports connect to the respective fields of the UDN `mynettype`.

The Verilog-A module `my_e2n_va` is used for the reverse communication, that is, to send Spice value to UDN `mynettype`. The first port of this Verilog-A module connects to Spice and the following remaining ports connect to the respective fields of this UDN.

Parameter values can also be passed to a Verilog-A interface element module using `<field>=<value>` syntax, as follows:

```
udn_e2n type=mynettype module=va_e2n param1=1.1 param2=2.2;
```

Here,

`param1=1.1` and `param2=2.2` are used to set the values of the parameters `param1` and `param2` of the Verilog-A module `va_e2n` to `1.1` and `2.2` respectively.

These Verilog-A interface element modules are included in the `interface_element.rpt` file.

Usage Example

The following example illustrates usage of custom UDNs at the mixed signal boundaries:

```
typedef struct {
  real voltage;
  real resistance;
  logic active;
  integer stat1; //field is not used in the analog domain
  real stat2;   //field is not used in the analog domain
} my_nettypeT;

function automatic my_nettypeT my_nettype_rf (...);
endfunction

nettype my_nettypeT my_nettype;
nettype my_nettypeT my_nettype_one with my_nettype_rf;
// VA module for N2E
module va_n2e (voltage, resistance, active, stat1, stat2, out);
  input voltage, resistance, active, stat1, stat2;
  output out;
```

```

electrical voltage, resistance;
logic active;
  electrical stat1, stat2;
  electrical out;
  real R;

analog begin
  @(initial_step) R = 1.0;
  R = V(resistance);
  if ~active R = 1.0E30;
  V(out) <+ V(voltage) - I(out)*R;
end
endmodule

// VA module for E2N
module va_e2n (in, voltage, resistance, active, stat1, stat2);
  input in;
  output voltage, resistance, active, stat1, stat2;
  electrical in;
  electrical voltage, resistance;
  electrical stat1, stat2;
  logic active;
  real p1v, p2v, p3v, p4v, p5v;
  real r_v, r_r, r_a, r_s1, r_s2;

analog begin
  @(initial_step) begin
    r_v = 0.0;
    r_r = 0.0;
    r_a = 0;
    r_s1 = 0.0;
    r_s2 = 0.0;
  end
  r_v = V(in);
  r_r = $snps_get_driver_strength(in);
  if (r_r > 10000000) r_a = 0;
  else                  r_a = 1;

  V(voltage) <+ r_v;
  V(resistance) <+ r_r;
  V(active) <+ r_a;
  V(stat1) <+ r_s1;
  V(stat2) <+ r_s2;
end
endmodule

```

Limitations

This feature has the following limitation:

- Fields in the UDN are always scalar such as real, logic, bit, and so on. UDN with field of type vector, struct, union, and array is not supported.

Enhancements in Type Coercion and Converter Insertions

Type-coercion and converter-insertion areas are enhanced to improve the quality. This section describes the various type coercion and converter insertion enhancement available in VCS:

Enhancements for Type Coercion

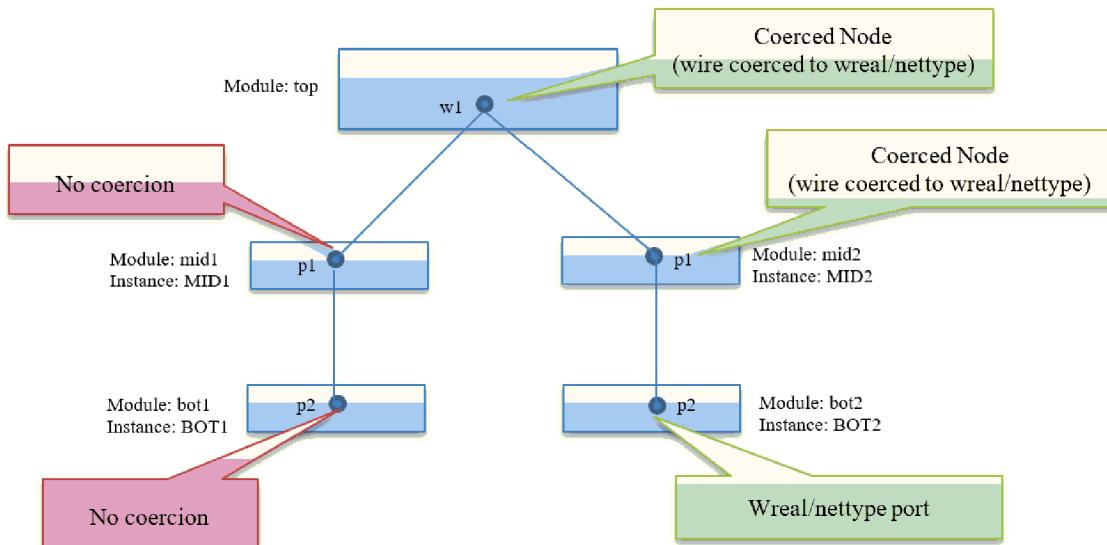
The following section describes the enhancements for type-coercion:

- [Support for Basic Type-Coercion](#)
- [Enhancement to Type Coercion Issues for SV Interface](#)

Support for Basic Type-Coercion

The basic RNM coercion feature allows concrete types to propagate bottom-up only. You can enable basic RNM coercion by using the `-coercion=basic` option.

Figure 185 Basic Type Coercion



Enhancement to Type Coercion Issues for SV Interface

VCS supports the following SV interface scenarios:

- An SV-interface that has one or more virtual-interfaces.
 - One or more wires/interconnects of SV-interface are coerced due to type-coercion.
 - The SV-interface is split because wires/interconnects of this interface have been coerced differently in different instances of this interface.

Enhancements in Diagnostics and Report Generation

The following diagnostics and reports are generated:

- Unified converter insertion report is generated in partition-compile flow.
 - Coercion report is generated for an erroneous design.

Enhancements to Converter Insertions

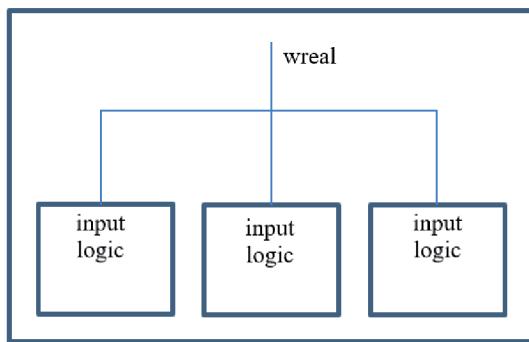
The following section describes the enhancements in converter insertions:

- Support for Merge Mode Converter Insertion

Support for Merge Mode Converter Insertion

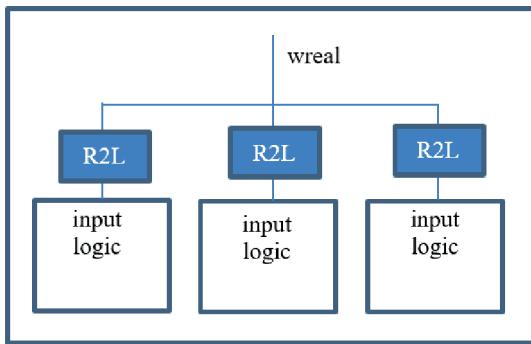
In the previous releases, converters are inserted at every port-connect boundary where types of high-conn and low-conn are incompatible. This is called split-mode converter insertion. This leads to large number of converter instances and may cause performance issue.

Figure 186 Design Before Converter Insertion.



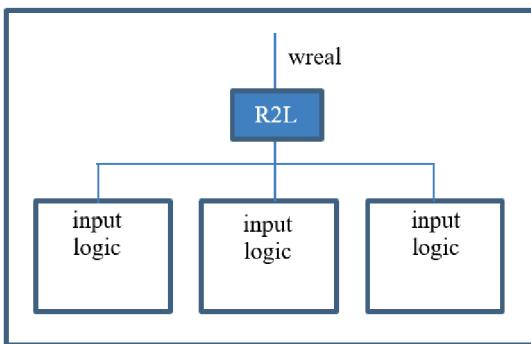
The following figure shows split-mode converter insertion where at every wreal/logic boundary one R2L (real-to-logic) converter has been instantiated.

Figure 187 Split-Mode Converter Insertion



VCS supports merge-mode converter insertion where the tool inserts one converter from type T1 to type T2 within a module and then shares this converter-instance for multiple port-connections inside that module those require conversion from T1 to T2. The following figure shows merge mode converter insertion where one R2L (real-to-logic) converter has been instantiated and it is shared by several wreal/logic boundaries.

Figure 188 Merge-Mode Converter Insertion



Quality Enhancements for Converter Insertion

The quality of converter insertion is enhanced by supporting the following:

- nets inside SV-Interface hierarchy
- nets of `wand`, `wor`, `tri*`, `supply0`, `supply1` types
- converter insertion at UDP port-connection

Options for Coercion Flow

The following describes the options enhanced in this release:

Options	Description
<code>-coerce</code>	Enables the coercion flow. Enables both <code>wreal</code> and <code>nettype</code> coercions. Obsolete warning is issued for all old options.
<code>-coerce=no_wire_coercion</code>	Disables wire coercion. Does not treat wire as interconnect. Does not coerce any wires connected to <code>wreal</code> / <code>nettype</code> .
<code>-coerce=legacy_flow</code>	Disables coercion flow with obsolete warning. Ignores all other <code>-coerce</code> options.
<code>-coerce=diag</code>	Generates type <code>-coercion-report</code> and <code>adaptor-insertion-report</code> in the new coercion flow. <ul style="list-style-type: none"> • Default coercion report file: <code>simv.daidir/rnm_coerced_info.log</code> • Default converter report file: <code>simv.daidir/rnm_converter_info.log</code> • Default Bi-Dir converter report file: <code>simv.daidir/rnm_bidir_converter_info.log</code>
<code>-coerce=basic</code>	Generates type <code>-coercion-report</code> and <code>adaptor-insertion-report</code> in the new coercion flow. <ul style="list-style-type: none"> • Default coercion report file: <code>simv.daidir/rnm_coerced_info.log</code> • Default converter report file: <code>simv.daidir/rnm_converter_info.log</code> • Default Bi-Dir converter report file: <code>simv.daidir/rnm_bidir_converter_info.log</code>
<code>-coerce=basic</code>	Enables basic coercion. Bottom up traversal only is supported.
<code>-coerce=detailed</code>	Enables detailed coercion. Coercion traverses up and down.
<code>-coerce=svreal</code>	Enables <code>-wreal</code> coerce. (wires connected to SV-real get coerced to <code>wreal</code>)

Enhancements to Event Functions in SV-SPICE Flow

The following sections describe the enhancements to the event functions in the SV-SPICE Flow:

- [Introduction](#)
- [Enhancements to Event Functions](#)
- [VCS Mixed-Signal Simulation](#)

Introduction

Analog event functions, which are defined in Verilog-AMS flow, are used to capture analog event in digital context. The following are the three analog event functions that are defined in Verilog-AMS LRM:

```
cross ( expr [, dir[ , time_tol [ , expr_tol [ , enable ] ] ] ] )
above ( expr [, time_tol [ , expr_tol [ , enable ] ] ] )
absdelta( expr, delta [, time_tol [ , expr_tol [ , enable ] ] ] )
```

Enhancements to Event Functions

VCS supports the following for the event functions in the SV-SPICE flow:

- The `snps_absdelta` event function in the SV-SPICE flow.

Following is the syntax of the `snps_absdelta()` event function:

```
snps_absdelta( expr, delta [ , time_tol [ , expr_tol [ , enable ] ] ] )
```

- Local digital variables in the first argument of all three analog event functions. Digital variables are also supported for the `delta` argument of the `snps_absdelta` event function.

The type of the digital local variables can be integer, reg, logic, or element of integer/int array.

- The `enable` argument in all the three analog event functions as follows:

```
snps_cross ( expr [, dir[ , time_tol [ , expr_tol [ , enable ] ] ] ] )
snps_above ( expr [, time_tol [ , expr_tol [ , enable ] ] ] )
```

```
snps_absdelta( expr, delta [, time_tol [ , expr_tol [ , enable ] ] ] )
```

The `enable` argument must be resolved to an integral type. Local digital variables can be used. The type of the digital variables can be integer, int, reg, logic, or element of integer/int array.

Examples

Example for the snps_cross Event Function

The following example in SV-SPICE shows a cross function to a SPICE net with enable from integer variable:

```
always @($snps_cross($snps_get_volt(spice.out0), - 0.6, 0, 1e-6, 1e-6,
dint)) begin
$display("cross to a spice net with local int variable enable has
triggered");
end
```

Example for the snps_above Event Function

In the following example, `enab1` is declared as a reg and toggles every 7ns:

```
always @($snps_above($snps_get_volt(rc1.out)-0.5,ttol,tol,enab1))
$display("Above");
always #7_000 enab1 = ~enab1;
initial enab1 = 1'b0;
endmodule
```

Example for the snps_absdelta Event Function

Following is the example for `snps_absdelta` with SPICE voltage and VerilogA real as `expr`, Verilog real as `delta`, and Verilog int as `enable`:

```
always @($snps_absdelta($snps_get_voltage(r1.p)-r1.areal, d1, tt1, tt2,
en_abs))
begin
$display($time,,, " inside absdelta");
end
```

Support for UPF Power-Aware Interface Elements in Mixed Signal Simulation Environment

The A2D interface element is automatically inserted between SystemVerilog and SPICE to allow the electrical to logic conversion. The D2A is automatically inserted between SystemVerilog and SPICE to allow the logic to electrical conversion.

The UPF power-aware interface elements are supported in the mixed-signal simulation environment. You can use the `-power=supply_aware_ie` option at compile time to obtain the UPF supply (power and ground) information for an interface element (inserted by VCS at the analog-digital boundary) within the digital-analog boundary.

Following is the use model: % vcs -power=supply_aware_ie -ad=vcsAD.init -upf <upf_file.upf> -sverilog <design files>

During simulation, VCS NLP updates the value of the UPF supply when it changes. AMS uses the UPF supply registered for a A2D signal to get the logic from voltage value and D2A signal to get the voltage value of logic.

This feature allows you to simulate the interface elements inserted by the simulation tool at the analog-digital boundaries in the mixed-signal simulation environment.

Association of UPF Supplies to Interface Elements

The following sections describe the semantics to determine the UPF power and ground information:

- [Boundary SPICE Port With SPA Specified in the UPF](#)
- [Boundary SPICE Port With SRSN Specified in the UPF](#)
- [SPICE Cell With Liberty Information](#)
- [SPICE Cell Without SPA, SRSN, and Liberty Information](#)

Boundary SPICE Port With SPA Specified in the UPF

If the boundary SPICE port has the `set_port_attributes` command specified in the UPF, then:

- For D2A interface element (SPICE input port), receiver supply (`-receiver_supply`) option of SPA is used.
- For A2D interface element (SPICE output port), driver supply (`-driver_supply`) option of SPA is used.

Boundary SPICE Port With SRSN Specified in the UPF

If the boundary SPICE port has the `set_related_supply_net` command specified in the UPF, then:

- If SPA is specified on the same SPICE port along with SRSN, then SPA overrides SRSN (that is, SRSN is ignored as per VCS NLP behavior).
- Otherwise, the supply information associated with SRSN is used.

SPICE Cell With Liberty Information

For a SPICE cell with liberty information:

- The UPF supply connected to the related power pin and ground pin of the boundary SPICE port is used.
- If the `connect_supply_net` command is specified on the related power and ground pins, the UPF supply information associated with the `connect_supply_net` is used.
- The following rules are applied to find the connected supply to the related power and ground pins:
 - If the SPICE cell is an isolation DB cell, then the UPF supply of the corresponding UPF isolation strategy is used.
 - If the associated isolation strategy does not have any isolation supply specified, then the default isolation supply of the power domain is considered.
 - When the DB cell is neither an isolation cell nor there is any associated isolation policy, the primary supply of the power domain in which the SPICE cell is instantiated is used.

SPICE Cell Without SPA, SRSN, and Liberty Information

If no SPA, SRSN, and liberty information is specified for a SPICE cell, then AMS assumes that the supply of the digital net is same as the supply of the power domain where the SPICE cell is instantiated.

Examples

The following sections describe the usage examples:

- [Example-1: Input Case](#)
- [Example-2: Output Case](#)

Example-1: Input Case

UPF

```
create_supply_set SS_srsn -function {power VDD_srsn} -function {ground
VSS_srsn}
set_related_supply_net -object_list { m/m/sp1/in1} -power VDD_srsn
-ground VSS_srsn
```

SRSN is defined for SPICE input port. Therefore, the supply associated with the pin is SRSN power supply.

vcsAD.init

```
use_spice -cell sp1;

test.spi

.subckt sp1 in1 out1 vdd vss
mn1 out1 in1 vss vss n w=20.00u l=0.35u
mp1 out1 in1 vdd vdd p w=40.00u l=0.35u
.ends
```

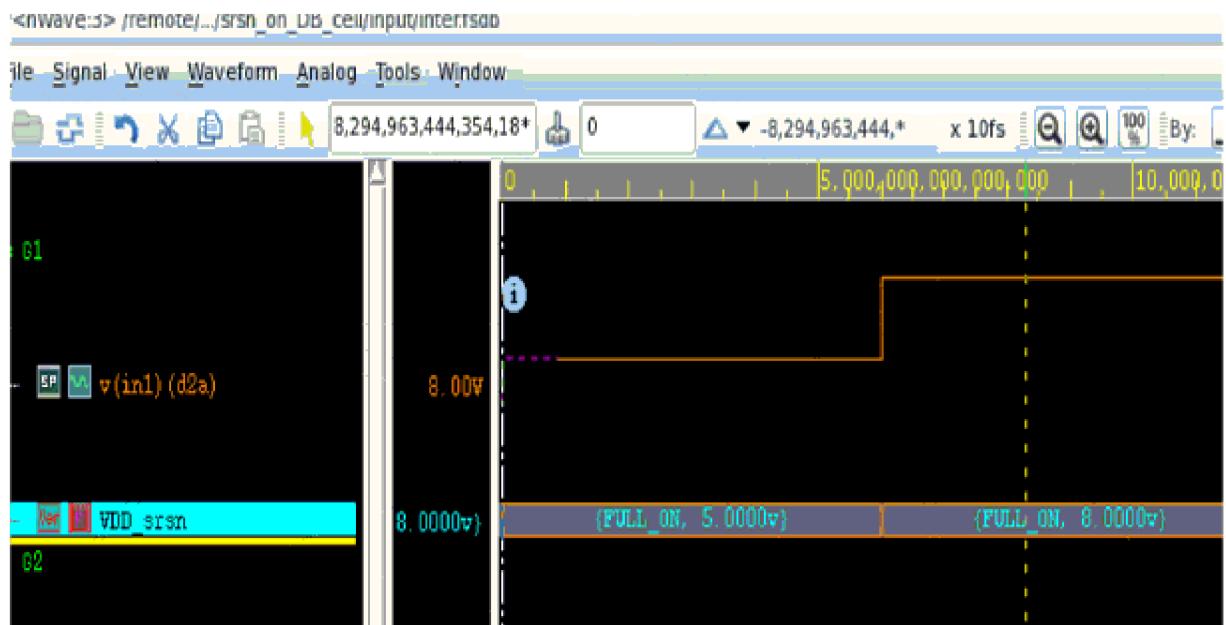
Where, sp1 is a not gate.

Testbench

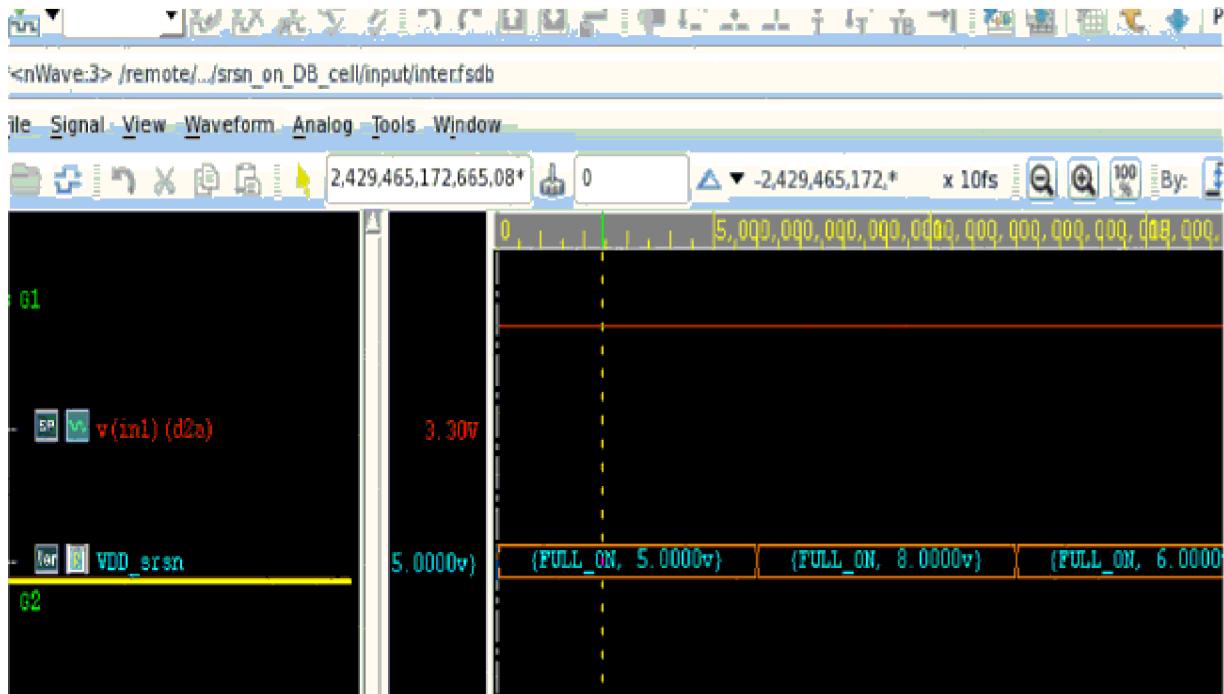
```
supply_on("VDD_srsn",5.0);
#60 ; supply_on("VDD_srsn",8.0);
```

Here, at time 0, SRSN power is 5.0. After 60s, it changes to 8.0.

Figure 189 Simulation Result With the -power=supply_aware_ie Option



In the above figure, VDD_srsn and SPICE input has the same voltage values. When VDD_srsn changes, SPICE input value also changes.

Figure 190 Simulation Result Without the -power=supply_aware_ie Option

In the above figure, SPICE values are not supply aware. When SRSN supply value changes, SPICE voltage value does not change and holds 3.3 voltage which is the default D2A voltage.

Reporting

VCS NLP determines the actual driver of the signal and identifies the supply. This enhancement generates the correct D2A interfaces in the `simv.msv/interface_element.rpt` file as follows:

```
d2a hiv=100% lov=0% vdd=tb/top_unit/VDD_srsn vss=tb/top_unit/VSS_srsn
node=tb.top_unit.m.m.sp1.in1;
```

Following is the output without the `-power=supply_aware_ie` option:

```
d2a hiv=3.3v lov=0v node=tb.top_unit.m.m.sp1.in1;
```

Example-2: Output Case

UPF

```
create_supply_set SS_srsn -function {power VDD_srsn} -function {ground
VSS_srsn}
set_related_supply_net -object_list {m/m/sp1/out1} -power VDD_srsn
-ground VSS_srsn
```

SRSN is defined for the SPICE output port. Therefore, the supply associated with the pin is the SRSN power supply.

vcsAD.init

```
use_spice -cell sp1;

test.spi

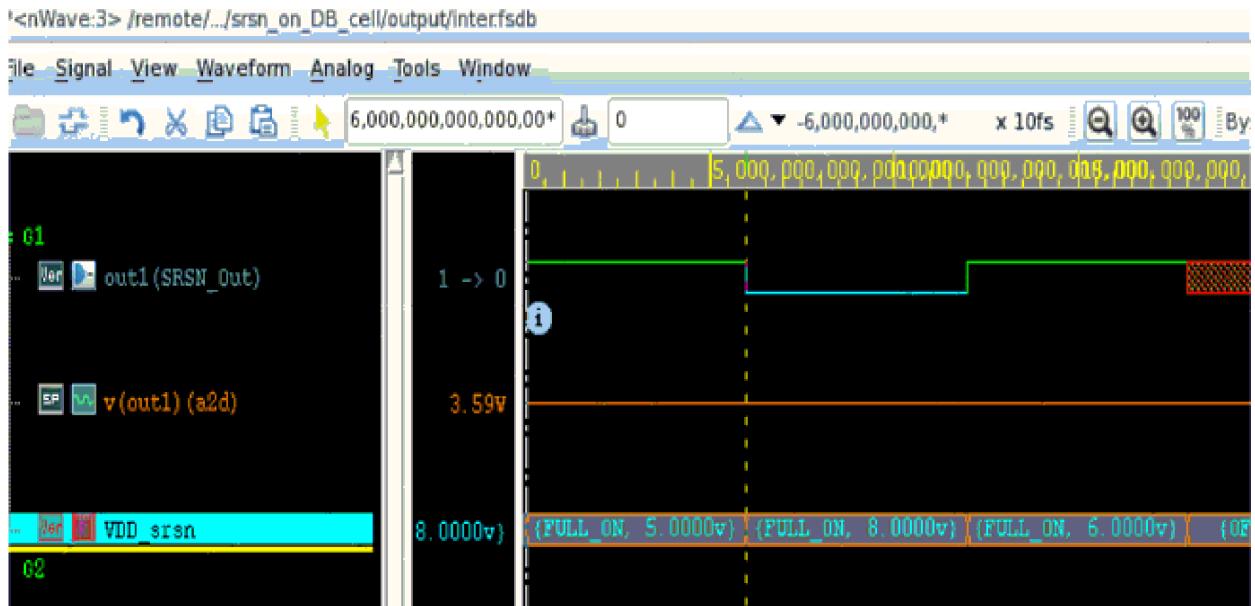
.subckt sp1 in1 out1 vdd vss
R1 temp1 temp 2k
R2 temp 0 3k
R3 temp out1 3k
V1 temp1 0 6v
.ends
```

This is a simple voltage divider. The voltage of `out1` is 3.6.

Testbench

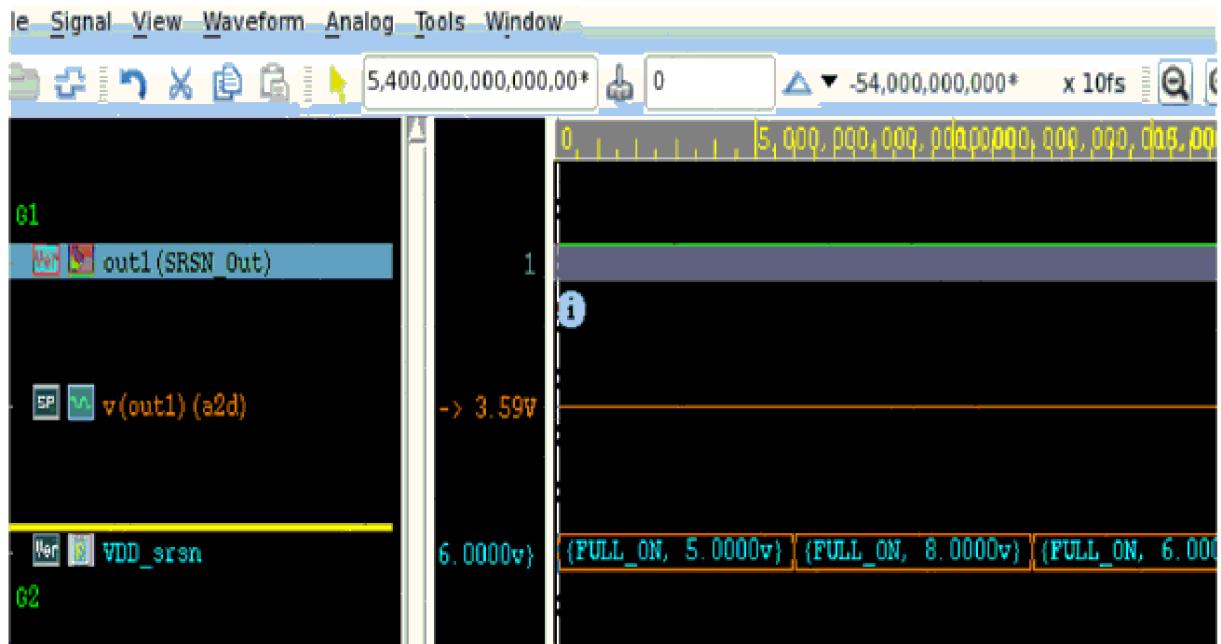
```
supply_on("VDD_srsn",5.0);
#60 ; supply_on("VDD_srsn",8.0);
```

Figure 191 Simulation Result With the `-power=supply_aware_ie` Option



In the above figure, when `VDD_srsn` changes to 8.0, `out1` changes to 0. In AMS, A2D higher and lower threshold values are 50% of the A2D supply voltage. When `VDD_srsn` is 8.0, higher threshold value is 4.0, which is greater than the SPICE output voltage. Therefore, the corresponding digital value is 0.

Figure 192 Simulation Result Without the -power=supply_aware_ie Option



In the above figure, when SRSN supply value changes, the digital value does not change and holds to 1.

Reporting

The supply usage for A2D conversion is included in the `simv.msv/interface_element.rpt` file:

```
a2d loth=50% hith=50% vdd=tb/top_unit/VDD_srsn vss=tb/top_unit/VSS_srsn
minv=0.1 minv_logic=x node=tb.top_unit.m.m.sp1.out1;
```

Real Number Modeling With VIZ Nettypes

There are many ways to model the behavior of an analog component in the digital domain using a real number model. Those models often capture the analog behavior in terms of node voltages, and in some cases in terms of port currents.

VIZ (Voltage, Current, Impedance) nettype and modeling is a way to model the ports of analog components via three parameters, which are Voltage, Current, Impedance, and then use Kirchoff's Current Law (KCL) to calculate the node voltages when those ports are connected to each other.

The difference between this method of analog behavioral modeling and the more conventional way of RNM is that in the conventional way a very simplistic resolution

function in Verilog is used to calculate a node voltage when multiple RNM ports drive voltage values (for example, the resolution function uses the average of all drivers).

The resolved values for node voltages calculated by the resolution functions are not accurate in that case. And for current drivers, there is not a very meaningful resolution function available in Verilog. In other words, if multiple RNM ports are connected to a node, the Verilog resolution functions do a very poor job of accurately calculating the node voltage. But VIZ is solving that problem.

By requiring each analog model to capture its behavior as values for V, I, and Z fields at its ports, the VIZ resolution function then uses the KCL equations to accurately calculate the node voltages, similar to the way SPICE simulators do so. The following are the two components of the VIZ RNM modeling:

- The modeling of analog ports using the three fields of the VIZ nettype (depending on the modeling requirement, the model may only use one, two, or all three fields)
- The implementation of the KCL equations in the resolution function

Using the above two components, models for basic analog primitives, such as Resistors, Capacitors, Inductors, Voltage, and Current sources can be generated and simulated with VCS.

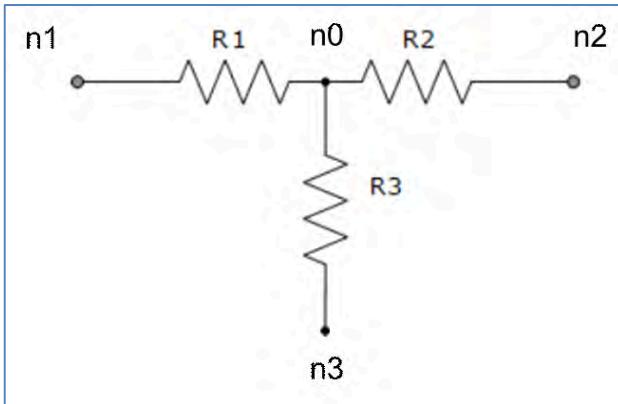
Examples

KCL in Resolution Function for Voltage Calculation

While the responsibility of a VIZ-based model is to accurately capture the behavior of the analog model at each one of the model's VIZ ports, the responsibility of the VIZ resolution function is to take all of the values driven by the VIZ ports connected to a node and produce the correct node voltage, that is, coming up with a resolved value for the V field of VIZ. To do that, the resolution function relies on the KCL set of equations.

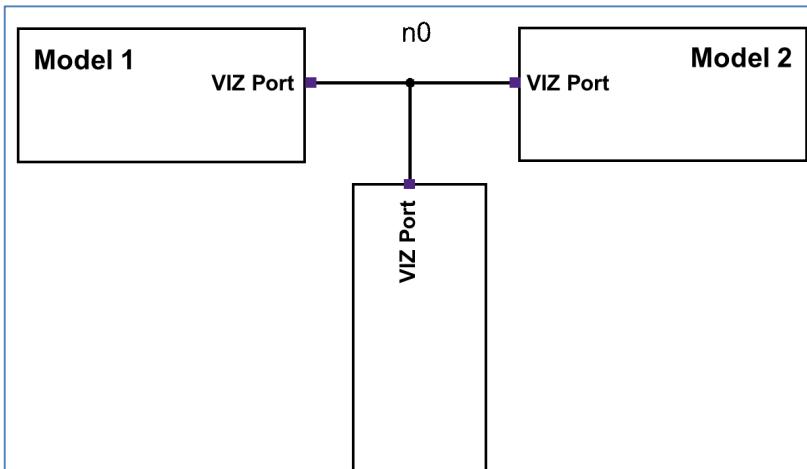
The KCL equations are based on the fact that the sum of all currents flowing into a node must be zero. For example, if there is a 10mA flow of current into a node from one branch, the other branch(es) must have a net flow of -10mA current flowing into the node (meaning a net outflow of 10mA current) to make the total current flow into the node 0.

The following figure shows an example circuit where three resistors are connected to node "n0". You can calculate the voltage of n0, assuming that you have the voltage values for n1, n2, n3, and values for R1, R2, and R3.

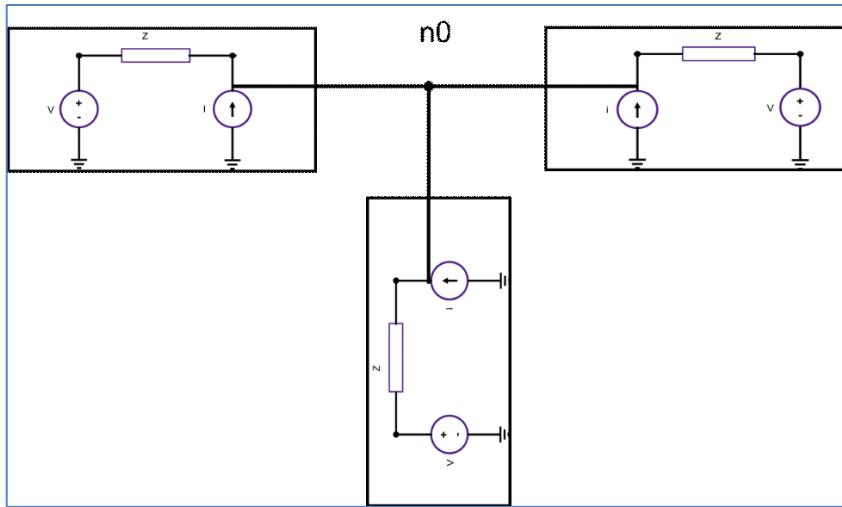


Using KCL for VIZ Models

In the previous section, the use of KCL for the calculation of node voltage in a resistive network was shown. That concept can be extended to other models, more complex than resistors. The following figure shows one such example, where three VIZ ports belonging to three different behavioral models connect to node n0.



The responsibility of each one of the three models, Model 1, 2, and 3, is to correctly capture the behavior of their ports connected to node n0 by driving the correct values of V, I, and Z for their corresponding ports. The following shows how each one of the three models drives the correct V, I, and Z values on their ports to capture the analog behavior of those ports and lead to a correct voltage calculation by the resolution function at node n0.



For example, if the VIZ port for Model 1 drives with $Z=0$, its **V** value determines the resolved voltage value for node **n0**. So, to model an ideal supply using a VIZ nettype, you need to set the **V** field to the desired voltage level, the **Z** field to 0, and the value of **I** does not matter (safe to set it to 0 as well).

Library of Predefined VIZ Models and Resolution Function

The library to be shipped with VCS includes the resolution function and a set of predefined models with VIZ ports. Those models could be as basic as Voltage and Current sources and Resistors, or as advanced (in future releases) as Capacitors, Inductors, or even MOS transistors.

Connectivity Technologies in VCS

This section summarizes multiple simulation technologies implemented in VCS toward connectivity.

Interconnect Versus Wire in System Verilog LRM

System Verilog LRM defines generic-interconnect to represent a generic form of net that is typeless. This interconnect definition provides additional flexibility over a wire. It describes only connectivity; thus, it is typeless or generic whose semantics are derived from the endpoints.

This allows the same model to be used for various levels of abstractions. For example, the same interconnect can be used to describe the connectivity to either a digital model,

a discrete analog model, or a continuous analog model (SPICE) – the only thing that changes is the endpoint, that is, the model itself does not have to be treated differently.

Figure 193 SV-Interconnect

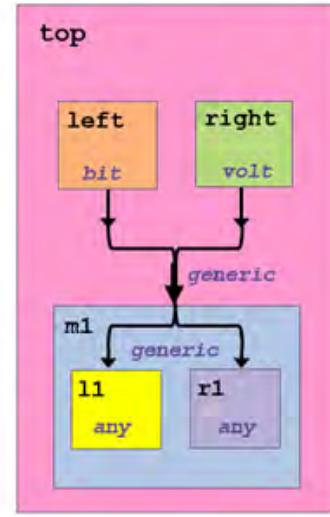
```
nettype real volt;

module top();
    interconnect [0:1] bus;
    bDriver left(bus[0]);
    vDriver right(bus[1]);
    rlMod m1(bus);
endmodule

module vDriver(output volt vout); ...
```

```
module bDriver(output bit bout); ...
```

```
module rlMod(interconnect [0:1] bus);
    lMod l1(ibus[0]);
    rMod r1(ibus[1]);
endmodule
```



Conversely, a Verilog wire has a very definitive semantic in the language: it is a logic (that is, digital) as specified in section 6.7.1 of the LRM:

If a data type is not specified in the net declaration or if only a range and/or signing is specified, then the data type of the net is implicitly declared as logic. For example:

```
wire w; // equivalent to "wire logic w;"  
wire [15:0] ww; // equivalent to "wire logic [15:0] ww;"
```

if there are no loads and drivers on a wire, it can be used as interconnect using the `-coerce` option. Thus, wires and interconnects can be used for connectivity.

Type-Coercion

Type-coercion algorithm determines the final type of all SV generic-interconnects and wire-interconnects in a design. There are two kinds of type-coercion: detailed coercion and basic coercion.

Detailed Coercion

The detailed type-coercion feature allows a wire/tri net (without any use in the design) to be converted and simulated as Wreal/Nettype if any Wreal/Nettype is present in its connected net hierarchy. The following figures show a design before and after coercion.

Figure 194 Design Before Coercion

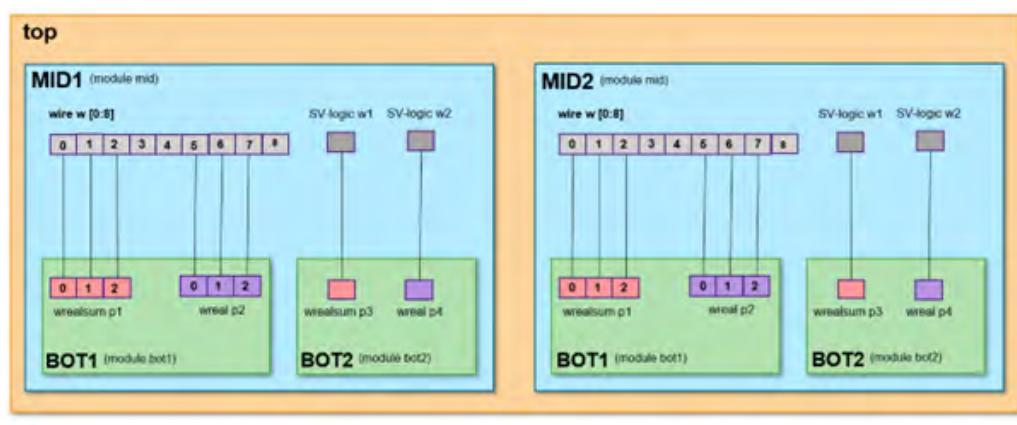
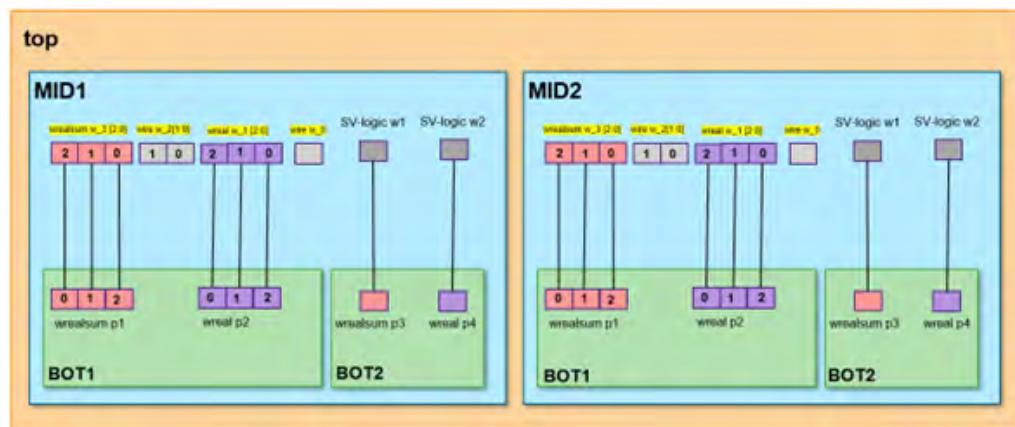


Figure 195 Design After Coercion



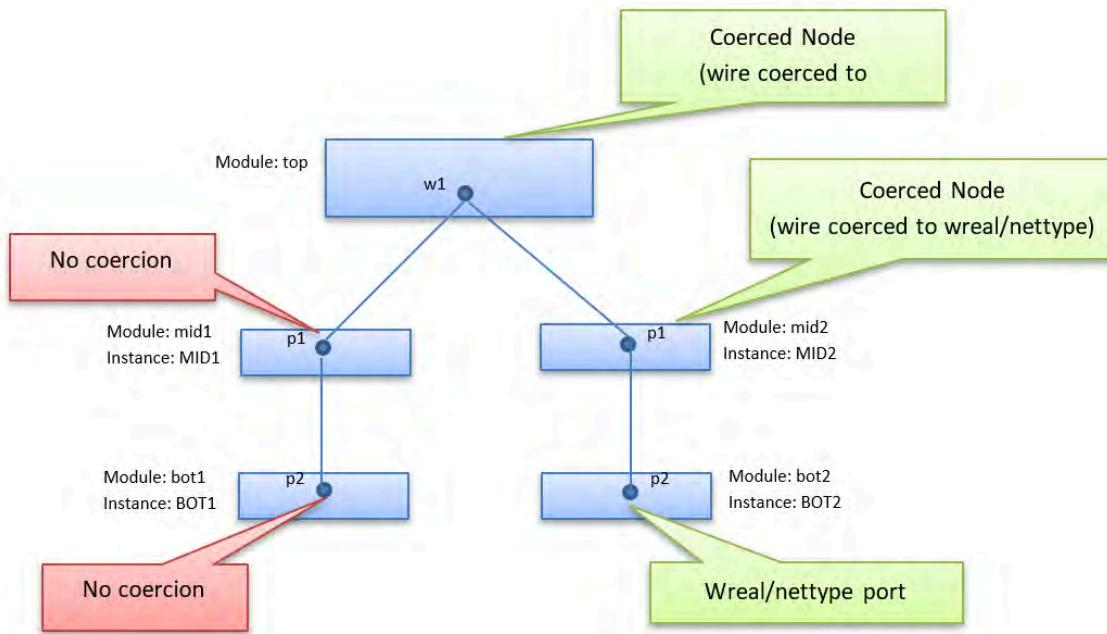
To use this feature, run the following command:

```
% VCS -coerce <other VCS options>
```

Basic Coercion

VCS supports basic (bottom-up) RNM coercion in which concrete types are propagated only in the bottom-up direction. Basic coercion is enabled by the `-coerce=basic` compile-time option.

Figure 196 Basic Coercion



Coercion in Wire Versus SV-Interconnect

System Verilog LRM specifies that a net can have only one type. It can be wire or SV-nettype. In VCS, SV-nettype also includes wreal, even though wreal is not part of the LRM.

SV-interconnect is a special case as it is typeless (generic) by default and takes its type from the connected nets on either side.

Before the introduction of the coercion technology, a similar concept existed in VCS for wires. If a wire did nothing else except connecting two nodes, it could be “coerced” or cast to the type of the connections the wire had on HighConn and LowConn.

This feature for the wire is now enabled within the LRM framework when connectivity is achieved using SV-interconnect.

Analog simulator (XA) accomplishes the same behavior when resolving through net.

Command-Line Options for Coercion

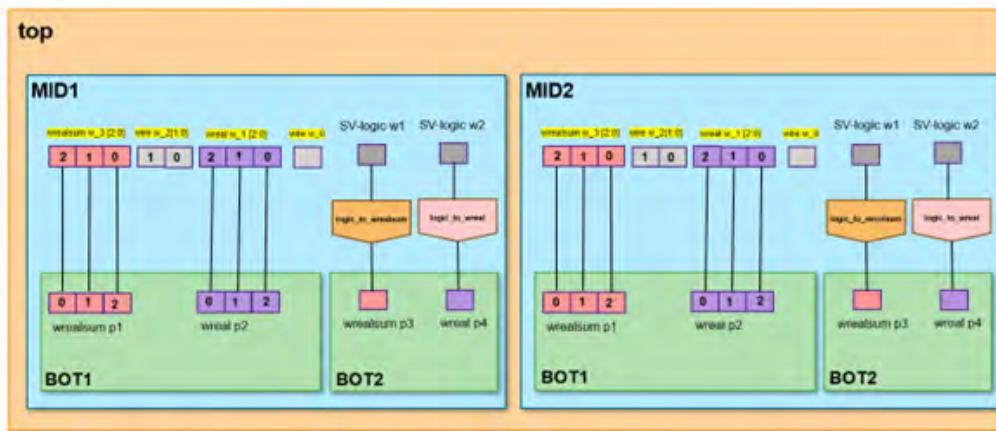
The following table contains information about the options relevant to the VCS coercion technology.

Option	Description
-coerce	Enables the type-coercion functionality. Enables both wreal and nettype coercions. For type-coercion of generic-interconnect (SV LRM feature), it has been made default.
-coerce=no_wire_coercion	Disables wire coercion. Does not treat wire as interconnect.
-coerce=legacy_flow	Disables the new coercion flow with obsolete warning.
-coerce=diag	Generates type-coercion-report and adaptor-insertion-report in the new coercion flow.
-coerce=basic	Enables basic coercion. Supports only bottom-up traversal.
-coerce=detailed	Enables detailed coercion. Coercion traverses both up and down. This is the default setting.
-coerce=svreal	Enables -wreal coercion. Wires connected to SV-real get coerced to wreal.

VCS Converters

VCS supports converter insertion at port level, gate primitives (local converters), bidirectional converters, power-aware converters, and signal-based converters.

Figure 197 Design After Converter Insertion



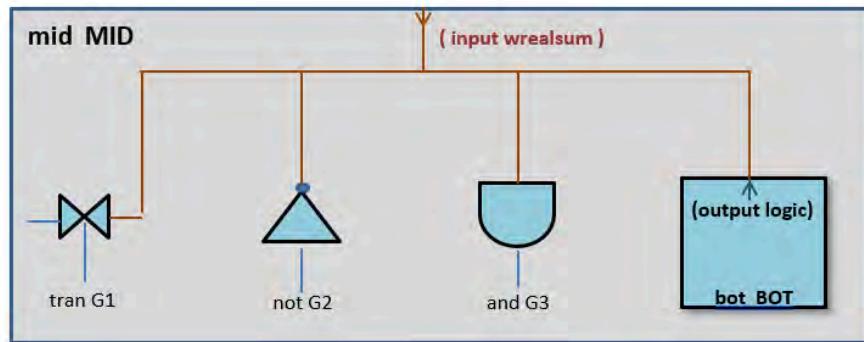
Local Converters

All Verilog gate primitives, such as tran, tranif, and, nand, buf, and so on are supported with local converters.

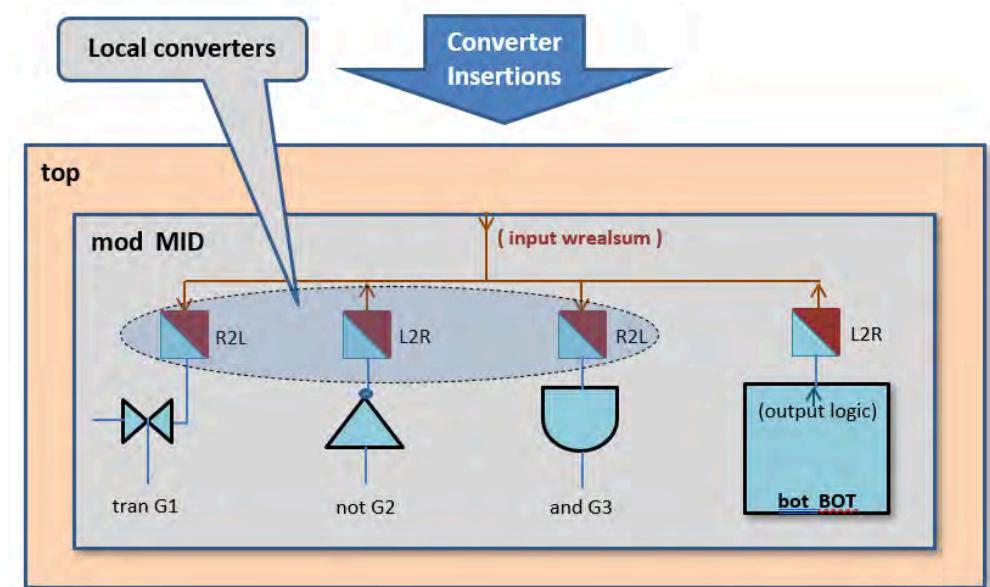
To use local converters, specify the following command in the configuration file.

```
enable_local_converter ;
converter <source type> <destination type> <converter module> ;
```

Example



```
enable_local_converter;
converter logic wrealsum L2R;
converter wrealsum logic R2L;
```



Converter Insertion

VCS allows you to insert converter modules between two SystemVerilog or Verilog-AMS modules for automatic type conversion between ports. In mixed-signal designs, this feature applies only to discrete ports.

Using this feature, if a SystemVerilog or Verilog-AMS port is connected to an incompatible SystemVerilog or Verilog-AMS port type, you can specify a converter module through the `+optconfigfile` option that will enable you to connect these types. VCS automatically inserts these converters at the ports where the data types mismatch.

Automatic insertion of converters happens after connectivity of the design is fully resolved. Based on the selected flow, the connectivity resolution step may include algorithms, such as discipline resolution and type coercion.

Converter Direction

The highconn and lowconn sides of a port along with the lowconn port direction decide the direction of a converter.

Table 66 Converter Direction

Highconn Type	Lowconn Type	Lowconn Direction	Converter Direction
logic	real	output	Unidirectional <code>real_to_logic</code>
logic	real	input	Unidirectional <code>logic_to_real</code>
real	logic	inout	Bidirectional <code>logic_real_bidir</code>

Use Model

To enable this feature, you must use `-xlrn enable_converter` elaboration option.

```
% vcs -sverilog -xlrn enable_converter  
+optconfigfile+converter.cfg top.v;
```

The converter modules are specified in a separate configuration file that is passed to VCS using the `+optconfigfile` option.

The converter configuration file has following entries:

- `converter <source_type> <dest_type> <converter_module> <named_parameter_overrides>;`

Where:

`source_type` specifies the type of the source signal.

`dest_type` specifies the type of the destination signal.

`converter_module` specifies the Verilog module that is either present in the predefined library provided as part of VCS or is provided by the user.

`named_parameter_override` specifies the parameter values for the inserted `converter_module` instances. These values are constants. For example,

```
converter logic real logic_to_real #( .Hi(5), .Low(0), .X(2.5), .Z(3));
```

The syntax implies that for all instances in the design where source is logic and destination is real, the converter module `logic_to_real` is inserted for value conversion.

- `ams_cm` command that enables hierarchical and port-based configuration of converters.
For example,

```
ams_cm vthHigh=2.1 inst=tb.DUT.MID1;
```

A detailed description of this command can be found in the “Analog Mixed Signal Features” chapter of the *VCS LCA Features Guide*.

If any converter needs to be inserted for user-defined types or nettypes, the type should be declared inside a package. For example,

```
package P;
  nettype logic logic_net;
endpackage
```

Configuration file:

```
converter P::logic_net real logic_to_real;
```

If the type is not a pre-defined SystemVerilog type and it is not specified as a package type as mentioned above, then VCS searches for the type definition in `snps_msv_netttype_pkg` package. If the type definition is not found, VCS generates an error.

Converters can have only two ports. If one port of a converter module is input, the other must be output. If one port of a converter module is inout, the other must be inout.

Converter modules cannot have any reference ports.

Usage Example

The following example shows a converter module that converts values from logic to real.

Example 245

```
module logic_to_real(output real realport, input logic logicport)
parameter Hi = 5;
parameter Low = 0;
parameter X = 2.5;
parameter Z = 3;
real treal;
assign real = treal;
always @(logicport) begin
if (logicport === 1'b1)
treal = Hi;
else if (logicport === 1'b0)
treal = Low;
else if (logicport === 1'bx)
treal = X;
else
treal = Z;
end
endmodule
```

For two-step flow:

```
%vcs -sverilog -xlrn enable_converter +optconfigfile+converter.cfg top.v;
```

For each instance in the design where port connection types do not match, VCS can select the converter module specified in the configuration file based on the type.

In case `-xlrn enable_converter` is specified but the converter is not provided, VCS selects a converter from the predefined set of converters. If such a converter with corresponding types is not available, VCS generates an error.

For three-step flow:

```
%vlogan -sverilog top.v;
%vcs -xlrn enable_converter top +optconfigfile+converter.cfg;
```

In three-step flow, VCS searches the user libraries for converter modules. If they are not found, VCS searches for converter modules in the libraries that are part of VCS distribution. The user-specified converter is always given precedence over other converters.

Predefined Converter Modules

The following table lists the predefined converter modules that VCS provides:

Source Type	Destination Type	Converter Module	Description
th_wire	logic	th_wire_to_logic	Conversion is done as described in section conversion_rule_for_th_wire_to_logic .
logic	th_wire	logic_to_th_wire	Conversion is done as described in section Conversion Rule for logic_to_th_wire .
logic	current_r	logic_to_current_r	Conversion is done as described in section Conversion Rule for logic_to_current_r/logic_to_real/logic_to_voltage_r .
logic	real	logic_to_real	
logic	voltage_r	logic_to_voltage_r	
logic	v_wire_4state	logic_to_v_wire_4state	Conversion is done as described in section Conversion Rule for logic_to_v_wire* .
logic	v_wire_avg	logic_to_v_wire_avg	
logic	v_wire_max	logic_to_v_wire_max	
logic	v_wire_min	logic_to_v_wire_min	
logic	v_wire_one	logic_to_v_wire_one	
logic	v_wire_sum	logic_to_v_wire_sum	
logic	wreal1driver	logic_to_wreal1driver	Conversion is done as described in section Conversion Rule for logic_to_wreal* .
logic	wreal4state	logic_to_wreal4state	
logic	wrealavg	logic_to_wrealavg	
logic	wrealmax	logic_to_wrealmax	
logic	wrealmin	logic_to_wrealmin	
logic	wrealsum	logic_to_wrealsum	

Source Type	Destination Type	Converter Module	Description
v_wire_4	logic	v_wire_4state_to_logic	Conversion is done as described in section Conversion Rule for v_wire*_to_logic .
v_wire_avg	logic	v_wire_avg_to_logic	
v_wire_max	logic	v_wire_max_to_logic	
v_wire_min	logic	v_wire_min_to_logic	
v_wire_one	logic	v_wire_one_to_logic	
v_wire_sum	logic	v_wire_sum_to_logic	
current_r	logic	current_r_to_logic	Conversion is done as described in section Conversion Rule for real_to_logic/voltage_r_to_logic/wreal*_to_logic/current_r_to_logic .
voltage_r	logic	voltage_r_to_logic	
real	logic	real_to_logic	
wreal1driver	logic	wreal1driver_to_logic	
wreal4state	logic	wreal4state_to_logic	
wrealavg	logic	wrealavg_to_logic	
wrealmax	logic	wrealmax_to_logic	
wrealmin	logic	wrealmin_to_logic	
wrealmin	wreal	wrealmin_to_wreal	Conversion is done as described in section Conversion Rule for wreal*_to_wreal*_type .
wrealsum	wreal	wrealsum_to_wreal	
wreal	wreal1driver	wreal_to_wreal1driver	
wreal	wreal4state	wreal_to_wreal4state	
wreal	wrealavg	wreal_to_wrealavg	
wreal	wrealmax	wreal_to_wrealmax	
wreal	wrealmin	wreal_to_wrealmin	
wreal	wrealsum	wreal_to_wrealsum	

Source Type	Destination Type	Converter Module	Description
wreal1driver	wreal	wreal1driver_to_wreal	
wreal4state	wreal	wreal4state_to_wreal	
Wrealsum	logic	wrealsum_to_logic	
Wrealmax	wreal	wrealmax_to_wreal	
Wrealavg	wreal	wrealavg_to_wreal	

Conversion Rules

This section describes the rules for automatic value conversion.

In all the conversion rules, `Hi`, `Lo`, `X`, and `Z` are parameters in the converter module with the following default values.

`Hi=5.0, Lo=0.0, X=`wrealXState, Z=`wrealZState`

Where:

`Hi` specifies the lower threshold for `logic '1'` or representation of `logic '1'` in real.

`Lo` specifies the representation of `logic '0'` in real.

`X` specifies the representation of `1'bx` in real.

`Z` specifies the representation of `HiZ` in real.

Users can override these values through the configuration file.

Conversion Rule for `wreal*_to_wreal*_type`

For all converters mentioned above, the value of the output variable is equal to the value of the input variable.

Conversion Rule for logic_to_current_r/logic_to_real/logic_to_voltage_r

The following list the conversion rules `logic_to_current_r` or `logic_to_real` or `logic_to_voltage_r`:

Table 67 Conversion Rule for logic_to_current_r or logic_to_real or logic_to_voltage_r

Input value (Source : logic l)	Output value (destination : current_r/real/voltage_r)
<code>l = 1'b0</code>	Lo
<code>l = 1'b1</code>	Hi
<code>l = 1'bx</code>	X
<code>l = 1'bz</code>	Z

Conversion Rule for logic_to_th_wire

The following list the conversion rules for `logic_to_th_wire`:

Table 68 Conversion Rule for logic_to_th_wire

Input value (Source : logic l)	Output value (destination : th_wire)
<code>l = 1'b0</code>	'{Lo,1.0,1'b1}
<code>l = 1'b1</code>	'{Hi,1.0,1'b1}
<code>l = 1'bx</code>	'{X,1.0,1'bx}
<code>l = 1'bz</code>	'{Z,1.0,1'bz}

Conversion Rule for logic_to_v_wire*

The following list the conversion rules for `logic_to_v_wire*`:

*Table 69 Conversion Rule for logic_to_v_wire**

Input value (Source : logic l)	Output value (destination : v_wire_4state/v_wire_avg/v_wire_max/v_wire_min/v_wire_on/v_wire_sum)
<code>l = 1'b0</code>	'{Lo,1'b1}

Table 69 Conversion Rule for logic_to_v_wire (Continued)*

Input value (Source : logic l)	Output value (destination : v_wire_4state/v_wire_avg/v_wire_max/v_wire_min/v_wire_on/v_wire_sum)
l = 1'b1	'{Hi,1'b1}
l = 1'bx	'{X,1'bx}
l = 1'bz	'{Z,1'bz}

Conversion Rule for logic_to_wreal*

The following list the conversion rules for `logic_to_wreal*`:

*Table 70 Conversion Rule for logic_to_wreal**

Input value (Source : logic l)	Output value (destination : v_wire_4state/v_wire_avg/v_wire_max/v_wire_min/v_wire_on/v_wire_sum)
l = 1'b0	Lo
l = 1'b1	Hi
l = 1'bx	X
l = 1'bz	Z

Conversion Rule for real_to_logic/voltage_r_to_logic/ wreal*_to_logic/current_r_to_logic

The following list the conversion rules for
`real_to_logic/voltage_r_to_logic/wreal*_to_logic/current_r_to_logic`:

*Table 71 Conversion Rule for real_to_logic/voltage_r_to_logic/
wreal*_to_logic/current_r_to_logic*

Input value(Source : real/voltage_r/wreal1driver/ wreal4state/wrealavg/wrealmax/wrealmin/wrealsum/curren t_r rl)	Output value (destination : logic)
rl >= Hi	1'b1
rl < Hi	1'b0
rl = X	1'bx

*Table 71 Conversion Rule for real_to_logic/voltage_r_to_logic/
wreal*_to_logic/current_r_to_logic (Continued)*

Input value(Source : real/voltage_r/wreal1driver/ wreal4state/wrealavg/wrealmax/wrealmin/wrealsum/curren t_r rl)	Output value (destination : logic)
rl = z	1'bz

Conversion Rule for th_wire_to_logic

The following list the conversion rules for th_wire_to_logic:

Table 72 Conversion Rule for th_wire_to_logic

Input value(Source : th_wire th)	Output value (destination : logic)
th = '?, ?, 1'bx	1'bx
th= '?, ?, 1'bz	1'bz
th= '?, ?, 1'b0	1'bz
th= '{x, ?, 1'b1}	1'bx
th= '{z, ?, 1'b1}	1'bz
th= '{(>=Hi), ?, 1'b1}	1'b1
th= '{(<Hi), ?, 1'b1}	1'b0

Conversion Rule for v_wire*_to_logic

The following list the conversion rules for v_wire*_to_logic:

*Table 73 Conversion Rule for v_wire*_to_logic*

Input value(Source : v_wire_4state/v_wire_avg/v_wire_max/v_wire_min/v_wire_ one/v_wire_sum th)	Output value (destination : logic)
th = '?, 1'bx	1'bx
th= '?, 1'bz	1'bz
th= '?, 1'b0	1'bz

*Table 73 Conversion Rule for v_wire*_to_logic (Continued)*

Input value(Source : v_wire_4state/v_wire_avg/v_wire_max/v_wire_min/v_wire_one/v_wire_sum th)	Output value (destination : logic)
th= '{X,1'b1}	1'bx
th= '{Z,1'b1}	1'bz
th= '{(>=Hi),1'b1}	1'b1
th= '{(<Hi),1'b1}	1'b0

Requirements

The feature has the following requirements:

- Converter modules are leaves in the design tree. They cannot have any instance of further modules.
- Converter modules cannot have any hierarchical references.
- Converters cannot use interconnects in their ports.
- Automatic insertion of converters within of VHDL scope is not supported.

Support for Wreal Nets

This feature describes the support provided by VCS for wreal nets and how you can migrate from the Verilog-AMS wreal flow to the SystemVerilog Used-Defined Nettype flow in the following sections:

- [Introduction](#)
- [Supported Features](#)
- [Use Model](#)
- [Migrating From wreal Flow to nettype Flow](#)

Introduction

There is increased usage of digital real numbers to develop analog or mixed-signal functional model, and to speed up the simulation performance using a digital simulator. This is useful to:

- Investigate the system characteristics, before starting transistor- level design.
- Verify connectivity of an entire system.
- Enable a top-down design methodology.

If the real valued model is entirely simulated by the digital simulator, it improves the simulation performance tremendously, and uses digital simulator and advanced digital methodology like testbench, assertion, and coverage to reduce the time required to perform intensive verification. However, there is a trade-off between performance and accuracy. If a SPICE model is available, you can swap a real valued model by SPICE model to check for more accuracy and finding circuit problem, then you can reuse the same testbench and may use both the digital top or SPICE top design style.

The digital methodology is used to develop real-valued behavioral model, using VHDL real number. VHDL language allows a user- defined resolution function, which in-turn allows a real-type or sub- type with multiple drivers. This real-type or sub-type with multiple drivers is powerful to develop the real-valued behavioral model.

Before the introduction of SystemVerilog User-Defined Nettypes, there was not a straightforward way to support multiple real-valued drivers in SystemVerilog.

Supported Features

VCS supports the following features:

- Resolution functions such as `default`, `min`, `max`, `avg`, `sum`, and `4state`.
- The wreal `z` and `x` states are represented using predefined macros ``wrealZState` and ``wrealXState`.
- Wreal and SV real connection.
- Array of wreal, including at (parameterized) port declaration.
- Wreal to SPICE and SPICE to wreal connection in mixed-signal verification.
- The `$stable_model` in digital domain using wreal.
- The INOUT wreal port.

Multiple Drivers and Resolution Function on Wreal Net

A resolution function is required when there are multiple drivers. VCS provides a built-in resolution function, which you can select using the `wreal <res_func>` compile-time option, where `res_func` can be either `res_def`, `res_sum`, `res_min`, `res_max`, `res_4state`, or `res_avg`.

- `res_def`: Single active driver only. Support for `x` and `z` state.
- `res_sum`: Resolved value is the sum of all the drivers value.
- `res_min`: Resolved value is the minimum value of all the drivers value.
- `res_max`: Resolved value is the maximum value of all the drivers value.
- `res_4state`: If there is more than one driver that is not `z`, the result is `x` unless all drivers have the same value. Otherwise, the result is the value that is not `z`.
- `res_avg`: Resolved value is the average value of all the drivers value.

Table 74 Example for Driver Resolution

DR#1	DR#2	default	sum	min	max	4state	avg
1.5	2.5	X	4.0	1.5	2.5	X	2
1.5	Z	1.5	1.5	1.5	1.5	1.5	1.5
1.5	X	X	X	X	X	X	X
Z	Z	Z	Z	Z	Z	Z	Z
1.5	1.5	X	3.0	1.5	1.5	1.5	1.5
X	X	X	X	X	X	X	X

Wreal X and Z states

Predefined macros are used to describe `x` (``wrealXState`) and `z` (``wrealZState`) states.

Example

```
module myMod(r);
parameter xval = 1.25;
parameter zval = 1.5;
inout r;
wreal r;
real r1;
reg clk;
initial clk = 0;

always #5 clk = ~clk;
assign r = r1;
```

```

always @(clk) begin
    If ( clk == 1'b1)
        r1 = 3.3;
    else
        r1 = `wrealZState;
end

always @(r) begin
    if (r == `wrealXState) begin
        $display($time,,,"%m: Wreal net r has value X");
        r1 = xval;
    end
end
endmodule;

```

Expressions

Expressions using wreal x and z states results in wreal x state (consistent with digital behavior). For a wreal net w1, the resulting expressions are as follows:

- $w1 (+, -, *, /) `wrealXState = `wrealXState$
If you take the value of wreal net w1 and perform one of the operations (+, -, *, /), when the second argument is `wrealXState, then the resultant value is `wrealXState.
- $w1 (+, -, *, /) `wrealZState = `wrealXState$
If you take the value of wreal net w1 and perform one of the operations (+, -, *, /), when the second argument is `wrealZState, then the resultant value is `wrealXState.

Arithmetic expressions involving four-state integers that have x or z bits and real numbers will also result in `wrealXState.

Operators

The following conditional expressions are supported on wreal:

- >, <, ==, ===, >=, <=
- Wreal and SV real numbers can be compared with signed and unsigned integers.
- The operations involving x or z values results in x, similar to integers.
- The ternary operator is supported.
- The logical operators && and || are supported, and are similar to integers.

Unsupported Operators

The following operators are not supported on wreal:

- Bitwise operators: `&`, `|`, `^`, `~`, `~&`, `~|`, `~^`
- Reduction operators: `&`, `|`, `^`, `~`, `~&`, `~|`, `~^`
- Shift operators: `>>`, `<<`, `>>>`, `<<<`

Initial Value of Wreal

The initial value of a wreal net is 0.0. Using the `-wreal init_val_z` option, the initial value of a wreal net can be set to `z`.

Use Model

- VCS allows wreal usage for pure digital simulation using Verilog or SystemVerilog with the `-realport` compile-time option.
- For mixed-signal verification using Verilog-AMS and the `-ams` option, the `-realport` option is default.
- The `-wreal <res_func>` compile-time option is required to:
 - Enable multiple drivers on wreal net
 - Choose the resolution function

```
% vcs -wreal res_def test.v
```

Migrating From wreal Flow to nettype Flow

With the wreal flow, you cannot define your own resolution functions to handle some special cases in a design. You do not have a mechanism to perform precision control in the resolution function to avoid infinite event loop due to some feedback loops in certain designs.

To overcome such restrictions of wreal, SystemVerilog introduced the concept of a User-Defined Nettype to model a net of any data type. The nettype is essentially a super set of wreal, and it provides a much more powerful and flexible methodology in real number modeling.

VCS supports the migration of wreal to nettype to perform real number modeling with the following capabilities:

- Support for equivalent nettypes for wreal resolution functions
- Support for real constants: `realX` and `realZ` in the predefined nettype package
- Allow ``wrealXState`/`wrealZState` in the parser with the `-sverilog` option enabled

- Interpret values represented by `wrealXState`/`wrealZState` as x and z for all real types
- Pass values of `wrealXState`/`wrealZState` to the VHDL boundary without truncation

Note:

It is mandatory to import the `snps_msv_nettype_pkg` package into the design.

Syntax: `import snps_msv_nettype_pkg::*;`

Nettypes and Their Corresponding Wreal Resolution Functions

[Table 75](#) lists the nettypes that you can use instead of the wreal resolution function:

Table 75 Wreal Nettypes and Their Corresponding Wreal Resolution Functions

Wreal Nettype	Corresponding Wreal Resolution Function	Description
wreal1driver	-wreal res_def	Single active driver only
wreal4state	-wreal 4state	Similar to <code>wreal1driver</code> except that multiple drivers are allowed if all of them are driving the same value
wrealavg	-wreal res_avg	Resolves to the average of all the driver values
wrealmin	-wreal res_min	Resolves to the least value of all the driver values
wrealmax	-wreal res_max	Resolves to the greatest value of all the driver values
wrealsum	-wreal res_sum	Resolves to the summation of all the driver values

Predefined Nettypes

The `voltage_r` and `current_r` predefined nettypes are available in the `snps_msv_nettype_pkg` nettype package. See [Table 76](#) for their corresponding wreal resolution function and description.

Table 76 Predefined Nettypes and Their Corresponding Wreal Resolution Functions

Predefined Nettype	Corresponding Wreal Resolution Function	Description
voltage_r	-wreal res_def	Single active driver only
current_r	-wreal res_sum	Resolves to the summation of all the driver values

Support for Real Constants (realZ/realX State)

You can use the following real constants available in the predefined `snps_msv_nettype_pkg` nettype package to replace ``wrealXState` and ``wrealZState`:

- `const real realX` is equivalent to ``wrealXState`
- `const real realZ` is equivalent to ``wrealZState`

You can use these real constants with any of the six wreal nettypes (defined in [Table 75](#)) in your design files.

Enabling ``wrealXState`/`wrealZState` in SystemVerilog Mode

To enable ``wrealXState`/`wrealZState` in the SystemVerilog mode, VCS allows ``wrealXState`/`wrealZState` in the parser with the `-sverilog` option when used in `vlogan` or `vcs` command line.

Interpreting ``wrealXState`/`wrealZState`

VCS interprets the values represented by ``wrealXState`/`wrealZState` as x and z for all real types and displays the ``wrealXState`/`wrealZState` during and after simulation. This includes all the display system task calls, such as `$display`, `$strobe`, and `$monitor`. This also includes UCLI output, DVE waveform display, and so on.

No Value Truncation at MX Boundary

The ``wrealXState`/`wrealZState` values are passed to the VHDL boundary without any truncation.

Support for Converting Wreal to Nettype

There is often usage of wreal in a dumped netlist. Wreal is not a type defined by SystemVerilog Language Reference Manual (LRM). Therefore, VCS needs either the `-ams` or the `-realport` option to be specified in the `vlogan` step to support this XLRM type. In the compile step, VCS expects you to specify the `-wreal res_xxx` option to define the resolution function that is to be used as a resolution of this wreal type.

This feature converts the wreal type to a nettype as per the resolution function specified in the VCS compile step. It maps the wreal net type and its resolution function to the equivalent SystemVerilog User-Defined Nettype specified in the `snpss_msv_nettype_pkg` package, which gets included and compiled automatically when the conversion option is specified. The exact equivalent mapping is specified in the [Migrating From wreal Flow to nettype Flow](#) section. Therefore, all compile steps treat wreal as nettype with the resolution function.

Use Model

To enable the feature, use the following compile-time option:

```
%vcs -adopt wreal_to_nettyp
```

However, you must also specify either the `-ams` or the `-realport` option in the vlogan step to allow wreal parsing.

Table 77 Wreal to Nettype Conversion

Verilog-AMS Flow	Global Wreal Resolution Function	Conversion Option	SystemVerilog User-Defined Nettype
Yes	None	Yes	<code>res_def</code>
No	Specified <code>res_xxx</code>	Yes	Equivalent to <code>res_xxx</code>
No	None	No	No conversion
No	Specified <code>res_xxx</code>	No	No Conversion

Limitations

The feature has the following limitation:

- The conversion of wreal to the equivalent SystemVerilog User-Defined Nettype happens only for the compiled files. It is not supported for the command files, such as the `optconfigfile` configuration file in which datatype converters insertion scenarios are specified

Support for Detail Discipline Resolution of Mixed Signals

In general, a mixed signal is a contiguous collection of nets, some with discrete disciplines and some with continuous disciplines. A continuous signal is a contiguous collection of nets where all the nets are in the continuous domain. Additionally, some of the nets can

have undeclared disciplines. Discipline resolution assigns disciplines and domains to those nets whose discipline is undeclared. This is done to:

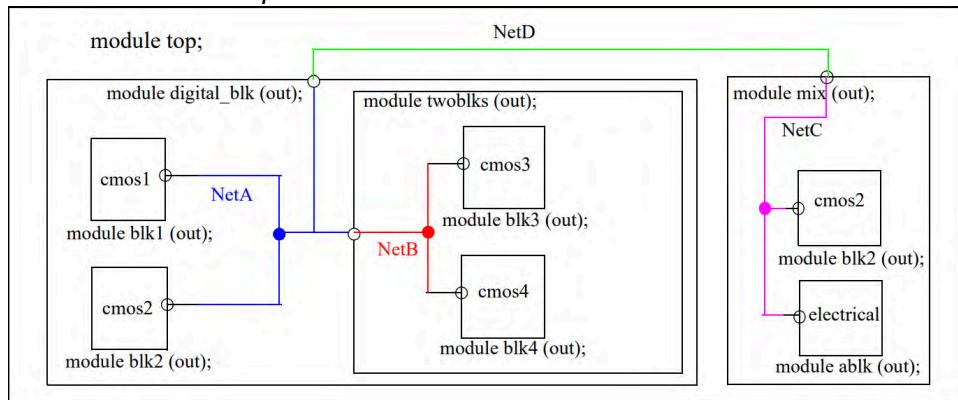
- Control automatic insertion of connect modules according to the rules embodied in connect statements.
- To ensure that the nets of all mixed signals and continuous signals have a known discipline and domain.

There are two modes of discipline resolution. VCS supports the basic mode by default. This feature extends the support of discipline resolution to the detail mode, which determines how known disciplines are used to resolve undeclared nets. For the entire design, undeclared nets are resolved at each level of the hierarchy where continuous (analog) has precedence over discrete (digital).

Detail Discipline Resolution Algorithm

In this mode, continuous disciplines propagate up and then back down to meet discrete disciplines. Discrete disciplines do not propagate through the hierarchy. This can result in more connect modules being inserted lower down into the discrete sections of the design hierarchy for added accuracy.

Figure 198 Detail Discipline Resolution Mode



In the above example, NetA, NetB, NetC, and NetD are undeclared interconnects. When you use the detail mode of discipline resolution in this example, it results in the following:

- **Continuous up:** NetC resolves to electrical based on continuous (electrical) winning over discrete (cmos2).
- **Continuous up:** NetD resolves to electrical based on continuous (electrical) winning over undeclared.

- **Continuous down:** NetA resolves to electrical based on continuous (electrical) winning over undeclared.
- **Continuous down:** NetB resolves to electrical based on continuous (electrical) winning over undeclared.

The detail discipline resolution algorithm is described as follows:

1. Elaborate the design.

After this step, every port in the design has both its upper (actual) connection and lower (formal) connection defined.

2. Apply all in-context node and signal declarations.

For example, `electrical sig;` makes all instances of `sig` electrical, unless they have been overridden by an out-of-context declaration.

3. Apply all out-of-context node and signal declarations.

For example, `electrical top.middle.bottom.sig;` overrides any discipline that may be declared for `sig` in the module where `sig` was declared, and it is applied to the `sig` that is in the `top.middle.bottom` hierarchy.

More than one conflicting in-context discipline declaration or more than one conflicting out-of-context discipline declaration for the same hierarchical segment of a signal is an error. In this case, conflicting simply means an attempt to declare more than one discipline regardless of whether the disciplines are compatible or not.

4. Traverse each signal hierarchically (bottom-up) when a net is encountered that has still not been assigned a discipline:

- a. It is determined whether the net is analog or digital. Any net that is used in digital behavioral code is considered digital.
- b. If the net has not yet been assigned a discipline, examine all the child nets of that net and construct a list of all continuous domain disciplines of these child nets :
 - If there are no disciplines in the list, then no discipline is assigned to this net at this (bottom-up) stage.
 - If there is only a single continuous domain discipline in the list, the net is of that discipline.
 - If there is more than one continuous domain discipline in the list, and the contents of the list match the discipline list of a connect resolution statement, then the discipline of this net is the resolved discipline given by this statement. If such matching connect resolution statement is not found, then that results in an error.

5. Traverse each signal hierarchically (top-down) when a net is encountered that still has not been assigned a discipline or that has been assigned a digital domain from step 4.
 - a. It is re-determined whether the net is analog or digital. Any net that is used in digital behavioral code is considered digital.
 - b. If the net has not yet been assigned a discipline, examine all the parent nets of that net, and construct a list of all continuous domain disciplines of these parent nets:
 - If there are no disciplines in the list, then assign the default discrete discipline to this net. The default discrete discipline may come from the '`default_discipline`' directive that is applicable to this net. If such directive is not found, then consider the discrete discipline specified through the command-line switch '`-ams_discipline <discrete-discipline>`'.
 - If there is only a single continuous domain discipline in the list, the net is of that discipline.
 - If there is more than one continuous domain discipline in the list, and the contents of the list match the discipline list of a connect resolution statement, then the discipline of this net is the resolved discipline given by this statement. If such matching connect resolution statement is not found, then that results in an error.

At this point, connection module selection and insertion can be performed.

Use Model

To use the detail discipline resolution algorithm, use the `-ams_dresolution` option with VCS executable during compilation of the design. This option is not applicable to a source files analysis step with `vlog` or `vhdlan`.

Usage Example

The following example illustrates the detail discipline resolution algorithm:

test.vams

```
`timescale 1ns/10ps
`include "disciplines.vams"

module top;
  wire w1;

  mid1 MID1(w1);
  mide MIDE(w1);

  initial begin
```

```

      #100 $finish;
    end
endmodule

module mid1 (w1);
  input w1;
  bot BOT(w1);
endmodule

module mide (we);
  input we;
  anabote ANABOTE(we);
endmodule

module anabote(we);
  input we;
  electrical we;

  analog begin
    V(we) <+ 5.0;
  end
endmodule

module bot (wbot);
  output wbot;
endmodule

cm.vams

connectmodule l2e (din, aout);

  input din;
  logical din;
  output aout;
  electrical aout;

  parameter real v_tol = 100u from (0:inf);
  parameter real t_rise = 10p from (0:inf);
  parameter real t_fall = 10p from (0:inf);
  parameter real t_tol = (t_rise+t_fall)/20 from (0:inf);
  parameter real tdelay = 0 from [0:inf];
  parameter real rout = 200 from (0:inf);
  real vout;
  analog begin
    vout = transition(din, tdelay, t_rise, t_fall, t_tol) / 2.0;
    I(aout) <+ (V(aout) - vout) / rout;
  end
endmodule

connectrules mixedsignal;
  connect l2e;
endconnectrules

```

test.init

```
choose xa ;
```

To run the example, use the following command:

```
% vcs -sverilog -ams_dresolution -ams test.vams cm.vams - ad=test.init  
-ams_discipline logical -full164
```

In the above example, the electrical discipline of the net `top/MIDE/ ANABOTE/we` is propagated up and then down according to the detail discipline resolution algorithm. As a result, all nets with undeclared discipline get electrical discipline after the resolution.

Limitations

This feature has the following limitation:

- Out-of-context discipline declaration is not yet supported. For example, the declaration `electrical top.middle.bottom.sig ;` is not yet supported.

34

Integrating VCS with Specman

The VCS ESI Adapter integrates VCS with the Specman Elite. This chapter describes how to prepare a stand-alone VHDL/Verilog design or mixed VHDL/Verilog design for use with the ESI interface. See the *Specman Elite User Guide* for further information.

VCS has two ESI adapters, one for Verilog and the other for VHDL. You can use both the adapters together for mixed HDL simulation. VHDL adapter is implemented as a VHPI foreign architecture, while the Verilog adapter is implemented as a Verilog PLI application.

VHDL adapter is called as `specman_vcsmx.vhd`. This file is generated using the `specman` command as explained when using VCS H-2013.06 release with Specman 12.2 version. Generate the VHDL stub file using the following `specman` command:

```
% specman -c "load xor_verify.e ; write stubs -vcsmx_vhdl"
```

Analyze the VHDL stub file using the following command:

```
% vhdlan -nc specman_vcsmx.vhd
```

This chapter includes the following topics:

- [Type Support](#)
- [Usage Flow](#)
- [Using specrun and specview](#)
- [Version Checker for Specman](#)
- [Precedence Order](#)

Type Support

The VCS ESI adapter supports the following VHDL types:

- Predefined types
 - `bit`
 - Boolean
 - `std_logic/std_ulogic`

- character
- array
- User-defined enum types
- VHDL memory
- in/out/inout/buffer ports
- Access to elements of the following composite types supported:
 - Access to individual elements of any of the supported scalar types
 - Predefined types based on any of the supported scalar types such as string, bit_vector, integer, and so on.

Note:

Calling VHDL procedure or functions through e code is not supported.

The VCS ESI adapter supports the following Verilog Types:

- nets
- wires
- registers
- integers
- array of registers (verilog memory)

Other Verilog support:

- Verilog macros
- Verilog tasks
- Verilog functions
- Verilog events
- in/out/inout ports

Usage Flow

The Specman usage model for VCS depends upon whether the e code can access both VHDL and Verilog, or just one language. If the e code can access just one language, then you do not have to specify the unused part.

This section explains how to integrate Specman with VCS flow.

Setting Up The Environment

To set up the environment to run Specman with VCS:

- Set your `VCS_HOME` and `VRST_HOME` environment variables:

```
% setenv VCS_HOME [vcs_installation_path]
% set path = ($VCS_HOME/bin $path)
% setenv VRST_HOME [specman installation]
```

- Source your `env.csh` file for Specman:

```
% source ${VRST_HOME}/env.csh
```

For 64-bit simulation, source your `env.csh` file as shown below:

```
% source ${VRST_HOME}/env.csh -64bit
```

- Source the `environ.csh` file for VCS:

```
% source $VCS_HOME/bin/environ.csh
```

- Set your environment for the VCS Specman ESI adapter:

```
% setenv SPECMAN_VCSMX_VHDL_ADAPTER
${VCS_HOME}/${ARCH}/lib/libvhdl_sn_adapter.so
```

Specman e code accessing VHDL only

Instantiate `SPECMAN_REFERENCE` in the top-level VHDL code as follows:

```
component comspec
end component;
for all: comspec use entity work.SPECMAN_REFERENCE(arch);
I: comspec;
```

Note:

In a Verilog-top design, instantiate `SPECMAN_REFERENCE` in one of the top-level VHDL files underneath the Verilog-top code.

Analyze Verilog design files as shown below:

```
% vlogan [vlogan_options] -f Verilog_filename_list
```

Analyze the VHDL stub file and then VHDL design files as shown below:

```
% vhdlan specman_vcsmx.vhd
% vhdlan [vhdlan_options] file1.vhd file2.vhd
```

Elaborate the design as given in the following table:

Elaboration Mode		Commands	Generated Executable
Compile	Execution with -o	"% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" -o <exe_name> <top_e_file>.e "	vcs_<exe_name>
	Execution without -o	"% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" <top_e_file>.e "	vcs_<top_e_file>
Loaded	Execution with -o	"% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" -o <exe_name>"	<exe_name>
	Execution without -o	"% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" "	vcs_specman

Simulate the design as given below:

- In Compiled mode:

```
% vcs_simv -ucli [simv_options]
ucli% sn "test"
ucli% run
ucli% quit
```

Note:

Notice the use of the -o option with this script in compile mode to change the name of the executable generated to vcs_simv from the default name given by the script which is vcs_<top_e_file>.

- In Loaded mode:

```
% simv -ucli [simv_options]
ucli% sn "load <top_e_file>; test"
```

```
ucli% run
ucli% quit
```

Note:

Notice the use of the `-o` option with this script in loaded mode to change the name of the executable generated to `simv` from the default name given by the script which is `vcs_specman`.

Specman e Code Accessing Verilog Only

Create the Verilog stub file `specman.v` and analyze/compile all Verilog files including `specman.v` as shown below:

Compile step:

```
% specman -c "load [top_e_file]; write stubs -verilog;"
% vlog [vlog_options] -f Verilog_filename_list specman.v
```

Analyze all VHDL design files as shown below:

```
% vhdlan [vhdlan_options] file1.vhd file2.vhd
```

Elaborate/Compile the design as given in the following table:

Elaboration Mode		Commands	Generated Executable
Compile	Execution with <code>-o</code>	<pre>% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" -o <exe_name> <top_e_file>.e "</pre>	<code>vcs_<exe_name></code>
	Execution without <code>-o</code>	<pre>% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" <top_e_file>.e "</pre>	<code>vcs_<top_e_file></code>
Loaded	Execution with <code>-o</code>	<pre>% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" -o <exe_name>"</pre>	<code><exe_name></code>

Elaboration Mode		Commands	Generated Executable
	Execution without -o	"% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" "	vcs_specman

Simulate the design as given below:

- In compiled mode:

```
% vcs_simv -ucli [simv_options]
ucli> sn "test"
ucli> run
ucli> quit
```

Note:

Notice the use of the `-o` option with this script in compile mode to change the name of the executable generated to `vcs_simv` from the default name given by the script which is `vcs_<top_e_file>`.

- In loaded mode:

```
% simv -ucli [simv_options]
ucli% sn "load <top_e_file>; test"
ucli% run
ucli% quit
```

Note:

Notice the use of the `-o` option with this script in loaded mode to change the name of the executable generated to `simv` from the default name given by the script which is `vcs_specman`.

Specman e code accessing both VHDL and Verilog

Instantiate `SPECMAN_REFERENCE` in the top-level VHDL code as follows:

```
component comspec
end component;
for all: comspec use entity work.SPECMAN_REFERENCE(arch);
I: comspec;
```

Note:

In a Verilog-top design, instantiate `SPECMAN_REFERENCE` in one of the top-level VHDL files underneath the Verilog-top code.

Create the Verilog stub file `specman.v` and analyze all Verilog files including `specman.v` as shown below:

```
% specman -c "load [top_e_file]; write stubs -verilog;"  
% vlog [vlog_options] -f Verilog_filename_list specman.v
```

Create the VHDL stub file `specman_vcsmx.vhd` and analyze all VHDL files including `specman_vcsmx.vhd` as shown below:

```
% specman -c "load xor_verify.e ; write stubs -vcsmx_vhdl"
```

Analyze the VHDL stub file and then VHDL design files as shown below:

```
% vhdlan specman_vcsmx.vhd  
% vhdlan [vhdlan_options] file1.vhd file2.vhd
```

Elaborate the design as given in the following table:

Elaboration Mode		Commands	Generated Executable
Compile	Execution with -o	"% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" -o <exe_name> <top_e_file>.e "	vcs_<exe_name>
	Execution without -o	"% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" <top_e_file>.e "	vcs_<top_e_file>
Loaded	Execution with -o	"% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" -o <exe_name>"	<exe_name>
	Execution without -o	"% sn_compile.sh -sim vcs \ -sim_flags "[compile-time_options] \ -debug_access top_cfg/entity/module" "	vcs_specman

Simulate the design as given below:

- In Compiled mode:

```
% vcs_simv -ucli [simv_options]
# sn "test"
# run
# quit
```

Note:

Notice the use of the `-o` option with this script in compile mode to change the name of the executable generated to `vcs_simv` from the default name given by the script which is `vcs_<top_e_file>`.

- In Loaded mode:

```
% simv -ucli [simv_options]
# sn "load <top_e_file>; test"
# run
# quit
```

Note:

Notice the use of the `-o` option with this script in loaded mode to change the name of the executable generated to `simv` from the default name given by the script which is `vcs_specman`.

Guidelines for Specifying HDL Path or Tick Access with VCS-Specman Interface

The guidelines to specify HDL path or tick access with VCS-Specman interface are as follows:

- You cannot mix `[]` and `("()")` in a single tick access or HDL path.
- HDL path or tick access notation should use `[]` on `e` side through VHDL generate. If you do not use `[]`, an adapter error is generated, to specify that the signal is not found. Apparently, `()` conflicts with the computed names in `e` code.
- Specman generates an error, if you use `("()")` in HDL path.
- In the tick access notation, you must use `[]` or `("()")`, instead of `()`. Apparently, `()` conflicts with the computed names in `e` code.
- You cannot use `:`, as a starting delimiter in the absolute HDL path in `e` code. Example:
`~:test_top"m1.b`

Using specrun and specview

VCS allows you to use the following Specman utilities to simulate your design:

- specrun
- specview

`specrun` invokes Specman in batch mode, while `specview` invokes the Specman GUI. The usage model is shown below:

Using specrun

- In Compiled Mode:

```
% specrun -p "test -seed=1;" simv [simv_options]
```

- In Loaded Mode:

```
% specrun -p "load [top_e_file]; test -seed=1;" \
simv [simv_options]
```

Example 246 To Invoke UCLI Using specrun

The following command invokes the simulation in UCLI mode:

```
% specrun -p "test -seed=1;" simv -ucli -i include.cmd
```

Using specview

Set the environment variable `SPECMAN_OUTPUT_TO_TTY` as shown below:

```
% setenv SPECMAN_OUTPUT_TO_TTY 1
```

- In Compiled Mode:

```
% specview -p "test -seed=1;" -sio simv -ucli
```

- In Loaded Mode:

```
% specview -p "load [top_e_file]; test -seed=1;" \
-sio simv -ucli
```

You can also specify VCS runtime options with `specview` or `specrun`.

Example 247 To Invoke UCLI Using specview

You can use `-ucli` with `specview` to invoke simulation in UCLI mode.

```
% specview -p "test -seed=1;" -sio simv -ucli
```

Version Checker for Specman

This section describes how to check the compatibility version of Specman with VCS. If non-compatible version of Specman is used, then VCS generates a warning message at compile-time/elaboration-time.

Use Model

- **Two-step Flow**

Through Command-line Options

```
% vcs +warn=V2V_CHECK_SPECMAN
```

To convert warning to error:

```
% vcs -error=V2V_CHECK_SPECMAN
```

Enabling at Runtime:

```
%simv +warn=V2V_CHECK_SPECMAN
```

You can use the `+warn=noV2V_CHECK_SPECMAN` option to turn off the warning message. In this option, `no` specifies disabling warning messages.

- **Three-step Flow**

Through command-line options:

```
% vlogan
```

```
% vhdlan
```

```
% vcs +warn=V2V_CHECK_SPECMAN
```

```
%simv +warn=V2V_CHECK_SPECMAN
```

To convert warning to error:

```
% vcs -error=V2V_CHECK_SPECMAN
```

You can use the `+warn=nov2V_CHECK_SPECMAN` option to turn off the warning message. In this option, `no` specifies disabling warning messages.

- Through `synopsys_sim.setup` file for VCS flow:

```
V2V_CHECK_SPECMAN=TRUE/FALSE
```

- Through new environment variable for VCS flow:

```
% setenv V2V_CHECK_SPECMAN TRUE/FALSE
```

Precedence Order

1. Command-line
2. Setup file
3. Environment variable

In VCS flow, command-line will have the highest priority compared to setup file and environment variable. Also, runtime enabling is automatically done, when enabled using environment variable or setup file.

35

Integrating VCS with Denali

Denali is a third-party Memory Modeler - Advanced Verification (MMAV) tool, which you can integrate with VCS using a set of APIs. Denali provides a complete solution for memory modeling and system verification. It automatically monitors all the timing and protocol requirements specified by the memory vendor.

Setting Up Denali Environment for VCS

To use Denali with VCS, set your Denali environment using the following commands:

```
% setenv DENALI [installation_path_of_DENALI]
% setenv LM_LICENSE_FILE [Denali_license]:$LM_LICENSE_FILE
% setenv LD_LIBRARY_PATH $DENALI/vhci:$LD_LIBRARY_PATH
```

Integrating Denali with VCS

The generic functionality of various memory architectures are captured in a set of highly-optimized C models. The vendor-specific features and the timing for any particular memory device are defined within the specification of memory architecture (SOMA) file. After the Denali model objects are linked into the simulation environment, modeling any type of memory is as simple as referencing the appropriate SOMA file for that particular memory device.

To access a particular SOMA file, include the following declaration in the source code:

For VHDL portions of designs:

```
GENERIC (
  memory_spec: string := soma_file_path;
  init_file: string := "";
);
```

For Verilog portions of designs:

```
parameter memory_spec = soma_file_path;
parameter init_file = "";
```

Note:

memory_spec and init_file are keywords.

Use Model

Denali provides you both Verilog and VHDL memory models. However, for mixed HDL designs, Synopsys recommends you to use either Verilog or VHDL memory model for the whole design. The use model does not allow mixing of PLI and VHPI calls.

This section describes the following topics:

- Use Model for VHDL Memory Models
 - Use Model for Verilog Memory Models
 - Execute Denali Commands at UCLI Prompt
-

Use Model for VHDL Memory Models

The VHDL memory models must be integrated with VCS using VHPI calls in the VHDL design code as shown in the following command:

```
attribute foreign of [architecture_name]: architecture is
    "vhpi:[library_name]:[elaboration_function_name]:
     [initialisation_function_name]:[model_name]";
```

For example,

```
attribute foreign of behavior: architecture is
    "vhpi:denvhpi:flashElabVHPI:flashInitVHPI:mobilesdram";
```

You can use VHDL memory models with the following types of design topologies:

- VHDL DUT and VHDL Testbench
- VHDL DUT and Verilog Testbench
- Verilog DUT and VHDL Testbench

The use model is as follows:

Analysis

```
% vlogan [vlogan_options] file2.v file3.v
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd \
  [memory_model.vhd] [memory_wrapper.vhd]
```

Elaboration

```
% vcs [vcs_options] top_entity/module/config
```

Simulation

```
% simv [simv_options]
```

Use Model for Verilog Memory Models

You can integrate Verilog memory models with VCS using PLIs. To use Verilog memory models, specify the `pli.tab` file and `denverlib.o` during compilation/elaboration.

You can use Verilog memory models with the following types of design topologies:

- Verilog DUT and Verilog Testbench
- VHDL DUT and Verilog Testbench
- Verilog DUT and VHDL Testbench

The use model is as follows:

Two-Step Flow

Compilation

```
% vcs -debug_access+f [vcs_options] verilog_filelist\  
-P $DENALI/verilog/pli.tab $DENALI/verilog/denverlib.o
```

Note:

To compile the design in 64-bit mode, use the `-lpthread` option.

Simulation

```
% simv [simv_options]
```

Three-Step Flow

Analysis

```
% vlogan [vlogan_options] file2.v file3.v \  
[memory_model.v] [memory_wrapper.v]  
  
% vhdlan [vhdlan_options] file3.vhd file2.vhd file1.vhd
```

Elaboration

```
% vcs -debug_access+f [vcs_options] top_entity/module/config -P  
$DENALI/verilog/pli.tab $DENALI/verilog/denverlib.o
```

Note:

To elaborate the design in 64-bit mode, use the `-lpthread` option.

Simulation

```
% simv [simv_options]
```

Execute Denali Commands at UCLI Prompt

VCS allows you to execute Denali commands at the UCLI prompt.

For example,

```
% simv -ucli  
ucli% mmload :top:I_dut:I_denali_model data_file
```

This UCLI command loads Denali memory in the `I_denali_model` instance with the data specified in the `data_file`.

For more information on invoking UCLI, see [Using UCLI](#).

36

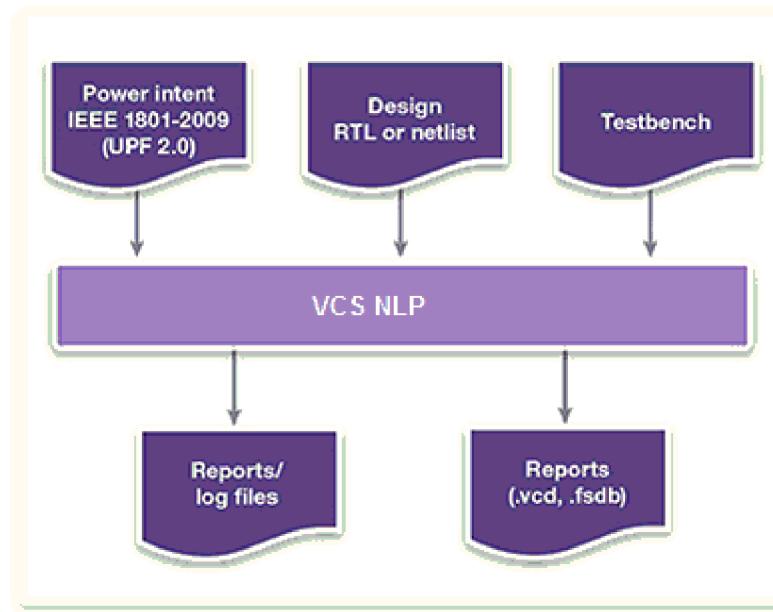
Integrating VCS With Native Low Power (NLP)

The VCS NLP add-on for VCS enables you to specify the UPF based power-intent of your design directly to VCS and generate a simulation model, which contains all power-objects directly instrumented in it.

VCS NLP equips VCS to natively perform voltage-level aware simulation with a complete understanding of the UPF-defined power network, including at RTL prior to implementation flows. This uniquely allows engineers to comprehensively verify correct behavior of designs that use advanced voltage control techniques for power management, and catch potentially design-killing low power bugs very early in the design process.

VCS NLP equips VCS to read this UPF, model the entire power network described in the UPF, and accurately understand the low power policies and voltage events. VCS in native low power mode generates a log file and an error and warnings report for all violations related to multi-voltage checks, as illustrated in [Figure 199](#).

Figure 199 VCS NLP Low Power Simulation Flow



For more details about VCS NLP add-on for VCS, refer to the [VCS Native Low Power \(NLP\) User Guide](#).

37

Unified UVM Library for VCS and Verdi

The unified UVM library integrates the instrumented UVM libraries available in VCS and Verdi. With the introduction of the unified UVM library, VCS and Verdi transaction recorder and message catcher coexist and are compiled together. You can directly use the Unified UVM library with Verdi recording mechanism during simulation and for debugging with Verdi. You do not need to set `VCS_UVM_HOME` pointing to `VERDI_HOME` for Verdi transactions.

Single compilation, UUM and UVM-VMM interoperability flows are supported in the unified UVM library. The unified UVM library can also be qualified and validated using Synopsys VIPs.

The UVM libraries are available in the following location:

- `$VCS_HOME/etc/uvm-1.1`: The unified UVM 1.1d library.
 - `$VCS_HOME/etc/uvm-1.2`: The unified UVM 1.2 library.
 - `$VCS_HOME/etc/uvm`: This path is symbolically linked to the `$VCS_HOME/etc/uvm-1.1` directory.
-

Transaction/Message Recording in Verdi with VCS

The following sections describe how to use the unified UVM library with Verdi transaction recorder and message catcher for the VCS simulator:

- [Compilation](#)
 - [Simulation](#)
-

Compilation

The compile flow is same for any GUI transaction recording. It is not required to point `VCS_UVM_HOME` to UVM library in Verdi installation and recompile.

Enabling FSDB Transaction Recording

Unified UVM library shipped with VCS has additional features that allow you to take advantage of FSDB transaction recording and Verdi transaction debugging capabilities.

For FSDB transaction recording, set *VERDI_HOME* and *VCS_HOME* environment variables. With the `-debug_access+tr` option, and NOVAS tab and PLI files, the Verdi transaction recorder is compiled.

You can enable the FSDB dumper using the following command:

```
% vcs -sverilog -ntb_opts uvm -debug_access
```

Recommended Use Model for FSDB Transaction Dumping

To enable FSDB transaction recorder with unified UVM library, it is recommended to use the `-debug_access` option, as follows:

```
% vcs -sverilog -debug_access+all -ntb_opts uvm-1.1 [compile_options]
```

Note:

- You must use the `-ntb_opts uvm-1.2` option for UVM-1.2 code.
- You must set *VERDI_HOME* and *VCS_HOME* environment variables.

To compile your UVM-1.1d or UVM-1.2 code, no extra compile-time option is needed. VCS transaction recorder, Verdi settings and recorder, `novas.tab` and `pli.a` files of FSDB dumper are automatically included.

Simulation

The following section describe how to perform simulation using the unified UVM library.

Dumping Transactions or Messages in Verdi Flow

Add the following runtime options to enable Verdi transaction recorder and message catcher:

- `+UVM_VERDI_TRACE=<Argument>`

Enables the Verdi flow when added during simulation.

You can use any of the following values as an input to the `<Argument>` parameter:

`UVM_AWARE | RAL | TLM | MSG | HIER | PRINT`

For more details, see the Verdi Application Note - *New Transaction Debug Platform in Verdi Application Note*.

- `+UVM_TR_RECORD`

Enables Verdi transaction recorder.

- `+UVM_LOG_RECORD`

Enables Verdi message catcher.

- +UVM_VPD_RECORD

Enables transaction dumping to VPD file.

Note:

By default, transaction dumping happens in the FSDB file. Using the +UVM_VPD_RECORD option, it happens in the VPD file.

For example,

```
%> ./simv +UVM_VERDI_TRACE +UVM_TR_RECORD \ +UVM_LOG_RECORD
```

38

Debugging with Verdi

This chapter describes debugging your designs with Verdi in the following sections:

- [Introduction](#)
 - [Generating Verdi KDB](#)
 - [Dumping FSDB File for Various Flows](#)
 - [Interactive and Post-Processing Debug](#)
-

Introduction

In a complex design, it is difficult to find the bugs associated with the logic at the HDL or testbench level. The process of debugging includes locating the design logic that is associated with an error, isolating the cause, and understanding how the design is supposed to behave and why it is not behaving that way. Design and verification engineers need sophisticated tools to find errors in the data produced by the simulator.

Verdi debug tool enables comprehensive debug for all design and verification flows. It includes powerful technology that helps you to comprehend complex and unfamiliar design behavior, and automate difficult and tedious debug processes.

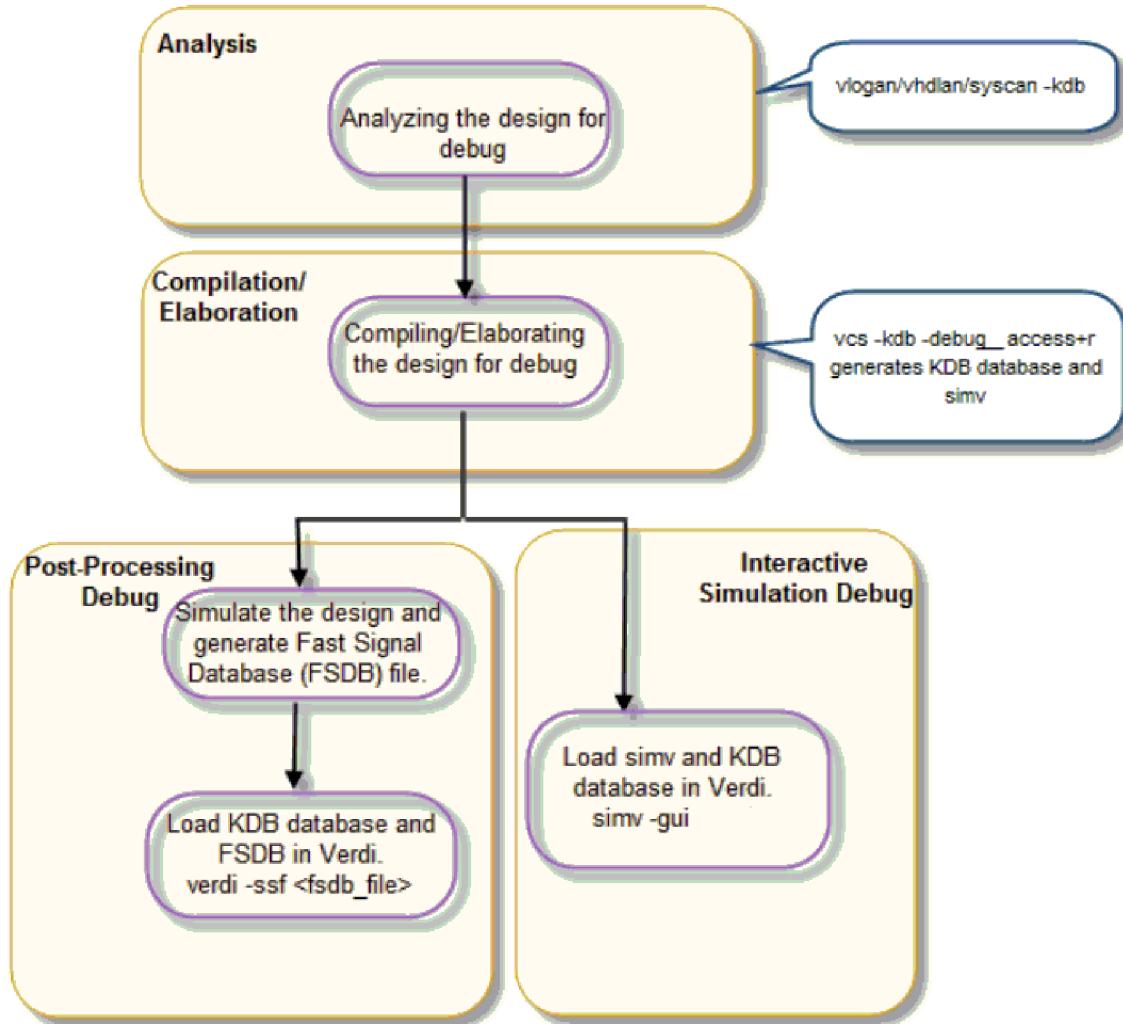
In addition to the standard features of a source code browser, schematics, waveforms, state machine diagrams, and waveform comparison, the Verdi tool includes advanced features for automatic tracing of signal activity using temporal flow views, assertion-based debug, and debug and analysis of transaction and message data.

You can debug your simulation using Verdi in the following modes:

- Interactive Simulation Debug Mode: Allows you to debug your design interactively.
- Post-Processing Debug Mode: Allows you to debug your design using a database.

The following flowchart shows the Verdi debug flow:

Figure 200 Verdi Debug Flow



Generating Verdi KDB

Verdi Knowledge Database (KDB) is supported in both VCS two-step and three-step flows. In the VCS two-step flow, add the `-kdb` option to the command line to generate Verdi Knowledge Database (KDB). In case of VCS three-step flow, add the `-kdb` option in all the `vlogan/vhdlan/vcs` command lines.

When you specify the `-kdb` option, Verdi creates the KDB and dumps the design into the libraries specified in the `synopsys_sim.setupfile`.

For example,

```
// Compile the design using VCS and generate both VCS database and Verdi
KDB //  
  
// -kdb in VCS two-step flow  
  
%> vcs -kdb <compile_options> <source files>  
  
// -kdb in VCS three-step flow  
  
% vlogan -kdb <vlogan_options> <source files>  
% vhdlan -kdb <vhdlan_options> <source files>  
% vcs -kdb <top_name>
```

To generate only the Verdi KDB and skip the simulation database generation, specify the following argument with the `-kdb` option:

`-kdb=only`

Generates only the Verdi KDB that is required for both post-process and interactive simulation debug with Verdi.

In the VCS two-step flow, this option does not generate the VCS compile data/executable and does not disturb the existing VCS compile data/executables.

Following is the use model in the VCS two-step flow:

```
% vcs -kdb=only <compile_options> <source files>
```

In the VCS three-step flow, `vlogan/vhdlan` does not support the `-kdb=only` option, it is recommended to use the `-kdb` option instead.

Following is the use model in the VCS three-step flow:

```
% vlogan -kdb <vlogan_options> <source files>
% vhdlan -kdb <vhdlan_options> <source files>
% vcs -kdb=only <top_name> <compile_options>
```

Reading Compiled Design with Verdi

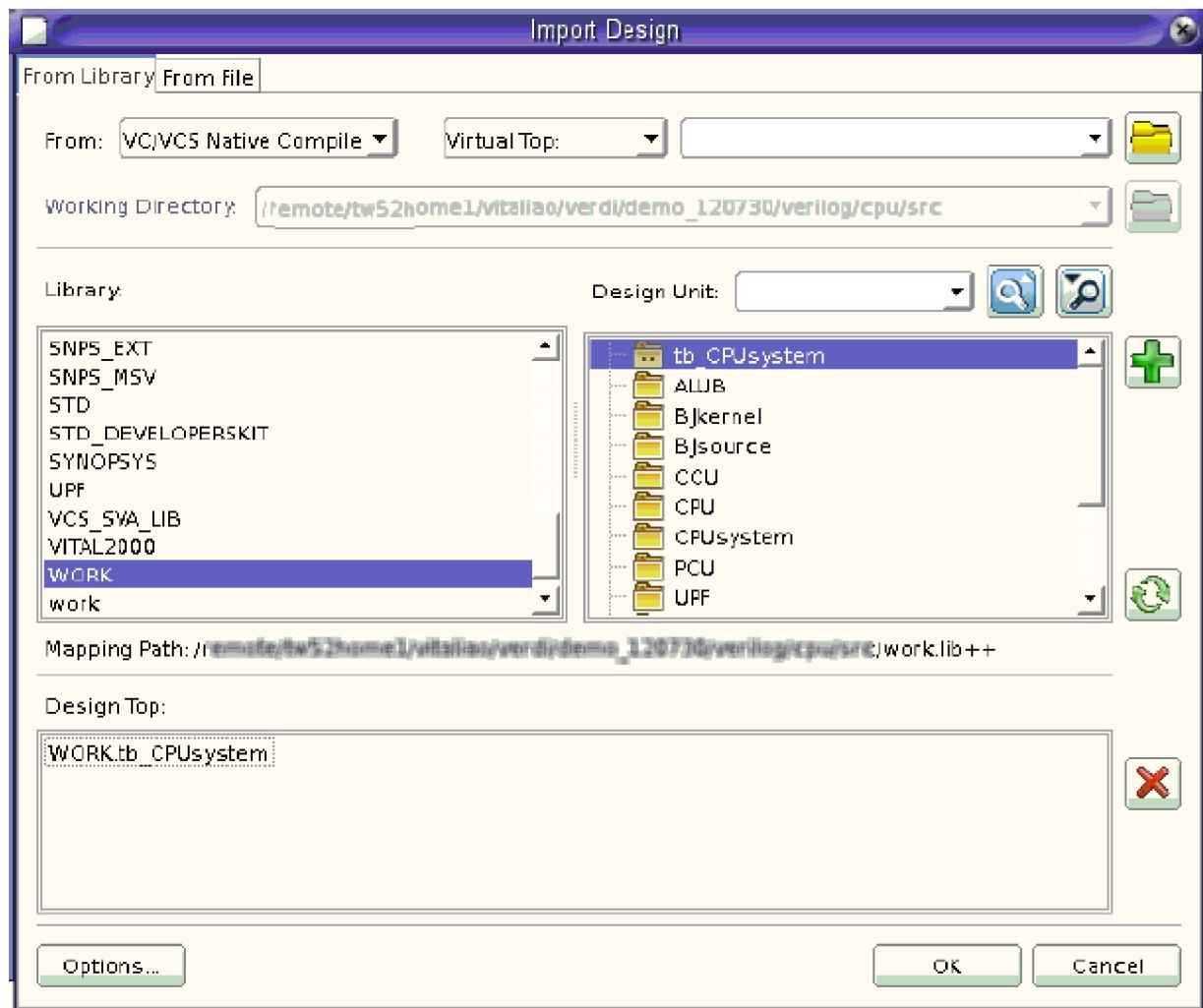
To read a compiled design, add the `-simflow` option to the Verdi command line. This imports the KDB and enables Verdi and its utilities to use the library mapping from the `synopsys_sim.setup` file. It is also used to import the design from the KDB library paths.

You can perform the same operations through the Verdi GUI as follows:

1. Click **File > Import Design** option.
2. In the **Import Design** form, select the **From Library** tab.

3. In the **From** field, select the **VC/VCS Native Compile** option, as shown in [Figure 201](#).

Figure 201 Import Design Form



You can also add the `-dbdir <path>` option to the Verdi command line to ensure that VCS and Verdi use the same data from the `synopsys_sim.setup` file. The `<path>` argument points to the library directory from where VCS is compiled. Use this option if you want to invoke Verdi from a working directory that is different from the VCS working directory.

You can also use the `-top` option with the `-dbdir` option to specify the top module in the specified library directory. For example,

```
%> verdi -simflow -dbdir [<path>] -top [<top module>]
```

If the `-top` option is not specified, the design top is used by default.

Example

Consider the following testcase, top.v:

```
module top;
    reg a,b;
    wire c;
    reg [0:1] mem1 [0:10];
    dut d1(a,b,c);
    initial begin
        a=0; b=0;
        #1 a=0; b=1;
        #1 a=1; b=0;
        #1 a=1; b=1;
        #1 $finish;
    end
endmodule

module dut(input a,b,output c);
    assign c=a&b;
    count cnt(a,b,c);
endmodule

module count(input a,b,output c);
    reg d;
    initial $monitor("A=%0d,B=%0d,C=%d,D=%0d",a,b,c,d);
endmodule
```

Compiling Designs

VCS Two-step Flow Compilation:

```
% vcs -debug_access+r -sverilog -kdb top.v
```

VCS Three-step Flow Compilation:

```
% vlogan top.v -kdb
% vcs top -debug_access+r -kdb
```

Setting Up Verdi

To dump an FSDB file, you must set the following environment variables:

```
% setenv VERDI_HOME verdi_installation
% setenv LM_LICENSE_FILE[verdi_license]:$LM_LICENSE_FILE
```

Import KDB

```
% verdi -simflow -dbdir <path> -top top -nologo &
```

Invoke Verdi in Interactive Mode

```
% simv -gui
```

Invoke Verdi in Post-Processing Mode

```
% verdi -ssf novas.fsdb -nologo
```

Note:

Invoking Verdi in interactive and post-processing modes loads KDB automatically.

Key Points to Note

- The `vericom` utility exists in Verdi. For VCS users, it is recommended to use the `-kdb` option at compile time to generate the KDB database for data consistency and better performance. For third-party simulator users, the compile flow does not change and continues to use `vericom`. When loading the compiled design library (KDB) from the GUI (loading from the command line stays the same), ensure that the `vericom` option is selected in the **From** field under the **From Library** tab of the *Import Design* form.
- As VCS and `vericom` are different Verilog compilers, there are some behavioral differences between them. In such cases, VCS generated KDB follows the behavior of VCS for consistency reasons. The supported language subset also follows the supported subset of VCS.
- All the compilation information including compile log of Verdi KDB is logged to the regular VCS compiler log file.
- The library mapping information is obtained from the `synopsys_sim.setup` file in VCS three-step flow. The library mapping information in the `novas.rc` resource file is ignored in the VCS three-step flow.
- VCS generated KDB does not apply to the import-from-file flow of Verdi. The import-from-file flow continues to use the `vericom` parser to read in the Verilog source code directly. It uses the library mapping information from the `novas.rc` resource file, which is similar to the Verdi behavior.
- In the VCS two-step flow, the VCS generated KDB (`kdb.elab++`) is saved under the `simv.daidir/` directory (like `simv.daidir/kdb.elab++`).
- In the VCS three-step flow, the `vlogan -work <work>` generated KDB (`work.lib+`) is saved in the same working directory as `AN.DB` and the physical directory path of the library is picked as per the mapping present in the `synopsys_sim.setp` file. You can use the `verdi -simflow -lib` option to specify the working directory to load the KDB.

Limitations

The following are the limitations with VCS generated KDB:

- Parallel compilation is not supported.
- Fault tolerance compilation is not supported.

Dumping FSDB File for Various Flows

This section describes the use model to set up Verdi and dump an FSDB file using VHDL procedures, Verilog system tasks, or UCLI in the following topics:

- [Setting Up Verdi](#)
- [Use Model for FSDB Dumping](#)
- [Examples](#)

Setting Up Verdi

To dump an FSDB file, you must set the following environment variables:

```
% setenv VERDI_HOME verdi_installation
% setenv LM_LICENSE_FILE[verdi_license]:$LM_LICENSE_FILE
```

Use Model for FSDB Dumping

This topic describes the use model to dump an FSDB file using VHDL procedures, Verilog system tasks, or UCLI.

- Using VHDL Procedures

The following are the two ways to dump an FSDB file using VHDL procedures:

- You can use the VHDL procedures `fsdbDumpfile()` and `fsdbDumpvars()` in your VHDL code to dump an FSDB file.

Note:

To use these procedures, you should include `SYNOPSYS` library in your VHDL file as follows:

```
--Your VHDL file
library SYNOPSYS;
use SYNOPSYS.novas.all;

entity test is
```

```

    ...
end test;

architecture arch of test is
  ...
end arch;

```

- You can use the Verdi provided VHDL file: compile the Verdi provided VHDL file `$VERDI_HOME/$VERDI_LIB/novas.vhd` using the VCS analyzer and `vhdlan`, and save it in the same directory where the design is saved. The `novas.vhd` VHDL file contains the definitions of the FSDB foreign functions.

Use the `novas` package in any VHDL design file that invokes FSDB foreign functions.

Example:

```

use work.novas.all; --using novas package.
entity testbench is end;
architecture blk testbench is Begin
  ...
  Process begin: dump
    fsdbDumpvars(0,  : , +fsdbfile+signal.fsdb ); -- call
    VHDL procedure wait;
  end process end;

```

Then recompile the VHDL files you have modified.

- Using Verilog System Tasks

You can use the Verilog system tasks `$fsdbDumpfile()` and `$fsdbDumpvars()` in your Verilog design to dump an FSDB file (see [Using VHDL Procedures or Verilog System Tasks](#)).

- UCLI

At a UCLI prompt, you can use either of the following commands to dump an FSDB file.

- Using the UCLI `dump` command.

```

dump [-file <filename>] [-type FSDB]
dump -add <list_of_nids> [-fid <fid>] [-depth <levels>]
[-aggregates]

```

For complete use model of the UCLI `dump` command, see the *Unified Command Line Interface User Guide*.

Note:

FSDB is the default dump type when the `VERDI_HOME` environment variable is set.

- Using the UCLI commands `fsdbDumpfile` and `fsdbDumpvars`.

For information on the Verdi FSDB dumping commands for Verilog, see *Linking Novas Files with Simulators and Enabling FSDB Dumping Manual* under Verdi documentation in SolvNetPlus.

You must use the `-debug_access` option to enable FSDB dumping.

Using VHDL Procedures or Verilog System Tasks

Analysis

```
% vlogan [vlogan_options] file1.v file2.v
% vhdlan [vhdlan_options] file3.vhd file4.vhd
```

Elaboration

```
% vcs -debug_access [elab_options] top_module/entity/cfg
```

For VHDL procedural flow, you must use the `-vhpi` option at runtime as follows:

```
% simv -vhpi novas:FSDBDumpCmd
```

Simulation

```
% simv [run_options]
```

Using UCLI

Simulation

```
% simv [run_options] -ucli
ucli> fsdbDumpfile your_fsdb_dumpfile
ucli> fsdbDumpvars level module/entity
```

Note:

The default FSDB file name is `novas.fsdb`.

Examples

Example 248 Using Verilog System Tasks

This example demonstrates the use of Verilog system tasks, `$fsdbDumpfile` and `$fsdbDumpvars`.

```
`timescale 1ns/1ns
module test;
initial
begin
$fsdbDumpfile("test.fsdb");
```

```
$fsdbDumpvars(0,test);
end
```

```
...
endmodule
```

Now, the use model to compile/elaborate and simulate the above design is as follows:

VCS Two-step Flow

Compilation:

```
% vcs -debug_access test.v
```

Simulation:

```
% simv
```

VCS Three-step Flow

Analysis:

```
% vlogan test.v
```

Elaboration:

```
% vcs -debug_access test
```

Simulation:

```
% simv
```

The above set of commands dumps all the instances in `test` into the `test.fsdb` file.

Example 249 Using UCLI

This example demonstrates the use of UCLI commands `fsdbDumpfile` and `fsdbDumpvars` at the UCLI prompt to dump an FSDB file:

Consider the following Verilog file:

```
'timescale 1ns/1ns
module test();
.....
endmodule
```

The use model to compile/elaborate the design to use UCLI commands is as follows:

VCS Two-step Flow

Compilation:

```
% vcs -debug_access test.v
```

Simulation:

```
% simv -ucli
```

Using UCLI dump command	Using UCLI Command Line Option
ucli> dump -file test.fsdb -type FSDBucli> dump -add test -depth 0 -fid FSDB0ucli> runucli> quit	ucli> fsdbDumpfile test.fsdbucli> fsdbDumpvars 0 testucli> runucli> quit

VCS Three-step Flow

Analysis:

```
% vlogan test.v
```

Elaboration:

```
% vcs -debug_access test
```

Simulation:

```
% simv -ucli
```

Using UCLI dump command	Using UCLI Command Line Option
ucli> dump -file test.fsdb -type FSDBucli> dump -add test -depth 0 -fid FSDB0ucli> runucli> quit	ucli> fsdbDumpfile test.fsdbucli> fsdbDumpvars 0 testucli> runucli> quit

The above command dumps the whole design `test` into the `test.fsdb` file.

Interactive and Post-Processing Debug

After the Verdi Knowledge Database (KDB) is generated, you can invoke Verdi with the KDB in a single step for the following debug modes respectively:

- Interactive Simulation Debug Mode

Verdi can be automatically invoked with the KDB through the simulator command line option to perform interactive simulation debugging without other configurations.

- Post-Processing Debug Mode

The KDB and the `synopsys_sim.setup` file information can be automatically loaded into Verdi through a command line option to perform post-processing debug. There is no need to manually specify the compiled design. VCS and Verdi will have the same information from the `synopsys_sim.setup` file.

For detailed information on using the Verdi platform, see *Verdi and Siloti Command Reference Manual* under Verdi documentation in SolvNetPlus.

Prerequisites

The following are the prerequisites to perform interactive simulation debug and post-process debug:

- Generate Verdi KDB
- Specify the `-debug_access+<option>` compile time option on the VCS command line. This option automatically picks Verdi tab file and Verdi PLI file, and there is no need to pass these files explicitly during compilation. For more information on the `-debug_access` option, see [Optimizing Simulation Performance for Desired Debug Visibility With the -debug_access Option](#) section.
- Enable FSDB file dumping using the dumping tasks present in the source file or at runtime using `fsdbDumpvars` from the UCLI command line.

Interactive Simulation Debug Flow

When executing the `simv` simulator executable, perform one of the following steps to invoke Verdi within the interactive simulation debug mode:

- Use the `-gui` option to invoke Verdi.

For example,

```
// invoke Verdi
%> simv <simv_options> -gui [-verdi_opts "<verdi_options>"]
```

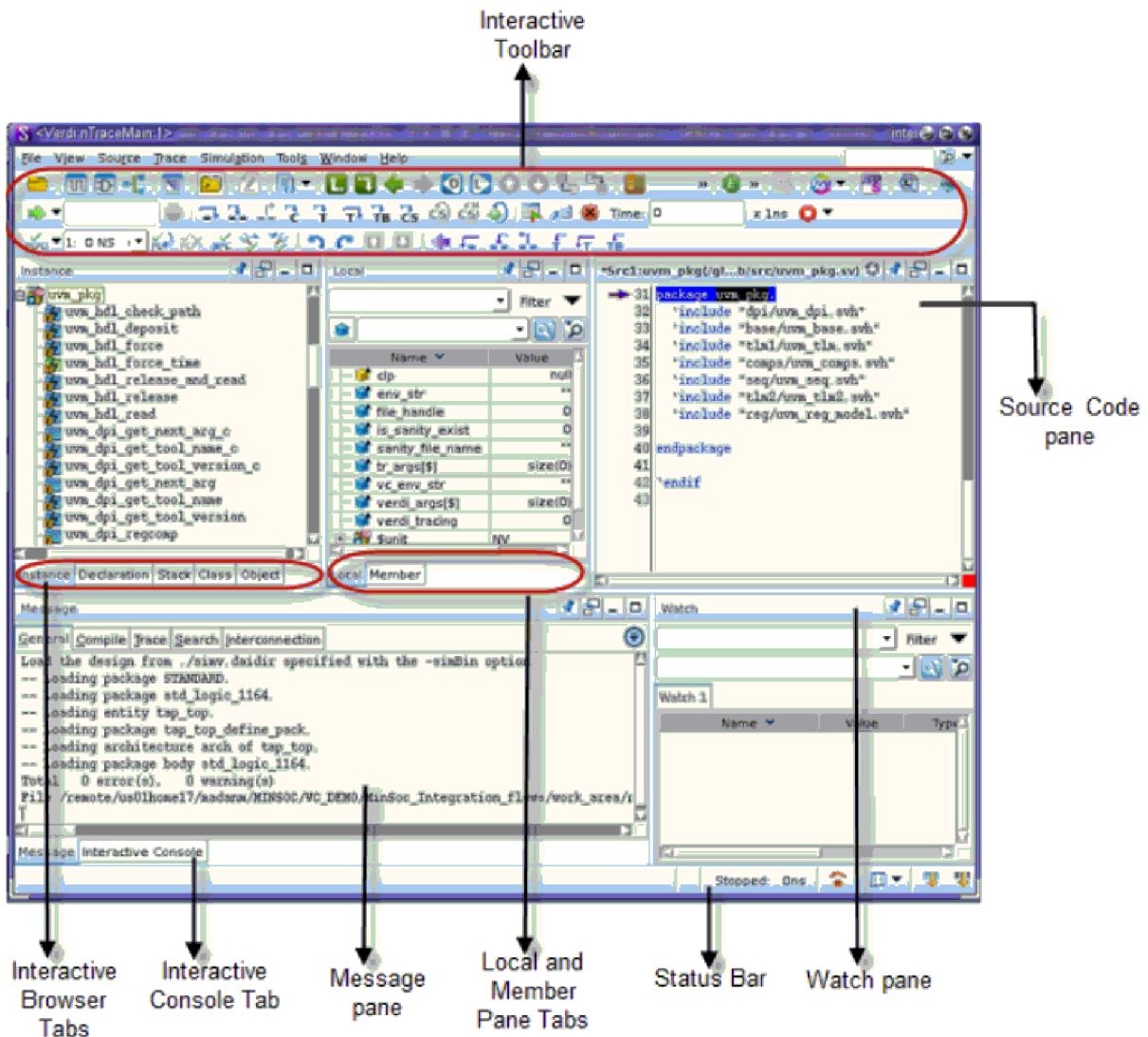
- Set the `SNPS_SIM_DEFAULT_GUI` environment variable to `verdi` to set the default debug tool as Verdi and the default dump type as FSDB.

Examples

```
//setting the default debug tool as Verdi and the default dump type as
FSDB

%> setenv SNPS_SIM_DEFAULT_GUI verdi
%> simv <simv_options> -gui [-verdi_opts "<verdi_options>"]
```

Figure 202 Verdi GUI



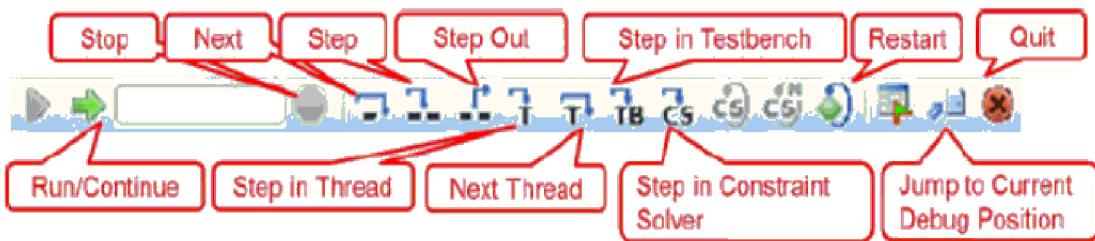
You can view the following panes in the Interactive Simulation Debug mode:

- Click *Instance* tab to view the *Instance pane*. This pane displays the static instance tree. The names in the instance tree are in the “instance name (definition name)” format. Top modules (or scopes) are at the top-level of the tree.
- Click *Stack* tab to view the *Stack pane*. This pane displays the testbench dynamic hierarchy tree along with all the testbench threads and their status.
- Click *Class* tab to view the *Class pane*. This pane displays all the classes defined in the design in a hierarchical view.

- Click *Object* tab to view the *Object pane*. This pane displays current dynamic objects and its values.
- Click *Local* tab to view the *Local pane*. This pane displays variables in a selected scope in the stack pane. This view is tied to the *Stack pane*, and the default view shows variables of the current active thread.
- Click *Member* tab to view the *Member pane*. This pane displays members and values of a class object selected in the *Object pane*.
- The *Interactive Console* frame provides the interface to input the Unified Command Line Interface (UCLI) commands and shows the results of simulator execution.
- The *Watch pane* allows you to monitor the status of your variables during simulation.
- The simulator status is shown in the *Status bar*.

Simulation Control Commands

The frequently used simulation control commands are listed in the Interactive toolbar as shown in the following figure to direct the simulator how to execute the program.



You can advance the simulation to the statement where a function is called using the **Step** command (by clicking the icon or selecting the **Simulation > Step/Next > Step** command). The Local and Watch tabs are refreshed accordingly.

When the **Next** or **Step** simulation control commands are used, and the scope of the current simulation belongs to a testbench or defined as a class object, all variables of the current simulation scope are shown automatically in the Local tab.

The *Watch tab* is used to observe the current values of variables in the current simulation scope. Both the values and current scopes will be updated as simulation time changes.

Values can be annotated to the variables on the source code frame using the **Source > Active Annotation** command. The values of the variables will be displayed in the *Source Code* frame.

When you run a simulation interactively, the line where the simulation stopped is marked by the blue arrow in the *Source Code* frame.

You can set breakpoints to stop the simulation. Breakpoints can be set by single-clicking or double-clicking the line number in the *Source Code* frame. Breakpoints execute each time a specified line is reached during simulation. A green flag  indicates that a breakpoint is set. Use **Simulation > Manage Breakpoints** command to open the Manage Breakpoints dialog box. This dialog box allows you to edit the breakpoints with more options.

The **Run Simulation** icon  or **Simulation > Run/Continue** command starts the simulator. If the simulator is invoked and stopped at a preset breakpoint (a flag and arrow combination indicates the current breakpoint and simulator position), invoke this command again to continue the simulation run.

For more information, see *Verdi SVTB Interactive Simulation Debug User Guide* and *Verdi and Silioti Command Reference Manual*.

Key Points to Note

- Use the `-verdi_opts` option to specify other Verdi specific options.
- The UVM Interactive Debug in Verdi is enabled by default while using the Unified Debug solution.
- If the design includes SystemC and the `default.ridb` is not available under the `simv.daidir/` directory, Verdi generates it automatically.
- In SystemC designs, for SystemC debug flow, you must create a `.ridb` file and set the `SNPS_VERDI_CBUG_LCA` environment variable.

Post-Processing Debug Flow

To automatically load the KDB, use one of the following Verdi command line options:

- `verdi -ssf <fsdb_file>`
To use this command line option, compile your design with `-kdb` and generate FSDB.
- `verdi -simflow -dbdir <path> -top <top_name> <other_verdi_options>`
Where,
`-simflow`

Enables Verdi and its utilities to use the library mapping from the `synopsys_sim.setup` file and also import the design from the KDB library paths.

```
-dbdir <path>
```

Specifies the path of the library directory when you want to invoke Verdi from a working directory that is different from the VCS working directory. For more information, see [Reading Compiled Design with Verdi](#).

Reducing Disk Space for Post-Process Only Debug

You can use the `trim_daidir` script file to delete the files that are not required for the post-process debug flow. This feature helps you to significantly reduce the `simv.daidir` disk space.

Use Model

Perform the following steps:

1. Compile the testcase. For example,

```
% vcs -sverilog -kdb -lca -debug_access+all file.sv
```

2. Run the simulation once and dump the FSDB file.

```
% simv <simv_options>
```

3. Use the `trim_daidir` script file to delete the files that are not required for the post-process debug flow.

Example: `trim_daidir simv.daidir`

4. Execute the following command to invoke Verdi:

```
% verdi -simflow -dbdir simv.daidir -ssf <fsdb_file>
```

A

VCS Environment Variables

This appendix covers the following topics:

- [Simulation Environment Variables](#)
 - [Setup Variables](#)
 - [Optional Environment Variables](#)
 - [Using Environment Variables in Verilog Source Code](#)
-

Simulation Environment Variables

To run VCS, you need to set the following basic environment variables:

`$VCS_HOME`

When you or someone at your site installed VCS, the installation created a directory that is called the `vcs_install_dir` directory. Set the `$VCS_HOME` environment variable to the path of the `vcs_install_dir` directory. For example:

```
setenv VCS_HOME /u/net/eda_tools/vcs2005.06
```

`PATH`

On UNIX, set this environment variable to `$VCS_HOME/bin`. Add the following directories to your `PATH` environment variable:

```
set path=($VCS_HOME/bin\  
         $VCS_HOME/'$VCS_HOME/bin/vcs -platform'/bin\  
         $path)
```

Also, make sure the `path` environment variable is set to a bin directory containing a `make` or `gmake` program.

`LM_LICENSE_FILE` or `SNPSLMD_LICENSE_FILE`

The definition can either be an absolute path name to a license file or to a port on the license server. Separate the arguments in this definition with colons. For example:

```
setenv LM_LICENSE_FILE 7182@serveroh:/u/net/server/eda_tools/license.dat
```

or

```
setenv SNPSLMD_LICENSE_FILE
7182@serveroh:/u/net/server/eda_tools/license.dat
```

Note:

- You can use `SNPSLMD_LICENSE_FILE` environment variable to set licenses explicitly for Synopsys tools.
- If you set the `SNPSLMD_LICENSE_FILE` environment variable, then VCS ignores the `LM_LICENSE_FILE` environment variable.

Setup Variables

You can configure the compilation and simulation behavior of VCS by assigning values to setup variables in the `synopsys_sim.setup` file. The variable assignment statements have the following syntax:

```
variable_name = value
```

This section lists the setup variables that affect VCS simulation. In addition to these variables, the setup file can contain other variable assignments that apply to other Synopsys tools. VCS ignores setup variables related to other products, but generates a warning for the unrecognized variables.

The setup variables described in this section are organized into the following parts:

- [Analysis Setup Variables](#)
- [Compilation/Elaboration Setup Variables](#)
- [Simulation Setup Variables](#)
- [C Compilation and Linking Setup Variables](#)
- [Timescale Implementation](#)

Analysis Setup Variables

The setup variables that configure the analysis behavior of VCS are listed here in alphabetical order.

`CHECK_PURE_FUNC`

By default, VCS performs checks for pure functions in compliance with the VHDL93 LRM and issues an error message at the vhdlan stage when an impure function is called from pure function in VHDL.

As per the VHDL LRM, a pure function cannot be the parent of an impure function. When CHECK_PURE_FUNC is set to FALSE, VCS disables checks for pure functions and does not issue warning or error message when an impure function is called from pure function in VHDL. The default value of CHECK_PURE_FUNC is TRUE.

`IGNORE_BINDING_HOMOGRAPH`

Controls the generation of warning messages when encountering homographs while doing component binding. When set to `TRUE`, VCS suppresses all component binding homograph messages. The default value of `IGNORE_BINDING_HOMOGRAPH` is `FALSE`.

`LIBRARY_SCAN`

When set to `TRUE`, it checks and searches for a matching entity in all libraries defined in the `synopsys_sim.setup` file to resolve a component instantiation. If one is not found, an error message is issued. The default value of `LIBRARY_SCAN` is `FALSE`.

`LICENSE_WAIT_TIME`

Enables license queuing and specifies the timeout time in minutes before `vhdlan` gives up waiting for a license.

The timeout time should be an integer greater than zero; any decimal part of the number is ignored.

With the `LICENSE_WAIT_TIME` variable in the setup file set to an integer, you do not have to specify the `-licwait` option. However, if you do specify the `-licwait` option, it overrides the setting in the setup file.

This variable affects analysis, compilation, and simulation steps. This variable is not set by default.

`OPTIMIZE`

When set to `TRUE`, the VCS analyzer optimizes the compiled event code by eliminating VHDL checks for:

- Arithmetic overflow
- Constraint checks
- Array size compatibility at assignment
- Subscripts out of bounds
- Negative exponents to integer

The `-optimize` option to the `vhdlan` command overrides the `OPTIMIZE` value. The default value of `OPTIMIZE` is `TRUE`.

Note:

If a VHDL error occurs when `OPTIMIZE` is `TRUE`, you may receive erroneous results or it can cause VCS to fail in an unpredictable way. If you have not completely debugged your design, it is recommended to temporarily set `OPTIMIZE` to `FALSE`.

`RELAX_CONFORMANCE`

When set to `TRUE`, the VCS analyzer relaxes any VITAL conformance violation error into a warning when analyzing VITAL models. The default value of `RELAX_CONFORMANCE` is `FALSE`.

`SPC`

When set to `TRUE`, the VCS analyzer performs synthesis policy checking while analyzing VHDL design files. The analyzer checks the VHDL design files against the VHDL subset supported by Synopsys synthesis tools. The analyzer does not check for synthesis elaboration errors.

To make the synthesis policy checking work correctly, you must install the synthesis software correctly and the `$SYNOPSYS` variable must point to your synthesis installation. The `-spc` option of the `vhdlan` command overrides the `SPC` value. The default value of `SPC` is `FALSE`.

`IEEE_1076_1987`

When set to `TRUE`, VHDL analyzer allows you to use VHDL-87 syntax. The default value of `IEEE_1076_1987` is `FALSE`.

`XLRM_TIME`

When set to `TRUE`, VCS (vlogan) relaxes timescale restriction, and issues a warning message when a module does not have timescale at analysis phase. For more information, refer to [Timescale Implementation](#).

Compilation/Elaboration Setup Variables

The following setup variables configure the compilation behavior of VCS.

`ERROR_WHEN_UNBOUND`

Set this variable to `TRUE` to change a warning message to an error message issued due to an unbound design unit. By default, VCS issues a warning message if there are any unbound design units.

`IGNORE_BINDING_HOMOGRAPHHS`

See [IGNORE_BINDING_HOMOGRAPHHS](#) for more information.

`LIBRARY_SCAN`

See [LIBRARY_SCAN](#) for more information.

`LICENSE_WAIT_TIME`

See [LICENSE_WAIT_TIME](#) for more information.

`NUM_COMPILERS`

Specifies the number of compilers used in parallel compilation. When `PARALLEL_COMPILE_OFF` is `FALSE`, `NUM_COMPILERS` is set to 4. You can override the default value by specifying another integer value. If `PARALLEL_COMPILE_OFF` is `TRUE`, `NUM_COMPILERS` is set to 1, that is, serial compilation. The default value of `NUM_COMPILERS` is 4.

`PARALLEL_COMPILE_OFF`

Speeds up the compilation of generated C files by controlling the parallelism between code generation and compilation, and between compilation of different files.

When set to `TRUE`, elaboration step uses serial compilation instead of parallel compilation. The default value of `PARALLEL_COMPILE_OFF` is `FALSE`.

`TIMEBASE`

Specifies the basic unit of time used in simulating the design. All units of time used and understood by VCS are non-negative, whole-number multiples of the timebase unit. Valid `TIMEBASE` values are `fs`, `ps`, `ns`, `us`, `ms`, and `sec`.

The `-time` option to the `vcs` command overrides the `TIMEBASE` value. The default value of `TIMEBASE` is `ns`.

`TIME_RESOLUTION`

Specifies the VCS time resolution. It basically sets the precision or the number of simulation ticks per base time unit.

`TIME_RESOLUTION = [1 | 10 | 100] [fs | ps | ns | us | ms | sec]`

If no numeric value (1, 10, or 100) is provided, then the default value is 1. For example:

`TIME_RESOLUTION = ps`

If a value beside 1, 10, or 100 is provided, a warning is issued during elaboration and a default setting of 1 <unit> is used (where unit is the specified time unit (fs, ps, etc.)).

Time resolution value cannot be higher than the time base value. An error is issued if this happens.

The `-time_resolution` option to the `vcs` command overrides the `TIME_RESOLUTION` value. The default value is `TIME_RESOLUTION = 1NS`.

`ELAB_EXPAND_ENV`

When set to `TRUE`, this environment variable supports the expansion of UNIX environment variable, which is used with VHDL string generic.

Simulation Setup Variables

The following setup variables configure the simulation behavior of VCS.

`ASSERT_IGNORE`

Controls the generation of messages in response to VHDL assertion violations or report statements. The possible values for this variable are `NOTE`, `WARNING`, `ERROR`, `FAILURE`, `NOIGNORE`, or `NOTSET`.

`ASSERT_IGNORE` has higher precedence than the individual assertion variable settings. If `ASSERT_IGNORE` equals `NOTSET`, simulation proceeds to check the values of the individual assertion variable settings, `ASSERT_IGNORE_NOTE`, `ASSERT_IGNORE_WARNING`, `ASSERT_IGNORE_ERROR`, and `ASSERT_IGNORE_FAILURE`. If `ASSERT_IGNORE` is set to any other value, the individual assertion variable settings are ignored.

If `ASSERT_IGNORE` equals `NOIGNORE`, the simulation prints messages for all assertion violations. The other values prevent simulation from printing a message unless the assertion violation is of greater severity than the value specified.

`ASSERT_IGNORE` has higher precedence than `ASSERT_STOP`. This means that when `ASSERT_IGNORE` is set, the simulator does not stop on `ASSERT_STOP` assertions. The default value of `ASSERT_IGNORE` is `NOTSET`.

`ASSERT_IGNORE_NOTE`

Controls the generation of messages in response to VHDL assertion violations of severity `NOTE`. If set to `TRUE`, all assertions of severity `NOTE` are ignored. VHDL assertions of severity other than `NOTE` are not affected by this variable.

`ASSERT_IGNORE` has higher precedence than `ASSERT_IGNORE_NOTE`. If `ASSERT_IGNORE` is set to any value other than `NOTSET`, the value of `ASSERT_IGNORE_NOTE` is ignored. The default value of `ASSERT_IGNORE_NOTE` is `FALSE`.

`ASSERT_IGNORE_WARNING`

Controls the generation of messages in response to VHDL assertion violations of severity `WARNING`. If set to `TRUE`, all assertions of severity `WARNING` are ignored. VHDL assertions of severity other than `WARNING` are not affected by this variable.

`ASSERT_IGNORE` has higher precedence than `ASSERT_IGNORE_WARNING`. If `ASSERT_IGNORE` is set to any value other than `NOTSET`, the value of

ASSERT_IGNORE_WARNING is ignored. The default value of **ASSERT_IGNORE_WARNING** is **FALSE**.

ASSERT_IGNORE_ERROR

Controls the generation of messages in response to VHDL assertion violations of severity **ERROR**. If set to **TRUE**, all assertions of severity **ERROR** are ignored. VHDL assertions of severity other than **ERROR** are not affected by this variable.

ASSERT_IGNORE has higher precedence than **ASSERT_IGNORE_ERROR**.

If **ASSERT_IGNORE** is set to any value other than **NOTSET**, the value of **ASSERT_IGNORE_ERROR** is ignored. The default value of **ASSERT_IGNORE_ERROR** is **FALSE**.

ASSERT_IGNORE_FAILURE

Controls the generation of messages in response to VHDL assertion violations of severity **FAILURE**. If set to **TRUE**, all assertions of severity **FAILURE** are ignored. VHDL assertions of severity other than **FAILURE** are not affected by this variable.

ASSERT_IGNORE has higher precedence than **ASSERT_IGNORE_FAILURE**.

If **ASSERT_IGNORE** is set to any value other than **NOTSET**, the value of **ASSERT_IGNORE_FAILURE** is ignored. The default value of **ASSERT_IGNORE_FAILURE** is **FALSE**.

ASSERT_IGNORE_OPTIMIZED_LIBS

Defines the maximum severity level of an assertion to be ignored in the built-in packages during simulation. For global scope, the value of **ASSERT_IGNORE** is used. For built-in simulation packages, the value of the higher severity level between **ASSERT_IGNORE** and **ASSERT_IGNORE_OPTIMIZED_LIBS** takes precedence. These built-in packages include all the Synopsys and IEEE packages included with VCS.

Valid values for this variable are **ERROR**, **NOTE**, **WARNING**, **FAILURE**, or **NOIGNORE**. The default value of **ASSERT_IGNORE_OPTIMIZED_LIBS** is **WARNING**.

ASSERT_STOP

Determines whether simulation stops in response to VHDL assertion violations. The possible values for this variable are **NOTE**, **WARNING**, **ERROR**, **FAILURE**, or **NOSTOP**.

If **ASSERT_STOP** equals **NOSTOP**, simulation never stops for assertion violations. The other values cause simulation to stop when it encounters assertion violations of severity equal to, or greater than, the value specified. The default value of **ASSERT_STOP** is **ERROR**.

CS_ASSERT_STOP_NEXT_WAIT

Controls the response of the compiled-code simulation mode to VHDL **ASSERT** statements. If set to **TRUE**, a failed VHDL assertion causes VCS to continue until the next **WAIT**

statement, then stop. If not set, or set to FALSE, VCS prompts you to choose whether to stop immediately or to continue until the next WAIT statement.

For example:

```
Assertion ERROR at 30 NS in design unit E(A) from process /E/_P0:  
  "Assertion violation."  
An ASSERT_STOP is currently pending in compiled code, and  
  CS_ASSERT_STOP_NEXT_WAIT is not set to TRUE in synopsys_sim.setup.  
Continue until next wait (y), or stop simulation immediately (n)? [y/n]:
```

If you choose to stop at the next WAIT statement, you can then continue the simulation by executing the VCS run command.

If you choose to stop immediately, you cannot continue the current simulation. You must either restart the simulation with the VCS restart command or quit VCS and start it again.

The CS_ASSERT_STOP_NEXT_WAIT has no effect on debug mode simulations. The default value of CS_ASSERT_STOP_NEXT_WAIT is TRUE.

`CS_ASSERT_STOP_PROMPT`

If set to TRUE when running batch mode simulation, this variable causes the simulation to stop immediately without the possibility of continuing if an assertion of severity equal or higher than ASSERT_STOP occurs. The default value of CS_ASSERT_STOP_PROMPT is FALSE.

`LICENSE_WAIT_TIME`

See [LICENSE_WAIT_TIME](#) for more information.

`MAX_DELTA`

Specifies the maximum number of delta cycles in a simulation time step. When MAX_DELTA is set to a positive value, *simv* monitors the delta cycle number and stops the simulation when it reaches the MAX_DELTA limit. *simv* then issues a warning and prints a list of signals with pending zero-delay transactions. Additionally, *simv* may print a list of processes with pending wait for 0 timeouts. With that information, you can immediately start debugging possible infinite zero-delay cycles.

If you decide there is nothing wrong, you can disable delta cycle monitoring by setting MAX_DELTA to zero, or to a negative value. The default value of MAX_DELTA is 0.

`MONITOR_TIME_DISPLAY`

If set to FALSE, the monitor command does not display time information. The default value of MONITOR_TIME_DISPLAY is TRUE.

`USE`

Specifies the list of directories, separated by spaces, that VCS searches for VHDL source files. This information is used for viewing the VHDL source code of a design during a simulation.

The settings for the `USE` variable are not cumulative. For example, if there is a `synopsys_sim.setup` file in your home directory with `USE = ./ ./asic_lib`, and in your design directory, the `USE` variable is set to `USE = ./my_lib ./temp_lib`, the final value for the `USE` variable is `USE = ./my_lib ./temp_lib`.

The default value of `USE` is:

```
USE = . $VCS_HOME/packages/synopsys/src \
        $VCS_HOME/packages/IEEE/src \
        $VCS_HOME/packages/IEEE_asic/src \
        $VCS_HOME/packages/gtechnox/src \
        $VCS_HOME/packages/gtech/src \
        $VCS_HOME/packages/gscomp/src \
        $VCS_HOME/packages/dware/src \
        $VCS_HOME/dw/dw01/src \
        $VCS_HOME/dw/dw02/src \
        $VCS_HOME/dw/dw03/src \
        $VCS_HOME/dw/dw04/src \
        $VCS_HOME/dw/dw05/src \
        $VCS_HOME/dw/dw06/src \
        $VCS_HOME/dw/dw07/src \
        $VCS_HOME/dw/dw08/src
```

VCD_IMMEDIATE_FLUSH

When set to `TRUE`, every time you issue a new VCD dump command, the VCD file is immediately updated with the correct header and signal information. By default, all VCD file information is flushed when you exit VCS.

Setting this variable to `TRUE` may slow down the simulation performance when tracing design objects. The default value of `VCD_IMMEDIATE_FLUSH` is `FALSE`.

VCD_OUTFILE

Specifies the output filename for the VCD file. To create a VCD file, use the `dump` command during simulation. The VCD file contains traced data that is used for post-processing. For example, you can set `VCD_OUTFILE = my_vcd_file.vcd`.

VPD_DELTA_CAPTURE

Enables delta-cycle capturing in interactive simulation. The default value of `VPD_DELTA_CAPTURE` is `OFF`.

WAVEFORM_UPDATE

When set to `TRUE`, objects in the Wave View are refreshed with every simulation time step. By default, the Wave View is refreshed when each simulation command is

completed. Setting this variable to `TRUE` slows down the simulation performance when tracing design objects. The default value of `WAVEFORM_UPDATE` is `FALSE`.

C Compilation and Linking Setup Variables

Following are the setup variables that configure the C compilation of the C code that VCS generates.

`CS_CCFLAGS_$ARCH`

Specifies the C compiler flags used to compile the VCS generated C code on the specific platform.

One reason to use this variable is to specify a different compiler optimization level, such as `-O3`.

To get a listing of flags for your C compiler, use the UNIX `man` utility.

The `CS_CCFLAGS` variable is still supported and it has higher precedence than the platform specific `CS_CCFLAGS_$ARCH` variables.

The `--ccflags` option to the `vhdlan` and `vcs` commands overrides the `CS_CCFLAGS_$ARCH` value.

The default value of `CS_CCFLAGS_$ARCH` is different for each platform. Default values for SparcOS5, Linux, and RS6000 are as follows:

- SparcOS5

`CS_CCFLAGS_SPARCOS5 = -c -O`

- Linux

`CS_CCFLAGS_LINUX = -c -O`

- RS6000

`CS_CCFLAGS_RS6000 = -c -qchars=signed -O -qmaxmem=2048000`

`CS_CCPATH_$ARCH`

Specifies the C compiler used to compile VCS generated C code on the specific platform.

The GCC compiler is incorporated in the VCS image for Sun SPARC operating systems (Solaris). This is the recommended compiler for the Solaris platform. VCS is optimized for performance with the GCC C compiler.

Note:

`CS_CCPATH` variable is supported and it has higher precedence than the platform specific `CS_CCPATH_$ARCH` variables.

The `-ccpath` option to the `vhdlan` and `vcs` commands overrides the `CS_CCPATH_$ARCH` value.

The default value of `CS_CCPATH_$ARCH` is different for each platform. Default values for SparcOS5, Linux, and RS6000 are as follows:

- SparcOS5

```
CS_CCPATH_SPARCOS5 = $VCS_HOME/sparcOS5/gcc/gcc-2.6.3/bin/gcc
```

- Linux

```
CS_CCPATH_LINUX = cc
```

- RS6000

```
CS_CCPATH_RS6000 = cc
```

Note:

It is your responsibility to set up the proper path for the C compiler on HPUX10, LINUX, and RS6000 platforms. This can be done in many different ways, for example:

- At tool's initial installation time, by editing the master `synopsys_sim.setup` file (from `/admin/setup`) and setting the proper C compiler path.
- For each user in their home directory, by having own `synopsys_sim.setup` file with proper C compile path.
- By setting the `PATH` environment variable to pick up the proper C compiler by default.

Timescale Implementation

VCS supports the timescale implementation as defined in the IEEE 1800 standard. For information on timescale directives, see the *Verilog Language Reference Manual*.

This section describes the following topics:

- [Understanding `timescale](#)
- [Verilog only and Verilog Top Mixed Design](#)
- [VHDL only and VHDL Top Mixed Designs](#)
- [Setting up Simulator Resolution From Command Line](#)
- [Other Useful Timescale Related Options](#)

- Non-Compatible Options
- Limitations

Understanding `timescale

In Verilog, all delays are governed by `timescale directive in the source file. The behavior is precisely defined in the *1364-1995 Verilog Language Reference Manual*. Now, there can be multiple `timescale compiler directives across multiple files. According to LRM:

The ‘timescale compiler directive specifies the unit of measurement for time and delay values, and the degree accuracy for delays in all modules that follow this directive until another ‘timescale compiler directive is read.

Consider the following three files:

a.v	b.v	c.v
`timescale 10ns/1ns module a; endmodule	`timescale 10ps/1ps module b; endmodule	module c; endmodule

You can see that the file *c.v* does not contain any timescale information, so it inherits the timescale from last encountered one during parsing.

Scenario 1:

```
% vlogan a.v b.v c.v
```

In this case, *a.v* and *b.v* have their own timescale, so they follow it. But for *c.v*, the last encountered timescale is from *b.v* (10ps/1ps), so the simulator assigns the same to *c.v*.

Scenario 2:

```
% vlogan a.v c.v b.v
```

In this case, *a.v* and *b.v* follow their own well-defined timescale. However, *c.v* inherits timescale from *a.v*, as it is the latest one as far as *c.v* is concerned.

Scenario 3:

```
% vlogan c.v a.v b.v
```

In this case, it is not very clear which timescale *c.v* gets, as no timescale is parsed before *c.v*.

Situation becomes more complex when you go for mixed language simulation, involving both Verilog and VHDL.

Therefore, VCS came up with well-defined set of rules for all the above scenarios. This new implementation is under a variable defined in `synopsys_sim.setup` file. The syntax for the same is as follows:

```
XLRM_TIME = TRUE
TIMEBASE=time_base
TIME_RESOLUTION=time_resolution
```

where,

```
time_number ::= 1 | 10 | 100
time_unit ::= s[ec] | ms | us | ns | ps | fs
time_base ::= time_unit
time_resolution ::= time_number time_unit
```

If you specify only `XLRM_TIME=TRUE` without `TIME_RESOLUTION`, then it will be set to the value of `TIMEBASE`. There is a default `TIMEBASE` defined in default `synopsys_sim.setup` (from `$VCS_HOME/bin`).

It is recommended that the `time_unit` for `TIMEBASE` and `TIME_RESOLUTION` should be the same. If the `TIMEBASE` is finer than `TIME_RESOLUTION`, then it is an error condition. You can resolve this error condition by correcting the `TIMEBASE` entry in `synopsys_sim.setup`.

The following are the new terms which you will be using for rest of the section:

ana module:

Verilog modules which get the timescale during the analysis phase (during vlogan time) is termed as “*ana module*”. Out of the three scenarios mentioned above, in Scenario 1 and Scenario 2, `module c` does not have its own timescale, but inherits it from other modules (`module b` in Scenario 1 and `module a` in Scenario 2) because of the parsing order. Since you know the timescale for all three modules now, all three modules are classified as “*ana modules*” in Scenario 1 and Scenario 2.

elab module:

Verilog module which does not have any timescale after analysis phase is termed as “*elab module*”. In the above mentioned Scenario 3, `module c` neither has its own timescale nor has inherited from the previous modules, as there is none. Therefore, `module c` is treated as “*elab module*”, whereas `module a` and `module b` are treated as “*ana module*”. To make it clear, remove timescale from file `b.v`, hence it is rewritten as follows:

```
module b;
endmodule
```

Consider the same command line again:

```
% vlogan c.v a.v b.v
```

In this case, *c.v* does not have any timescale (by its own or by inheritance), *a.v* has its own, and *b.v* gets the one from *a.v* by inheritance.

Hence, module *c* is treated as “*elab module*”, whereas module *a* and module *b* are treated as “*ana module*”.

During elaboration phase, VCS assigns timescale to all “*elab modules*”. All it does is to calculate simulator precision and use it as a timescale for all “*elab modules*”. This means

Timescale for all elab modules = time_unit/time_precision

Verilog only and Verilog Top Mixed Design

For this topology of the design, simulator precision is determined by the finest of time resolution from all “*ana modules*”. If none of the Verilog modules in the design has timescale, then it is determined by TIME_RESOLUTION mentioned in the synopsys_sim.setup file.

VHDL world is also governed by simulator_precision. For example, reconsider Scenario 3. Also, consider the following synopsys_sim.setup file:

```
XLRM_TIME = TRUE
TIMEBASE=fs
TIME_RESOLUTION=1fs
```

Only module *a* and module *b* have timescales, and the finest resolution comes from module *b*, such as “1ps”. Hence it is treated as simulator_precision. Therefore, timescale assigned to module *c* is “1ps/1ps”. Note that TIME_RESOLUTION from the setup file is not considered here. Also, delays in VHDL files are rounded to resolution of “1ps” and not to “1fs” (from the synopsys_sim.setup file).

VHDL only and VHDL Top Mixed Designs

In this case, simulator_precision is determined by TIME_RESOLUTION in synopsys_sim.setup file irrespective of the finest time precision from all “*ana modules*”. If the finest time precision from all “*ana modules*” is finer than TIME_RESOLUTION in synopsys_sim.setup file, then it is an error condition, and therefore VCS issues a proper

error message. Consider the above given Verilog files (*a.v*, *b.v*, and *c.v*) and VHDL top given below:

```
library work;
use work.all;

entity top is
end top;

architecture top_arch of top is
component a is
end component;
component b is
end component;
component c is
end component;

begin
    U1:a;
    U2:b;
    U3:c;
end top_arch;
```

Now, `simulator_precision` is taken from *synopsys_sim.setup* file, that is “1fs”, and timescale given to module *c* is “1fs/1fs” (and not “1ps/1ps” as in case of Verilog top design).

Setting up Simulator Resolution From Command Line

You can set the simulator resolution from the command line irrespective of the design topology using the `-sim_res` option. The syntax of `-sim_res` is as follows:

`-sim_res=<time_resolution>`

Where,

```
time_resolution ::= time_number time_unit
time_number ::= 1 | 10 | 100
time_unit ::= s[ec] | ms | us | ns | ps | fs
```

This option supersedes the setting from *synopsys_sim.setup* file (in case of VHDL top designs) or finest resolution from Verilog “ana modules” (in case of Verilog only or Verilog top designs).

Also, the same is used to construct the timescale for all *elab* modules.

For example, if you pass `-sim_res=1fs`, then the timescale for *eab* module is “1fs/1fs”. Also, the overall simulator resolution is “1fs”.

Note:

- With the current implementation of `XLRM_TIME`, if `-sim_res` is coarser than `TIME_RESOLUTION` in `synopsys_sim.setup` (for VHDL top designs) or the finest time resolution from “*ana* modules” (for Verilog top designs), VCS issues an error message.
- For Verilog top designs, it is an error if the time resolution from the design is coarser than the time base from setup file.
- While using the `-sim_res` option, you must add the `XLRM_TIME=TRUE` in the `synopsys_sim.setup` file. Otherwise, the `-sim_res` option does not have any effect during simulation.

Other Useful Timescale Related Options

`-timescale=<time_unit/time_resolution>`

This is analysis time option. If present on the `vlogan` command line, it is applied to all files which have no timescale of their own, or not yet hit any timescale directive from other files during parsing order.

For example, consider following three files:

<code>a.v</code>	<code>b.v</code>	<code>c.v</code>
<code>'timescale 10ns/1ns module a; endmodule</code>	<code>'timescale 10ps/1ps module b; endmodule</code>	<code>module c; endmodule</code>

The command line is:

```
% vlogan -timescale=1fs/1fs a.v b.v c.v
```

In this case, `a.v` and `b.v` have their own timescale and `c.v` inherits it from `b.v`, so `timescale` has no effect in this case. Alter `c.v` to add ``resetall` in it, as follows:

```
'resetall
module c;
endmodule
```

`'resetall` nullifies all compiler directives hit so far during parsing. Therefore, `c.v` instead of inheriting timescale from `b.v`, now takes it from the command line option. This is same as having the following command line:

```
% vlog -timescale=1fs/1fs c.v a.v b.v
```

It is recommended to have the `-timescale` option accompanied with every `vlog` command line to avoid any ambiguity at later stage.

`-override_timescale=<time_unit/time_resolution>`

If applied at the analysis time, this option overrides the timescale of all analyzed modules into the same work library from all previous analysis commands.

Example:

```
% vlog -work lib1 a.v b.v
```

```
% vlog -work lib1 -override_timescale=10fs/1fs c.v d.v
```

In this case, timescale from `a.v` and `b.v` are replaced with the one from `-override_timescale` (from the second `vlog` command) and `c.v` and `d.v` are replaced with the one from `-override_timescale`. Hence, `-override_timescale` replaces timescale of all the modules that are analyzed so far into the work library `lib1`. Any other modules that are analyzed later into the work library `lib1` through other `vlog` commands are not affected by this option, and have their own timescale, that is, derived from the command line.

If applied at the elaboration time, this option is applied to all the modules in the design, irrespective of how they are analyzed.

Also, simulator precision is determined by `time_resolution` part of `-override_timescale`. This supersedes the `-sim_res` option.

Non-Compatible Options

Under this implementation, all older timescale related switches are ignored and appropriate warning is issued.

The following elaboration time options are ignored:

- -t [ime]
- -time_res[olution]
- -timescale (At elab time)

Limitations

- SystemC designs are not supported
- Separate compile flow is not supported

Optional Environment Variables

VCS also includes the following environment variables that you can set in certain circumstances.

`DISPLAY_VCS_HOME`

Enables the display, at compile time, of the path to the directory specified in the `VCS_HOME` environment variable. Specify a value other than 0 to enable the display. For example:

```
setenv DISPLAY_VCS_HOME 1
```

`MATCH_ENUM_VECTOR`

A port map aspect associates signals or values with the formal ports of a block.

When the `MATCH_ENUM_VECTOR` variable is set to TRUE, VCS allows types to be mappable in port maps as long as the element type of the array is a matching enumeration type. However, this does not comply with *VHDL LRM*. By default, this variable is not set.

It is recommended that the code be fixed to make it LRM compliant.

`SYSTEMC_OVERRIDE`

Specifies the location of the SystemC simulator used with the VCS/SystemC co-simulation interface. See [Using SystemC](#).

`TMPDIR`

Specifies the directory used by VCS and the C compiler to store temporary files during compilation.

`VCS_CC`

Indicates the C compiler to be used. To use the gcc compiler, specify the following:

```
setenv VCS_CC gcc
```

VCS_COM

Specifies the path to the VCS compiler executable named `vcs1`, not the compile script. If you receive a patch for VCS, you might need to set this environment variable to specify the patch. This variable is used for solving problems that require patches from VCS and should not be set by default.

VCS_LIC_EXPIRE_WARNING

By default, VCS displays a warning message 30 days before a license expires. You can specify that this warning message begin fewer days before the license expires with this environment variable, for example:

```
VCS_LIC_EXPIRE_WARNING 5
```

To disable the warning, enter the 0 value:

```
VCS_LIC_EXPIRE_WARNING 0
```

VCS_LOG

Specifies the runtime log file name and location.

VCS_TARGET_ARCH

Enables elaboration and simulation in 64-bit mode architecture. When running 64-bit simulations, there is no need to use `-full64` option whenever this environment variable is set. For example,

```
% setenv VCS_TARGET_ARCH linux64
```

VCS_NO_RT_STACK_TRACE

Tells VCS not to return a stack trace when there is a fatal error, and instead dump a core file for debugging purposes.

VCS_SWIFT_NOTES

Enables the `printf` PCL command. PCL is the Processor Control Language that works with SWIFT microprocessor models. To enable it, set the value of this environment variable to 1.

VCS_DIAGTOOL

Generates valgrind data for `vcs1`, if you set this environment variable as follows:

```
% setenv VCS_DIAGTOOL "valgrind --tool=memcheck"
```

Once you set this environment variable, any subsequent invocation of `vcs1` generates valgrind data.

OVERWRITE_PREDECLARATION_IN_LIB

By default, if the same vlogan command encounters more than one implementation bearing the same module name, VCS generates the following error message:

```
Error-[MPD] Module previously declared
The module was previously declared at:
"child_ccf_.v", 1
It is redeclared later at:
"child_ccf_.v", 1: token is 'child_ccf'
module child_ccf (clk );
^
Please remove one of the declarations and compile again.
```

To get a similar message for the scenario where several vlogan commands write to the same logical library, you should add OVERWRITE_PREDECLARATION_IN_LIB = TRUE to the synopsys_sim.setup file.

Consider the following example:

```
%cat child_ccf.v

module child_ccf ();
    initial $display("from child_ccf.v" );
endmodule

%cat child_ccf_.v

module child_ccf ();
    initial $display("from child_ccf_.v");
endmodule

%cat synopsys_sim.setup

rtl_one_lib : rtl_one_lib
OVERWRITE_PREDECLARATION_IN_LIB = TRUE
```

Run the following commands that parse two different files (one with an underscore before the file extension) to the same library:

```
%vlogan -sverilog child_ccf.v -work rtl_one_lib
%vlogan -sverilog child_ccf_.v -work rtl_one_lib
```

VCS generates the following warning message:

```
Parsing design file 'child_ccf_.v'

Warning-[OPDL] Overwrite pre-declaration in library
child_ccf_.v, 1
child_ccf_
A previously declared module 'child_ccf' at "child_ccf.v,1" is found in
library 'RTL_ONE_LIB'. It will be Overwritten by current module with the
same name.
```

Using Environment Variables in Verilog Source Code

To make the Verilog source code reusable, environment variables are often used in the source code instead of providing the complete file path.

For example:

```
module test();  
initial  
    bank_reload("$cwd/f.txt");  
endmodule
```

The SystemVerilog LRM does not support using environment variables in Verilog source code. Therefore, by default, VCS does not support using environment variables in Verilog source code. You should either use the `-x1rm env_expand` option or use the `ELAB_EXPAND_ENV` variable in the `synopsys_sim.setup` file to use environment variables in the Verilog source code.

For example:

```
% vlogan t.sv -sverilog -x1rm env_expand
```

Content of `synopsys_sim.setup` file:

```
ELAB_EXPAND_ENV=TRUE
```

B

Analysis Utilities

This chapter describes the following utilities, which you can use during the VCS analysis process:

- [The vhdlan Utility](#)
 - [Using Smart Order](#)
 - [The vlogan Utility](#)
-

The vhdlan Utility

The `vhdlan` utility analyzes VHDL source files and produces intermediate files for simulation. It checks for syntactic errors and if it finds any, generates error messages for them. The `vhdlan` utility uses the `synopsys_sim.setup` file to determine the logical-to-physical mapping of VHDL libraries.

Syntax

`vhdlan [vhdlan_options] <VHDL_filename_list>`

where, `vhdlan_options` are as follows:

`-help`

Prints usage information for `vhdlan`.

`-nc`

Suppresses the Synopsys copyright message.

`-q`

Suppresses compiler messages.

`-version`

Prints the version number of `vhdlan` and exits without running analysis.

`-4state`

Turns on Compact Data Representation (CDR) optimization. This option benefits designs that use std logic/ulogic vectors as 4state (for example, X, Z, 0, 1). Values other than X, Z, 0, 1 are reduced to the following:

'H' is converted to '1'

'L' is converted to '0'

'W' and '-' are converted to 'X'

If `-verbose` mode is specified, a warning will be issued about the values conversions performed if the information is statically visible in the design during analysis.

Performance benefits are seen because internally these values are represented in a compact form allowing for better data locality.

Note:

`-4state` optimizes the code and hence debugging is turned off under this mode.

`-work library`

Maps a design library name to the logical library name `WORK`, which receives the output of `vhdlan`. Mapping with the command-line option overrides any assignment of `WORK` to another library name in the setup file.

`library` can also be a physical path that corresponds to a logical library name defined in the setup file.

`-vhdl87`

Lets you analyze non-portable VHDL code that contains object names that are now, by default, VHDL-93 reserved words. VCS is VHDL-93 compliant.

`-vhdl02`

Lets you analyze the VHDL 2002 protected types. For more information, see chapter [Support for VHDL 2002, 2008 and 2019](#).

`-vhdl08`

Lets you analyze the VHDL 2008 constructs. For more information, see chapter [Support for VHDL 2002, 2008 and 2019](#).

`-output outfile`

Redirects standard output from VCS analysis (that usually goes to the screen) to the file you specify as `outfile`.

`-list`

Creates a list file (`.lis`) containing the VHDL source code of the analyzed files, the names of the analyzed design units, and warning or error messages produced during analysis.

`-sva`

Enables SVAs inlined in the VHDL source code.

`-sv_opts "vlog_opts_to_SVAs"`

Specifies Verilog options for SVAs inlined in the VHDL source code.

`-optimize`

It improves simulation performance by generating optimized code, eliminating the following VHDL checks:

- Arithmetic overflow
- Constraint checks
- Array size compatibility at assignment
- Subscripts out of bounds
- Negative exponents to integer

This option overrides the value of the `OPTIMIZE` variable specified in the `synopsys_sim.setup` file. Use this option after you have successfully debugged the design and want to achieve better simulation performance. This option is on by default. The `-no_opt` option takes precedence over the `-optimize` option on the `vhdlan` command line.

`-no_opt`

Enables all VHDL language checks by canceling the effect of the `-optimize` option. Use this option while debugging the VHDL source files in your design.

The `-no_opt` option takes precedence over the `-optimize` option on the `vhdlan` command line.

`-ccpath path`

Specifies the C compiler that the Analyzer must use for compiling the code from VHDL to C. This option has already been set for the SPARC OS5 platform to use the C compiler included with this software. We recommend that you do not change this value. This option overrides the value of the `CS_CCPATH_SEARCH` variable specified in the `synopsys_sim.setup` file.

`-ccflags "flags"`

Specifies the flags that vhdlan passes to the C compiler. The default flags are set in the synopsys_sim.setup file. This option overrides the value of the CS_CCFLAGS_\$ARCH variable specified in the synopsys_sim.setup file.

`-xlrn`

Enables VHDL features beyond those described in LRM.

`-f optionsfile`

Specifies an *optionsfile* that expands the vhdlan command-line options.

`-functional_vital`

Specifies generating code for functional VITAL simulation mode.

`-full64`

Enables compilation and simulation in 64-bit mode.

`-no_functional_vital`

Specifies generating code for full-timing VITAL simulation mode.

`-keep_vital_ifs`

Turns off some of the aggressive functional VITAL optimizations related to `if` statements in Level 0 VITAL cells.

`-keep_vital_path_delay`

Preserves the calls to `VitalPathDelay`. Use this option if non-zero assignments to the outputs is required to preserve correct functionality.

`-keep_vital_wire_delay`

Preserves the calls to `VitalWireDelay`. Use this option if delays on the inputs are required to preserve correct functionality.

`-keep_vital_signal_delay`

Preserves the calls to `VitalSignalDelay`. Use this option if delays on signals are required to preserve correct functionality.

`-keep_vital_timing_checks`

Preserves the timing checks within the VITAL cell.

`-keep_vital_primitives`

Preserves calls to VITAL primitive subprograms.

`-sva`

Enables SVAs inlined in your VHDL code.

```
-sv_opts "vlog_opts_to_SVAs"
```

Specifies Verilog options like `timescale`, `+define+macro` to SVAs inlined in your VHDL code.

For example:

```
% vhdlan -sva -sv_opts "+define+SVA1" file1.vhd
```

`VHDL_filename_list`

Specifies the VHDL source file names to be analyzed. If you do not provide an extension, `.vhd` is assumed.

Note:

The maximum identifier name length is 250 for package, package body, and configuration names. The combined length of an entity name plus architecture name must not exceed 250 characters as well. All other VHDL identifier names and string literals do not have a limitation.

Using Smart Order

The `smart_order` option with `vhdlan`, allows you to automatically identify the file order dependencies internally and then do file-by-file analysis of all VHDL files passed to it. They are ordered as per the dependencies of design units contained within them.

Identifying the dependencies between design units, establishing an order for design files that contain them, and then running `vhdlan` to analyze these files is a difficult and time-consuming process in most cases.

According to *IEEE Standard VHDL 1076-2008 LRM*, Section 11.4, VHDL design units must be analyzed in the order of their dependency, that is, before analyzing a particular unit, its dependent unit must be analyzed. For example, if `unit1` is dependent on `unit2`, then `unit2` must be analyzed before analyzing `unit1`.

Note:

By default, design files that you input to `vhdlan` are analyzed in the order in which they are listed in the command line.

Use Model

Order-independent analysis of VHDL files using the `smart_order` option:

Specify the `-smart_order` option in the `vhdlan` command line or set `SMART_ORDER=TRUE` in the `synopsys_sim.setup` file.

Syntax:

```
vhdlan -smart_order [vhdlan_options] VHDL_filelist
```

Example:

```
vhdlan -smart_order -work lib bottom.vhd mid.vhd top.vhd
vhdlan -smart_order -work lib *.vhd
vhdlan -smart_order -work lib t*.vhd
vhdlan -smart_order -f flist
```

- Using the `smart_script` option along with `smart_order`:

When used along with the `-smart_order` option, the `-smart_script` option generates a re-analysis script, which is a complete `vhdlan` command line. The script includes an ordered file list and all options (except for the `-file` option since it is expanded and replaced) specified in the original `vhdlan` command line.

Specify `-smart_script` followed by a user-specified file name in the `vhdlan` command line. The `-smart_script` option must be used with the `-smart_order` option to generate the re-analysis script.

Syntax:

```
vhdlan -smart_order -smart_script script_name [vhdlan_options]
VHDL_filelist
```

Example:

```
vhdlan -smart_order -smart_script ana.sh -work lib bottom.vhd mid.vhd
top.vhd
```

```
vhdlan -smart_order -smart_script ana.sh -work lib *.vhd
```

Note:

The ordered file list dumped by `smart_script` can be re-used directly with the `vhdlan` as the ordered file list, thereby avoiding the need to use `-smart_order -smart_script` often.

Limitations

Following are the limitations of the `smart_order` option:

- You cannot resolve a design unit that was analyzed into one logical library, but referenced with another logical library prefix (these two libraries point to a same UNIX path) when using the `smart_order` option. For example:

```
%vhdlan -work lib1 leaf.vhd top.vhd
```

`leaf` is referred in `top` as follows:

```
Library lib2;
Use lib2.leaf;
```

- If there is no explicit configuration for a component instance, then this component instance must have a port map clause when it is defined.
- Identifying file order dependencies across different logical libraries is not supported.

Note:

- The primary design units (package, entity, and configuration) in the listed design files must have unique names, otherwise, `vhdlan` generates an error message and aborts sorting of the design files.
- For Mixed HDL Designs (Verilog + VHDL), you need to analyze all Verilog files that are instantiated in VHDL first, otherwise, `vhdlan` generates warning messages for unresolved references. This is a general flow for Mixed HDL designs, and is not specific when `smart_order` is used. The `smart_order` option does not identify Verilog dependencies.

The vlogan Utility

VCS uses the `vlogan` utility to analyze Verilog portions of a design instantiated within a VHDL design.

The syntax of the `vlogan` command line is as follows:

```
vlogan [vlogan_options] Verilog_source_filename
```

Here, the `vlogan_options` are:

`-help`

Displays a succinct description of the most commonly used compile-time and runtime options.

`-nc`

Suppresses the Synopsys copyright message.

`-q`

Suppresses compiler messages.

`-f filename`

Specifies a file that contains a list of path names to source files and required analysis options.

You can use Verilog comment characters such as `//` and `/* */` to comment out entries in the file.

Note that the following restrictions apply to the contents of this file:

- You can only specify the following analysis options that begin with a minus(-) character:

<code>-f</code>	<code>-l</code>	<code>-y</code>
<code>-u</code>	<code>-v</code>	

- You cannot specify escape characters and meta characters like `$`, `'`, and `!`.

Note:

- The maximum line length in the specified file `filename` should be less than 1024 characters. VCS truncates the line exceeding this limit, and issues a warning message.

`-fauto`

Changes the lifetime of all functions that do not specify an explicit lifetime to automatic. Using `-fauto` is a best practice because it gives simulation the same semantics as synthesis.

`-file filename`

Specifies a file that contains a list of path names to source files and required analysis options.

You can use this option to overcome the limitation of the `-f` option related to the compile-time options that begin with a minus(-) character.

`-full64`

Enables compilation and simulation in 64-bit mode.

`-ID`

Displays the hostid or dongle ID for your machine.

`-ignore keyword_argument`

Suppresses warning messages depending on which keyword argument is specified. The keyword arguments are as follows:

`unique_checks`

Suppresses warning messages about `unique if` and `unique case` statements.

`priority_checks`

Suppresses warning messages about `priority if` and `priority case` statements.

`all`

Suppresses warning messages about `unique if`, `unique case`, `priority if` and `priority case` statements.

`-l filename`

Specifies a log file where VCS records compilation messages and runtime messages if you include the `-R` option.

`-liblist logical_lib1+logical_lib2+...logical_libn`

It specifies the library search order for resolving imported package definitions. The `vlogan -liblist` option restricts the libraries in which `vlogan` should search for resolving package references found while analyzing.

If the `-liblist` option is not included, `vlogan` searches all logical libraries listed in the `synopsys_sim.setup` file.

When using multiple `vlogan` commands with same `-work` logical library, run the commands sequentially, and if one command uses `-liblist`, then ensure that all the remaining `vlogan` commands are using the same `-liblist` argument list, as shown in the following example:

```
%vlogan a.v -work shared_lib -liblist shared_lib+ovm_lib+common_lib
%vlogan b.v -work shared_lib -liblist shared_lib+ovm_lib+common_lib
%vlogan c.v -work shared_lib -liblist shared_lib+ovm_lib+common_lib
```

`-location`

Displays the location of the `vlogan` installation.

`-libmap filename`

Specifies a library mapping file.

`-notice`

Enables verbose diagnostic messages related to ports coerced to inout, ports coerced to input, and ports coerced to output.

`-ntb`

Enables the use of the OpenVera testbench language constructs described in the *OpenVera Language Reference Manual: Native Testbench*.

`-ntb_define macro`

Specifies any OpenVera macro name on the command line. You can specify multiple macro names using the plus (+) character.

`-ntb_fileext .ext`

Specifies an OpenVera file name extension. You can specify multiple file name extensions using the plus (+) character.

`-ntb_incdir directory_path`

Specifies the include directory path for OpenVera files. You can specify multiple include directories using the plus (+) character.

`-ntb_opts keyword_argument`

The keyword arguments are as follows:

`ansi`

Preprocesses the OpenVera files in the ANSI mode. The default preprocessing mode is the Kernighan and Ritchie mode of the C language.

`check`

Reports errors, during compilation or simulation, when there is an out-of-bound or illegal array access.

`dep_check`

Enables dependency analysis and incremental compilation. Detects files with circular dependencies and issues an error message when VCS cannot determine which file to compile first.

`no_file_by_file_pp`

By default, VCS does file-by-file preprocessing on each input file, feeding the concatenated result to the parser. This argument disables this behavior.

`print_deps`

Tells VCS to display the dependencies for the source files. Enter this argument with the `dep_check` argument.

`rvm`

Use `rvm` when RVM or VMM is used in the testbench.

Example: `vlogan vmm_test.sv -sverilog -ntb_opts rvm`

For more information, refer to the [Using VMM with VCS](#) section.

`tb_timescale=value`

Specifies an overriding timescale for the testbench, whenever the required testbench timescale is different from that of the design. It must be used in conjunction with the `-timescale` option that specifies the timescale for the design.

If the required testbench timescale is different from the design or DUT timescale, then both the testbench timescale and the DUT timescale must be passed during VCS compilation.

Example:

The following command specifies a required testbench timescale of 10ns/10ps and a design timescale of 1ns/1ps:

```
%> vlogan -ntb_opts tb_timescale=10ns/10ps      -timescale=1ns/1ps file.sv
tokens
```

Pre-processes the OpenVera files to generate two files, `tokens.vr` and `tokens.vrp`.

The `tokens.vr` file contains the preprocessed result of the non-encrypted OpenVera files, while the `tokens.vrp` file contains the preprocessed result of the encrypted OpenVera files. If there is no encrypted OpenVera file, VCS sends all the OpenVera preprocessed results to the `tokens.vr` file.

`use_sigprop`

Enables the signal property access functions. For example, `vera_get_ifc_name()`.

`vera_portname`

Specifies the following:

- The Vera shell module name is named `vera_shell`.
- The interface ports are named `ifc_signal`.
- Bind signals are named, for example, as: `\ifc_signal[3:0]`.

`-platform`

Returns the name of the platform directory in your VCS installation directory.

-sv_pragma

Analyzes SystemVerilog Assertions that follow the `sv_pragma` keyword in a single line or multi-line comment.

-p1800_macro_expansion

This option is used for LRM compliance to support macro expansion. The option produces result that is accurate especially for SystemVerilog macros.

The syntax is:

```
% vlogan [vlogan_options] test.sv -sverilog
-p1800_macro_expansion
```

For example, consider the following test case, `test.sv`:

```
module top;
logic [3:0] addr0_for_bank0='d10;
`define VAR(ANUM,BNUM) addr``ANUM``for_bank``BNUM
`define NAME(STR) $display(`"``\``"STR`\``" is %d\n`",STR);
`define ARG addr0_for_bank0

initial begin
`NAME(`VAR(0,0));
`NAME(`ARG)
end
endmodule
```

If you run the test case without the `-p1800_macro_expansion` option, VCS generates the following output:

```
"`VAR(0,0)" is 10
"addr0_for_bank0" is 10
```

If you run the test case with the `-p1800_macro_expansion` option, VCS generates the following output:

```
"addr0_for_bank0" is 10
"addr0_for_bank0" is 10
```

Note:

The double quote are only used in macro definition body.

`-timescale=time_unit/time_precision`

This option enables you to specify the timescale for the source files that do not contain `'timescale` compiler directive and precede the source files that do.

Do not include spaces when specifying the arguments to this option as shown in the following example:

```
% vlogan -timescale=1ns/1ns file1.v file2.v file3.v
```

```
-override_timescale=time_unit/time_precision
```

Overrides the time unit and precision unit for all the `'timescale` compiler directives in the source code and, like `-timescale`, provides a timescale for all module definitions that do not have a `'timescale` compiler directive.

```
-noinherit_timescale
```

This is a VCS or vlogan option that allows you to specify a global timescale within the compilation unit of any source file that does not have an explicit `'timescale` directive.

If you specify the `-noinherit_timescale` option in the command line and there is no time unit/time precision specification present inside the module, program, interface, or package definition, then the time unit/time precision is determined using the following rules:

- If the module or interface definition is nested, then the time unit/time precision shall be inherited from the enclosing module or interface.
- Else, if `'timescale` directive is previously specified within the file, then the time unit/time precision shall be set to that of the `'timescale` directive.
- Else, if the time unit/time precision is specified outside all other declarations within a compilation unit, then the time unit/time precision shall be set to that specification.
- Else, the global time unit/time precision specified by the `-noinherit_timescale` option shall be used.

Similarly in presence of other timescale options specified in this document, the following is the precedence:

```
-override_timescale > -noinherit_timescale > -timescale/-unit_timescale
```

Therefore, when the combination of options are used, the behavior is as follows:

- The `-override_timescale` option has the highest precedence. Therefore, the timescale specified by the `-noinherit_timescale` option is overridden by the `-override_timescale` option.
- If the `-timescale` (or the `-unit_timescale`) and the `-noinherit_timescale` options are used in the same command line, timescale from the `-noinherit_timescale` option shall be used and `-timescale` (or `-unit_timescale`) option is ignored.
- In case of three-step flow, if the `-timescale` (or `-unit_timescale`) and the `-noinherit_timescale` options are used in different command line, such as one is in `vlogan` and another is in `vcs` command line, timescale specified at `vlogan`

stage cannot be overridden at `vcs` stage (except for the case when you use the `-overridde_timescale` option).

Figure 203 Example for -noinherit_timescale Option

a.v	b.v	c.v
<pre>module A; ... endmodule module B; ... endmodule</pre>	<pre>`timescale 1ns/1ns module X; ... endmodule `timescale 1ns/1ns module Y; ... endmodule</pre>	<pre>module P; timeunit 100ps; timeprecision 1ps; ... endmodule module Q; ... endmodule</pre>

Any of the above commands result into the following timescale:

- module A - 10ps/1ps
- module B - 10ps/1ps
- module P - 100ps/1ps
- module Q - 10ps/1ps
- module X - 1ns/1ns
- module Y - 1ns/1ns

Key Points to Note

- The files that are read using a ``include` directive inherit the timescale of the file containing the ``include` directive. That is, the global timescale is only applied to files that are not read using an ``include` directive.
- Once the file is compiled or analyzed, the timescale of a module is not changed. This means that the timescale of a logic library is determined when the library is compiled, not when one of its modules is incorporated using the `-y` option.

`+delay_mode_path`

Uses only the delay specifications in module path delays in specify blocks. Overrides all the delay specifications on all gates, switches, and continuous assignments to zero.

`+delay_mode_zero`

Removes delay specifications on all gates, switches, continuous assignments, and module paths.

`+delay_mode_unit`

Overrides all the delay specifications in module path delays in specify blocks to zero delays. Overrides all the delay specifications on all gates, switches, and continuous assignments to the shortest time precision argument of all the '`timescale`' compiler directives in the source code. The default time unit and time precision argument of the '`timescale`' compiler directive is 1s.

`+delay_mode_distributed`

Overrides all the delay specifications in module path delays in specify blocks to zero delays. Uses only the delay specifications on all gates, switches, and continuous assignments.

`-u`

Changes all characters in identifiers to uppercase.

`-V[t]`

Enables warning messages and displays the time used by each command.

`-v library_file`

Specifies a Verilog library file to search for module definitions.

`-y library_directory`

Specifies a Verilog library directory to search for module definitions. Use this option with `+libext+extension`. See below for the description of `+libext+extension`.

`-work VHDL_logical_library`

Specifies creating the VERILOG directory and writing intermediate files in the physical directory associated with this logical library.

`+define+macro`

Defines a text macro. Test for this definition in your Verilog source code using the '`ifdef`' compiler directive.

`+libext+extension+`

Specifies that VCS searches only for files with the specified file name extensions in a library directory. You can specify more than one extension, separating the extensions with the plus (+) character. For example, `+libext+.v+.v+` specifies searching for files with either the `.v` or `.V` extension in a library. The order in which you add file name extensions

to this option does not specify an order in which VCS searches files in the library with these file name extensions.

+lint=[no] ID|none|all

Enables messages that tell you when your Verilog code contains something that is bad style but is often used in designs.

Here:

no

Specifies disabling lint messages that have the ID that follows. There is no space between the keyword `no` and the ID.

none

Specifies disabling all lint messages. IDs that follow in a comma separated list are exceptions.

all

Specifies enabling all lint messages. IDs that follow preceded by the keyword `no` in a comma separated list are exceptions.

The following examples show how to use this option:

- Enable all lint messages except the message with the GCWM ID: `+lint=all,noGCWM`
- Enable the lint message with the NCEID ID: `+lint=NCEID`
- Enable the lint messages with the GCWM and NCEID IDs: `+lint=GCWM,NCEID`
- Disable all lint messages. This is the default. `+lint=none`

The syntax of the `+lint` option is very similar to the syntax of the `+warn` option for enabling or disabling warning messages. Additionally, some of the messages of these options have the same ID. This is because when there is a condition in your code that causes VCS to display both a warning and a lint message, the corresponding lint message contains more information than the warning message and can be considered more verbose.

The number of possible lint messages is not large. They are as follows:

`Lint-[IRIMW] Illegal range in memory word`

`Lint-[NCEID] Non-constant expression in delay`

`Lint-[GCWM] Gate connection width mismatch`

`Lint-[CAWM] Continuous Assignment width mismatch`

```

    Lint-[IGSFPG] Illegal gate strength for pull gate
    Lint-[TFIPC] Too few instance port connections
    Lint-[IPDP] Identifier previously declared as port
    Lint-[PCWM] Port connect width mismatch
    Lint-[VCDE] Verilog compiler directive encountered
+incdir+directory

```

Specifies the directories that contain the files you specified with the '`include`' compiler directive. You can specify more than one directory, separating each path name with the "+" character.

`+nowarnTFMPC`

Suppress the "Too few module port connections" warning messages during Verilog Compilation.

`-sverilog`

Enables the analysis of SystemVerilog source code.

`+systemverilogext+ext`

Specifies a file name extension for SystemVerilog source files. If you use a different file name extension for the SystemVerilog part of your source code and you use this option, the `-sverilog` option has to be omitted.

This compile-time option also works similar to the `-sverilog` option in which it enables SystemVerilog LRM (IEEE Std 1800-2012) rules for all the source files on the `vcs` command line and not only the files with the specified extension.

`+verilog2001ext+ext`

Specifies a file name extension for Verilog 2001 source files.

`+verilog1995ext+ext`

Specifies a file name extension for Verilog 1995 files. Using this option allows you to write Verilog 1995 code that would be invalid in Verilog 2001 or SystemVerilog code, such as using Verilog 2001 or SystemVerilog keywords, like `localparam` and `logic`, as names.

Note:

Do not specify the `+systemverilogext+ext`, `+verilog2001ext+ext`, and `+verilog1995ext+ext` options on the same command line.

`-extinclude`

If a source file for one version of Verilog contains the `'include` compiler directive, `vlogan` by default compiles the included file for the same version of Verilog, even if the included file has a different filename extension. If you want `vlogan` to compile the included file with the version specified by its extension, enter this option. The following code examples show using this option:

If source file `a.v` contains the following:

```
`include "b.sv"
module a();
reg ar;
endmodule
```

and if source file `b.sv` contains the following:

```
module b();
logic ar;
endmodule
```

`vlogan` compiles `b.sv` for SystemVerilog with the following command line:

```
vlogan a.v +systemverilogext+.sv -extinclude
+warn=[no] ID|none|all
```

Uses warning message IDs to enable or disable display of warning messages. In the following warning message, the text string `TFIIPC` is the message ID:

```
Warning-[TFIIPC] Too few instance port connections
```

The syntax of this option is as follows:

```
+warn=[no] ID|none|all,...
```

Where:

-
- | | |
|------|-----------------------------------------------------------------------------------------------------------------------------------------------------|
| no | Specifies disabling warning messages with the ID that follows. There is no space between the keyword <code>no</code> and the ID. |
| none | Specifies disabling all warning messages. IDs that follow, in a comma-separated list, specifies exceptions. |
| all | Specifies enabling all warning messages. IDs that follow preceded by the keyword <code>no</code> , in a comma-separated list, specifies exceptions. |
-

The following are examples that show how to use this option:

+warn=noIPDW	Enables all warning messages except the warning with the IPDW ID.
--------------	-------------------------------------------------------------------

Appendix B: Analysis Utilities

The vlogan Utility

+warn=none,TFIPC	Disables all warning messages except the warning with the TFIPC ID.
+warn=noIPDW,noTFIPC	Disables the warning messages with the IPDW and TFIPC IDs.
+warn=all	Enables all warning messages. This is the default.

`Verilog_source_filename`

Specifies the name of the Verilog source file.

C

Compilation/Elaboration Options

The `vcs` command performs compilation/elaboration of your design and creates a simulation executable. Compiled event code is generated and used by default. The generated simulation executable, `simv`, can then be used to run multiple simulations.

This section describes the `vcs` command and related options.

Syntax for two-step flow:

```
% vcs source_files [source_or_object_files] [options]
```

Here:

`source_files`

The Verilog or OVA source files for your design separated by spaces.

`source_or_object_files`

Optional C files (`.c`), object files (`.o`), or archived libraries (`.a`). These are DirectC or PLI applications that you want VCS to link into the binary executable file along with the object files from your Verilog source files. When including object files include the `-cc` and `-ld` options to specify the compiler and linker that generated them.

`options`

Compile-time options that control how VCS compiles your design.

Syntax for three-step flow:

```
% vcs [libname.]design_unit [options]
```

Here:

`[libname.]design_unit`

Specifies the `design_unit` you want to simulate, with an optional logical library name. By default, the `WORK` library is assumed.

The `design_unit` can be one of the following:

`cfgname`

Name of the top-level event configuration to be simulated.

entname [__archname]

Name of the entity and architecture to be simulated. By default, `archname` is the most recently analyzed architecture.

module

Name of the top-level Verilog module to be simulated

options

Elaboration options that control how VCS elaborates your design.

This appendix lists the following:

- [Option for Code Generation](#)
- [Options for Accessing Verilog Libraries](#)
- [Options for Incremental Compilation](#)
- [Option to Save Disk Space](#)
- [Options for Help](#)
- [Option for SystemVerilog](#)
- [Option to Enable Optimized Debug Capabilities and FSDB Dumping for Testbench Root Cause Analysis \(TBRCA\)](#)
- [Options for SystemVerilog Assertions](#)
- [Options to Enable Compilation of OVA Case Pragmas](#)
- [Options for Native Testbench](#)
- [Options for Different Versions of Verilog](#)
- [Option for Initializing Verilog Variables, Registers and Memories with Random Values](#)
- [Option for Initializing Verilog Variables, Registers and Memories in Selective Parts of a Design](#)
- [Options for Selecting Register or Memory Initialization](#)
- [Options for Using Radiant Technology](#)
- [Options for Starting Simulation Right After Compilation](#)
- [Options for Specifying Delays and SDF Files](#)
- [Options for Compiling an SDF File](#)
- [Options for Specify Blocks and Timing Checks](#)

- Options for Pulse Filtering
- Options for Negative Timing Checks
- Options for Profiling Your Design
- Options to Specify Source Files and Compilation/Elaboration Options in a File
- Options for Compiling Runtime Options Into the Executable
- Options for PLI Applications
- Options to Enable the VCS DirectC Interface
- Options for Flushing Certain Output Text File Buffers
- Options for Simulating SWIFT VMC Models and SmartModels
- Options for Controlling Messages
- Option to Run VCS in Syntax Checking Mode
- Options for Cell Definition
- Options for Licensing
- Options for Controlling the Linker
- Options for Controlling the C Compiler
- Options for Source Protection
- Options for Mixed Analog/Digital Simulation
- Unified Option to Change Generic and Parameter Values
- Options for Changing Parameter Values
- Checking for x and z Values in Conditional Expressions
- Options for Detecting Race Conditions
- Options to Specify the Time Scale
- Option to Exclude Environment Variables During Timestamp Checks
- Options for Overriding Generics and Parameters
- Global -check_all Option
- Option to Enable Bounds Check at Compile-Time
- Runtime Checks for Out-of-Bounds Condition

- Option to Enable Extra Runtime Checks in VHDL
- Options to Detect Multiple Conflicts on Buses
- Gate-Level Simulations
- Options for Additional Multiple Driver Checks
- Option for Function Call Evaluation at Time Zero
- General Options

Option for Code Generation

`-mcg`

Enables mixed code generation model in VCS backend. Part of code is aggressively optimized by the available C compilers.

Options for Accessing Verilog Libraries

`-v filename`

Specifies a Verilog library file. VCS looks in this file for definitions of the module and UDP instances that VCS found in your source code, but for which it did not find the corresponding module or UDP definitions in your source code.

`-y directory`

Specifies a Verilog library directory. VCS looks in the source files in this directory for definitions of the module and UDP instances that VCS found in your source code, but for which it did not find the corresponding module or UDP definitions in your source code. VCS looks in this directory for a file with the same name as the module or UDP identifier in the instance (not the instance name). If it finds this file, VCS looks in the file for the module or UDP definition to resolve the instance.

If multiple `-y` options are on the `vcs` command line, VCS starts searching the directory passed with the first `-y` option, then second and so on.

For example:

If `rev1/cell.v`, `rev2/cell.v` and `rev3/cell.v` all exist and define the module `cell()`, and you issue the following command:

```
% vcs -y rev1 -y rev2 -y rev3 +libext+.v top.v
```

VCS picks `cell.v` from `rev1`.

However, if the `top.v` file has a ``uselib` compiler directive as shown below, then ``uselib` takes priority:

```
//top.v
`uselib directory = /proj/libraries/rev3
//rest of top module code
//end top.v
```

In this case, VCS will use `rev3/cell.v` when you issue the following command:

```
% vcs -y rev1 -y rev2 +libext+.v top.v
```

Include the `+libext` compile-time option to specify the file name extension of the files you want VCS to look for in these directories.

`+libext+extension+`

Specifies that VCS searches only for files with the specified file name extensions in a library directory. You can specify more than one extension, separating the extensions with the plus (+) character. For example, `+libext+.v+.v+` specifies searching for files with either the `.v` or `.V` extension in a library. The order in which you add file name extensions to this option does not specify an order in which VCS searches files in the library with these file name extensions.

`+liborder`

Specifies searching for module definitions for unresolved module instances through the remainder of the library where VCS finds the instance, then searching the next and then the next library on the `vcs` command line before searching in the first library on the command line.

`+librescan`

Specifies always searching libraries for module definitions for unresolved module instances beginning with the first library on the `vcs` command line.

`+liborder` and `+librescan` switches on elaboration command line will have impact only when the user specifies `-y/-v` on elaboration command line.

`+libverbose`

Tells VCS to display a message when it finds a module definition in a source file in a Verilog library directory that resolves a module instantiation statement that VCS read in your source files, a library file, or in another file in a library directory. The message is as follows:

```
Resolving module "module_identifier"
```

By default, VCS does not display this message when it finds a module definition in a Verilog library file that resolves a module instantiation statement.

VCS also displays verbose information of the module instances while resolving library modules under the `+libverbose` option. If you do not want this verbose information to be displayed while resolving library modules, then use the `+libverbose+1` option to avoid messages related to the library modules.

Options for Incremental Compilation

`-Mdirectory=directory`

Specifies the incremental compile directory. The default name for this directory is `csrc`, and its default location is your current directory. You can substitute the shorter `-Mdir` for `-Mdirectory`.

`-Mlib=dir`

This option provides VCS with a central place to look for the descriptor information before it compiles a module and a central place to get the object files when it links together the executable. This option allows you to use the parts of a design that have been already tested and debugged by other members of your team without recompiling the modules for these parts of the design.

You can specify more than one place for VCS to look for descriptor information and object files by providing multiple arguments with this option.

Example:

```
% vcs design.v -Mlib=/design/dir1 -Mlib=/design/dir2
```

Or, you can specify more than one directory with this option, using a colon (:) as a delimiter between them, as shown below:

```
% vcs design.v -Mlib=/design/dir1:/design/dir2
```

`-Mupdate [=0]`

By default, VCS overwrites the makefile between compilations. If you wish to preserve the makefile between compilations, enter this option with the 0 argument.

Entering this argument without the 0 argument, specifies the default condition, incremental compilation and updating the makefile.

`-Mmakep=make_path`

Specifies the make path.

`-noincrcomp`

Disables incremental compilation.

Option to Save Disk Space

`-diskopt`

Saves disk space by compressing various files under the `simv.daidir` directory.

Options for Help

`-h` or `-help`

Lists descriptions of the most commonly used VCS compile and runtime options.

Option for SystemVerilog

`-sverilog`

Enables SystemVerilog constructs specified in the IEEE Standard of SystemVerilog, IEEE Std 1800-2009.

Option to Enable Optimized Debug Capabilities and FSDB Dumping for Testbench Root Cause Analysis (TBRCA)

You can use the `-tbrca` compile-time option with other debug options to enable optimal debug capabilities for TBRCA.

For example:

- `-tbrca -debug_region=cell`: This option applies debug capabilities to both real cell modules and the ports of real cell modules.
- `-tbrca -debug_access+line`: This option enables line debugging. It allows you to use the commands for step or next and line breakpoints.
- `-tbrca -debug_access+w`: This option applies the write or deposit capability to the registers and variables of the entire testbench.

The `-tbrca` option supports normal user dumping with `$fsdbDumpvars` similar to `-debug_access`.

Options for SystemVerilog Assertions

`-ignore keyword_argument`

Suppresses warning messages depending on which keyword argument is specified. The keyword arguments are as follows:

`unique_checks`

Suppresses warning messages about `unique if` and `unique case` statements.

`priority_checks`

Suppresses warning messages about `priority if` and `priority case` statements.

`all`

Suppresses warning messages about `unique if`, `unique case`, `priority if` and `priority case` statements.

You can tell VCS to report errors for both `unique` and `priority` violations with the `-error` compile-time option as shown below:

`-error=UNIQUE`

VCS reports `unique` violations as error conditions.

`-error=PRIORITY`

VCS reports `priority` violations as error conditions.

`-error=UNIQUE,PRIORITY`

VCS reports `unique` and `priority` violations as error conditions.

`-assert keyword_argument`

This runtime option is enabled only when the `-assert enable_diag` option is used at compile/elaboration time.

The following is the list of `keyword_argument` that are enabled when the `-assert enable_diag` compilation/elaboration option is used:

- `-assert success`
- `-assert summary`
- `-assert maxcover`
- `-assert maxsuccess`
- `-assert quiet1`
- `-assert verbose`

The following is the list of assertion options that are enabled when the `-assert enable_hier` compilation/elaboration option is used:

- `-assert hier`
- `-assert maxfail=N`
- `-assert finish_maxfail=N`

The following is the list of assertion options that do not require the `-assert enable_diag` or `-assert enable_hier` option:

- `-assert dumpoff`
 - `-assert nocovdb`
 - `-assert nopostproc`
 - `-assert quiet`
 - `-assert no_fatal_action`
 - `-assert report`
 - `-assert vacuous`
 - `-assert global_finish_maxfail=N`
- `-assert sync_disable`

Converts disabled signal to its sampled value. Use this option when there is a clock and reset changes at the same time, and you have to use the sample value of the disable signal. Therefore, if you have an assertion assert property (@(posedge clk) disable iff(rst) a) and if you apply the `-assert async_disable` option, this assertion behaves like assert property (@(posedge clk) disable iff(\$sampled(rst)) a).

For example,

Consider the following test.v file.

```
module test;
bit clk;
always #5 clk = ~clk;
logic [2:0] a;
logic reset;
initial begin
    reset =1'b1;
    #10
    a = 3'b011;
    #5
    reset =1'b0;
```

```

a = 3'b000;
#5
a = 3'b111;
#20
$finish;
end
// clk is posedge at 5,15,25
// reset value 1,0, 0
// sampled reset 1,1, 0
LOL Assert: assert property (@(posedge clk) disable iff(reset) 0);
endmodule

```

Run the `test.v` file with the following commands:

- % vcs -sverilog test.v -R
- % vcs -sverilog test.v -assert async_disable -R

When you run the `test.v` file without the `-assert async_disable` option, you will get a failure at 15 s. However, with the `-assert async_disable` option, you will not get a failure at 15 s.

`enable_diag`

Enables further control of results reporting with runtime options. The runtime assert options are enabled only if you compile the design with this option.

`funchier`

Enables enhanced reporting for assertions in functions.

`hier=file_name`

You can use the `-assert hier=file_name` compile-time option to specify the configuration file for enabling and disabling SystemVerilog assertions. You can either enable or disable:

Assertions in a module or in a hierarchy.

An individual assertion.

This option works at runtime only for mixed HDL designs.

If you pass an empty assert hier file at compile-time or runtime, VCS generates the CM-ASHR-EF error, as shown below:

```

Error-[CM-ASHR-EF] Empty file
The file 'foo.txt' given to the assertion hier control option was
found, but
it is empty.
Please fix the file and try again.

```

You can convert this error message to a warning message, as shown below, using the `-error=noCM-ASHR-EF` option at compile-time or runtime:

```
Warning-[CM-ASHR-EFW] Empty file
  The file 'foo.txt' given to the assertion hier control option was
  found, but
  it is empty.
  Please fix the file and try again.
```

Note:

If the assertion filter used in assert hier file does not match any assertion in the design, VCS generates the SVA-FILTUNUSED warning message, as shown below, at compile-time or runtime:

```
Warning-[SVA-FILTUNUSED] Unused filter in hier file
'-assert (.)*GLITCH_CBO_TAP_holdinreset' in hier file
.cbo_basic_1213083628.hier does not match any module-instance
hierarchy/assertion.
```

You can convert this warning message to an error message, as shown below, using the `-error=SVA-UNUSEDFLT` option at compile-time or runtime:

```
Error-[SVA-UNUSEDFLT] Unused filter in hier file
'-assert (.)*GLITCH_CBO_TAP_holdinreset' in hier file
.cbo_basic_1213083628.hier does not match any module-instance
hierarchy/assertion.
```

The types of entries that you can specify in the file are as follows:

```
-assert <assertion_name>
```

or

```
-assert <assertion_hierarchical_name>
```

If `assertion_name` is provided, VCS disables the assertions based on wildcard matching of the name in the full design. If `assertion_hierarchical_name` is provided, VCS disables the assertions based on wildcard matching of the name in the particular hierarchy given.

Examples

```
-assert my_assert
```

Disables all assertions with name `my_assert` in the full design.

```
-assert A*
```

Disables all assertions whose name starts with `A` in the full design.

```
-assert *
```

Disables all assertions in the full design.

```
-assert top.INST2.A
```

Disables all assertions whose names start with `A` in the hierarchy `top.INST2`. If assertions whose name starts with `A` exists in inner scopes under `top.INST2`, they are not disabled. This command has effect on assertions only in scope `top.INST2`.

```
+assert <assertion_name> OR +assert <assertion_hierarchical_name>
```

If `assertion_name` is provided, VCS enables the assertions based on wildcard matching of the name in the full design. If `assertion_hierarchical_name` is provided, then VCS enables the assertions based on wildcard matching of the name in the given hierarchy.

Examples

```
+assert my_assert
```

Enables all assertions with name `my_assert` in the full design.

```
+assert A*
```

Enables all assertions whose name starts with `A` in the full design.

```
+assert *
```

Enables all assertions in the full design.

```
+assert top.INST2.A
```

Enables assertion `A` in the hierarchy `top.INST2`.

```
+tree <module_instance_name> OR +tree <assertion_hierarchical_name>
```

If `module_instance_name` is provided, VCS enables assertions in the specified module instance and all module instances hierarchically under that instance. If `assertion_hierarchical_name` is provided, VCS enables the specified SystemVerilog assertion. Wildcard characters can also be used for specifying the hierarchy.

Examples

```
+tree top.inst1
```

Enables the assertions in module instance `top.inst1` and all the assertions in the module instances under this instance.

```
+tree top.inst1.a1
```

Enables the SystemVerilog assertion with the hierarchical name `top.inst1.a1`.

```
+tree top.INST*.A1
```

Enables assertion `A1` from all the instances whose names start with `INST` under module `top`.

`-tree <module_instance_name> or -tree <assertion_hierarchical_name>`

If `module_instance_name` is provided, VCS disables the assertions in the specified module instance and all module instances hierarchically under that instance. If `assertion_hierarchical_name` is provided, VCS disables the specified SystemVerilog assertion. Wildcard characters can also be used for specifying the hierarchy.

Examples

`-tree top.inst1`

Disables the assertions in module instance `top.inst1` and all the assertions in the module instances under this instance.

`-tree top.inst1.a1`

Disables SystemVerilog assertion with the hierarchical name `top.inst1.a1`.

`-tree top.INST*.A1`

Disables assertion `A1` from all the instances whose names start with `INST` under module `top`.

`+module module_identifier`

VCS enables all the assertions in all instances of the specified module, for example:

`+module dev`

VCS enables the assertions in all instances of `module dev`.

`-module module_identifier`

VCS disables all the assertions in all instances of the specified module, for example:

`-module dev`

VCS disables the assertions in all instances of `module dev`.

The specifications are applied serially as they appear in file `file_name` except for `+-assert`. The `+- assert` patterns are always serially applied after all other patterns are done. The result of applying the specifications in this file is that a group of assertions get excluded. The remaining assertions are available for further exclusion by other means,

such as the `$assertoff` system task in the source code. However, the following should be noted:

- The first specification denotes the default exclusion for interpreting the file. If the first specification is a minus(-), then all assertions are included before applying the first and the following specifications. Conversely, if the first specification is a plus(+), then all assertions are excluded prior to applying the first and the following specifications.
- Unlike `-/+module` and `-/+tree` specifications, any assertion excluded by applying `-assert` specification cannot be included by the later specifications in the file.

`enable_hier`

Enables the use of the runtime option `-assert hier=file.txt`, which allows turning assertions on or off.

`filter_past`

For assertions that are defined with the `$past` system task, ignore these assertions when the past history buffer is empty. For instance, at the very beginning of the simulation, the past history buffer is empty. Therefore, the first sampling point and subsequent sampling points should be ignored until the past buffer has been filled with respect to the sampling point.

`offending_values`

Enables the reporting of the values of all variables used in the assertion failure messages. For more information, see [Reporting Values of Variables in the Assertion Failure Messages](#).

`disable`

Disables all SystemVerilog assertions in the design.

`disable_cover`

When you include the `-cm assert` compile-time and runtime option, VCS includes information about cover statements in the assertion coverage reports. This keyword prevents cover statements from appearing in these reports.

`disable_assert`

Disables only the `assert` and `assume` directives without affecting the cover directives. It complements the existing control options which allows you to disable only cover directives or all of the assertions such as `assert/assume/cover`.

`disable_rep_opt`

Specifying a delay or a repetition value greater than 200 in the assertion expression will affect both compile-time and runtime performance. Therefore, VCS optimizes expression and issues a warning message as shown below:

```
Warning-[SVA-LDRF] Large delay or repetition found.
Large delay or repetition found (data ##[0:200] (!data)). VCS will optimize co
mpile time. However it may affect runtime.
"test.v", 8
```

However, if you want to disable the default optimizations, use the following option:

```
-assert disable_rep_opt
```

```
dumpoff
```

Disables the dumping of SVA information in the VPD file during simulation.

```
vpiSeqBeginTime
```

Enables you to see the simulation time that a SystemVerilog assertion sequence starts when using Debussy.

```
vpiSeqFail
```

Enables you to see the simulation time that a SystemVerilog assertion sequence doesn't match when using Debussy.

```
+lint=PWLNT:<max_count>
```

Enables the `PWLNT` lint messages when `$past` is used in the code with the number of clock ticks exceeding 5. You can restrict the number of `PWLNT` lint messages for a particular compilation using the `max_count` argument.

For example, `+lint=PWLNT:10` restricts the number of `PWLNT` lint messages to a maximum of 10 for one compilation.

Options to Enable Compilation of OVA Case Pragmas

```
-ova_enable_case
```

Enables the compilation of OVA case pragmas only, when used without `-Xova` or `-ova_inline`. All inlined OVA assertion pragmas are ignored.

Options for Native Testbench

```
-ntb
```

Enables the use of the OpenVera testbench language constructs described in the *OpenVera Language Reference Manual: Native Testbench*.

`-ntb_define macro`

Specifies any OpenVera macro name on the command line. You can specify multiple macro names using the plus (+) character.

The macro can also be defined to be a fixed number. For example, in the following:

```
program test
{
    integer x;
    x =12345;
printf ("DEBUG==> my value = %d and x = %d\n", MYVALUE, x);
```

When you compile and run:

```
% vcs -ntb -ntb_define MYVALUE=10000 myprog.vr -R
```

This outputs are:

```
DEBUG==> my value = 10000 and x = 12345
```

`-ntb_fileext .ext`

Specifies an OpenVera file name extension. You can specify multiple file name extensions using the plus (+) character.

`-ntb_incdir directory_path`

Specifies the include directory path for OpenVera files. You can specify multiple include directories using the plus (+) character.

`-ntb_noshell`

Tells VCS not to generate the shell file. Use this option when you recompile a testbench.

`-ntb_opts keyword_argument`

The keyword arguments are as follows:

`ansi`

Preprocesses the OpenVera files in the ANSI mode. The default preprocessing mode is the Kernighan and Ritchie mode of the C language.

`check`

Does a bounds check on dynamic type arrays (dynamic, associative, queues) and issues an error at runtime.

`check=dynamic`

Same as check. Does a bounds check on dynamic type arrays (dynamic, associative, queues) and issues an error at runtime.

`check=fixed`

Does a bounds check only on fixed size arrays and issues an error at runtime.

`check=all`

Does a bounds check on both fixed size and dynamic type arrays and issues errors at runtime.

The following error messages are displayed during runtime:

ERROR-[DT-OBAE] Out-of-bound access for queues

This error message is displayed, if a queue element is accessed with an out-of-bounds index condition.

For example,

```
module tb();
int temp; // temp signal
int int_queue[$] = { 1, 2, 3}; //Queue
initial
begin
  //Queue
  temp = int_queue[9];
end
endmodule
```

The following error message is displayed:

```
Error-[DT-OBAE] Out of bound access
test2.sv, 9
Out of bound access on smart queue (size:3, index:9)
Simulation time = 0
Please make sure that the index is positive and less than size.
```

ERROR-[DT-OBAE] Out-of-bound access for dynamic arrays

This error message is displayed, if a dynamic array element is accessed with an out-of-bounds index condition.

For example,

```
module tb();
reg some_bit; // temp signal
reg nibble[]; // Dynamic array
int int_queue[$] = { 1, 2, 3}; //Queue
initial
begin
  //Dynamic array
```

Appendix C: Compilation/Elaboration Options Options for Native Testbench

```

    nibble = new[3]; // Create a 3-element array.
    some_bit = nibble[3];
end
endmodule

```

The following error message is displayed:

```

Error-[DT-OBAE] Out of bound access
test2.sv, 11
Out of bound access on dynamic array (size:3, index:3)
Simulation time = 0
Please make sure that the index is positive and less than size.

```

ERROR-[OBA] Out-of-bound access for fixed size unpacked array

This error message is displayed, if a fixed size unpack array is accessed with an out-of-bounds index condition.

For example,

```

module tb();
reg [1:0] fifo_data[2:0]; // fifo memory
reg [1:0] some_signal; // temp signal
initial
begin
  //Unpacked dimension
  some_signal = fifo_data[3];
end
initial $display("some_signal = %0d",some_signal);
endmodule

```

The following error message is displayed:

```

Error-[OBA] Out of bound access
test2.sv, 9
Out of bound access on array (index value: 3)
Simulation time = 0

```

ERROR-[DT-IV] Out-of-bound access for fixed size unpacked array

This error message is displayed, if a fixed size unpacked array element is accessed with an index value X or Z.

For example,

```

module tb();
reg [17:0] fifo_data[255:0]; // fifo memory
reg [7:0] rd_ptr = 8'bxxxxxxxx;
wire [7:0] some_signal; // temp signal

assign some_signal = fifo_data[rd_ptr];
initial $display("some_signal = %0d", some_signal);
endmodule

```

The following error message is displayed:

```
Error-[DT-IV] Illegal value
test2.sv, 6
Illegal index value specified for array
Simulation time = 0
Please make sure that the value is properly initialized with none of the
bits set to x or z.
```

`dep_check`

Enables dependency analysis and incremental compilation. Detects files with circular dependencies and issues an error message when VCS cannot determine which file to compile first.

`no_file_by_file_pp`

By default, VCS does file-by-file preprocessing on each input file, feeding the concatenated result to the parser. This argument disables this behavior.

`print_deps`

Tells VCS to display the dependencies for the source files on the screen. Enter this argument with the `dep_check` argument.

`rvm`

Use `rvm` when RVM or VMM is used in the testbench. For more information, refer to the [Using VMM with VCS](#) section.

`sv_fmt` The default padding used in displayed or printed strings is right padding. The `sv_fmt` option specifies left padding. For example, when `-ntb_opts sv_fmt` is used, the result of

```
$display("%10s", "my_string");
```

is to put 10 spaces to the left of `my_string`.

To specify right padding when `-ntb_opts sv_fmt` is used, put a dash before the number of spaces. For example, the result of

```
$display("%-10s", "my_string");
```

is to put 10 spaces to the right of `my_string`.

`tb_timescale=value`

Specifies an overriding timescale for the testbench, whenever the required testbench timescale is different from that of the design. It must be used in conjunction with the `-timescale` option that specifies the timescale for the design.

If the required testbench timescale is different from the design or DUT timescale, then both the testbench timescale and the DUT timescale must be passed during VCS compilation.

Example:

The following command specifies a required testbench timescale of 10ns/10ps and a design timescale of 1ns/1ps:

```
%> vcs -ntb_opts tb_timescale=10ns/10ps      -timescale=1ns/1ps file.sv
tokens
```

Pre-processes the OpenVera files to generate two files, `tokens.vr` and `tokens.vrp`. The `tokens.vr` contains the preprocessed result of the non-encrypted OpenVera files, while the `tokens.vrp` contains the pre-processed result of the encrypted OpenVera files. If there is no encrypted OpenVera file, VCS sends all the OpenVera pre-processed results to the `tokens.vr` file.

`use_sigprop`

Enables the signal property access functions. For example, `vera_get_ifc_name()`.

`vera_portname`

Specifies the following:

The Vera shell module name is named `vera_shell`.

The interface ports are named `ifc_signal`.

Bind signals are named, for example, as: `\ifc_signal[3:0]`.

`-ntb_shell_only`

Generates only a `.vshell` file. Use this option when compiling a testbench separately from the design file.

`-ntb_sfname filename`

Specifies the file name of the testbench shell.

`-ntb_sname module_name`

Specifies the name and directory where VCS writes the testbench shell module.

`-ntb_spath`

Specifies the directory where VCS writes the testbench shell and shared object files. The default is the compilation directory.

`-ntb_vipext .ext`

Specifies an OpenVera encrypted-mode file extension to mark files for processing in OpenVera encrypted IP mode. Unlike the `-ntb_fileext` option, the default encrypted-mode extensions `.vrp` and `.vrhp` are not overridden and will always be in effect. You can pass multiple file extensions at the same time using the plus (+) character.

Options for Different Versions of Verilog

`-v95`

Specifies not recognizing Verilog 2001 keywords.

`+systemverilogext+ext`

Specifies a file name extension for SystemVerilog source files. If you use a different file name extension for the SystemVerilog part of your source code and you use this option, the `-sverilog` option has to be omitted.

This compile-time option also works similar to the `-sverilog` option in which it enables SystemVerilog LRM (IEEE Std 1800-2012) rules for all the source files on the vcs command line and not only the files with the specified extension.

`+verilog2001ext+ext`

Specifies a file name extension for Verilog 2001 source files.

`+verilog1995ext+ext`

Specifies a file name extension for Verilog 1995 files. Using this option allows you to write Verilog 1995 code that would be invalid in Verilog 2001 or SystemVerilog code, such as using Verilog 2001 or SystemVerilog keywords, like `localparam` and `logic`, as names.

Note:

Do not enter all three of these options on the same command line.

`-extinclude`

If a source file for one version of Verilog contains the `'include` compiler directive, VCS by default compiles the included file for the same version of Verilog, even if the included file has a different filename extension. If you want VCS to compile the included file with the version specified by its extension, enter this compile-time option. The following code examples show using this option.

If source file `a.v` contains the following:

```
`include "b.sv"
module a();
reg ar;
endmodule
```

and if source file `b.sv` contains the following:

```
module b();
  logic ar;
endmodule
```

VCS compiles `b.sv` for SystemVerilog with the following command line:

```
vcs a.v +systemverilogext.sv -extinclude
```

Option for Initializing Verilog Variables, Registers and Memories with Random Values

```
+vcs+initreg+random
```

Initializes all bits of the Verilog variables, registers defined in sequential UDPs, and memories including multi-dimensional arrays (MDAs) in your design to random value 0 or 1, at time zero. The default random seed is used.

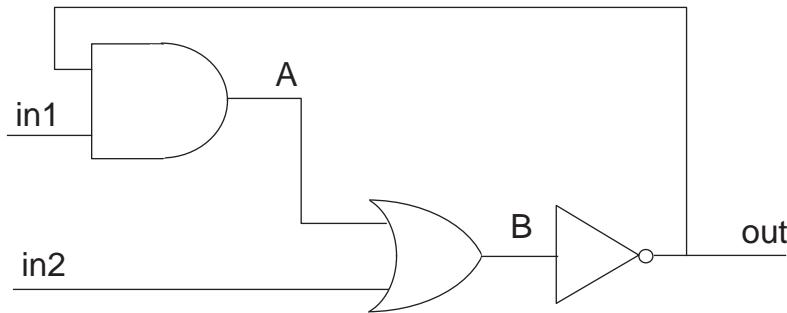
The supported data types are:

- reg
- bit
- integer
- int
- logic

For Example:

```
% vcs +vcs+initreg+random [other_vcs_options] file1.v file2.v file3.v
```

The initialization option may expose an infinite simulation loop at time zero in combinational logic with a feedback loop, as shown in [Figure 205](#).

Figure 204 Combinational Logic With a Feedback Loop

In Figure 205, the `reg` variables `in1`, `in2`, `A` and `B` have the default initial values `x`. Assigning value 0 or 1 to `in1` or `in2` does not alter the value of `A`, `B` and `out`. The feedback loop is stabilized and the simulation advances. Some combinations of initial values assigned to these `reg` variables trigger a continuous re-evaluation of the combinational logic which results in an infinite simulation loop.

When the initialization option is used, the initialized values of variables may conflict with the initial variable assignments specified in a design.

The following are steps to prevent potential race conditions:

- Avoid assigning initial values to `reg` variables in the variable declarations when the assigned values are different from the values specified with the `+vcs+initreg+random` option.

For example:

```
reg [7:0] r1=8'b01010101;
```

- Avoid assigning values to registers or memory elements at simulation time 0 when the assigned values are different from the values specified with the `+vcs+initreg+random` option.

For example:

```
reg [7:0] mem [7:0][15:0];
initial
begin
  mem[1][1]=8'b00000001;
```

- Avoid initializing state variables to an unknown, `x`, state.
- Avoid inconsistent states in the design due to randomization.

The initialization option can potentially be used to reduce the amount of time spent on initialization related issues in gate-level simulations. At time 0, all uninitialized `reg`

variables are assigned the default value `x`, which is a non-deterministic and unknown state of a design. The value `x` can propagate during a gate-level simulation and cause unexpected behaviors. You can use the `+vcs+initreg+random` option to initialize all bits of Verilog variables, registers and memories to prevent propagation of `x` values in a gate-level simulation.

Note:

The initialization option is targeted for initializing variables in gate level simulations (including UDP variables). Initialization of variables in RTL constructs such as named blocks, structures, or in user-defined tasks or functions is not supported.

The `+vcs+initreg+random` option only applies to the Verilog portion of a mixed language design.

The `+vcs+initreg+0` and `+vcs+initreg+1` compile-time options are no longer supported. You must use the `+vcs+initreg+random` option at compile-time.

Option for Initializing Verilog Variables, Registers and Memories in Selective Parts of a Design

`+vcs+initreg+config+config_file`

Specifies a configuration file for initializing Verilog variables, registers defined in sequential UDPs, and memories including multi-dimensional arrays (MDAs) in your design, at time zero. In the configuration file, you can define the parts of a design to apply the initialization and the initialization values of the variables.

Following is the syntax of the configuration file entries:

```
defaultvalue x|z|0|1|random <seed_value>
instance instance_hierarchical_name x|z|0|1|random <seed_value>
tree instance_hierarchical_name depth x|z|0|1|random <seed_value>
module module_name x|z|0|1|random <seed_value>
modtree module_name depth x|z|0|1|random <seed_value>
```

Note:

VCS supports initializing parts of the design with `z|0|1` values in the initreg configuration file. Value `x` specifies the items to be excluded from initialization. Value `random` specifies initialization with random values of `0` and `1`.

The `defaultvalue` entry

```
defaultvalue x|z|0|1|random <seed_value>
```

A `defaultvalue` entry starts with the keyword `defaultvalue` and should be the first entry in a configuration file. Only one `defaultvalue` entry is allowed in a configuration file. This entry specifies the default values (`z|0|1`) for all Verilog variables, registers and memories in the design.

`defaultvalue x` excludes the design from initialization. The design can be initialized with subsequent configuration file entries.

`defaultvalue random` specifies design initialization with random values of `0` and `1`. With `random <seed_value>`, you can also specify a seed value for the VCS random value generator.

In the absence of the `defaultvalue` entry, initialization is limited to the scopes and hierarchies specified in the configuration file.

The `instance` entry

```
instance instance_hierarchical_name x|z|0|1|random <seed_value>
```

An `instance` entry starts with the keyword `instance`. This entry specifies a module instance using hierarchical name and the initial values (`z|0|1`) for the Verilog variables, registers and memories in this instance. Value `x` excludes the specified instance from initialization.

The `tree` entry

```
tree instance_hierarchical_name depth x|z|0|1|random <seed_value>
```

A `tree` entry starts with the keyword `tree`. This entry specifies a sub-hierarchy and the initial values (`z|0|1`) for the Verilog variables, registers and memories in this sub-hierarchy. Value `x` excludes the specified tree from initialization.

When a hierarchical name of a module instance is specified, the initialization applies to the specified instance and the module instances that are hierarchically beneath the specified instance. You can specify a depth value to limit the levels down the hierarchy for applying the initialization.

Depth Value	Level of Initialization
0	Initialize all levels down the sub-hierarchy until the leaf level instances.
1	Initialize only the specified instance
2 and up	Initialize the specified number of levels down the sub-hierarchy from the specified instance

0	Initialize all levels down the sub-hierarchy until the leaf level instances.
1	Initialize only the specified instance
2 and up	Initialize the specified number of levels down the sub-hierarchy from the specified instance

The `module` entry

```
module module_name x|z|0|1|random <seed_value>
```

A **module** entry starts with the keyword **module**. This entry specifies initial values (**z|0|1**) for all instances of the specified module. Value **x** excludes all instances of the specified module from initialization.

The **modtree** entry

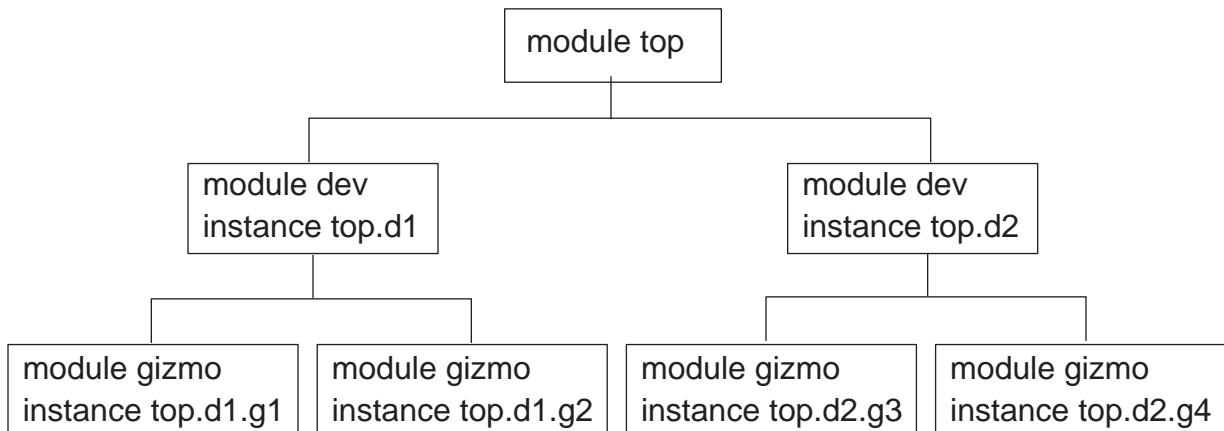
```
modtree module_name depth x|z|0|1|random <seed_value>
```

A **modtree** entry starts with the keyword **modtree**. This entry specifies initial values (**z|0|1**) for all instances of the specified module and all instances that are hierarchically beneath those instances. Value **x** excludes the specified modtree from initialization.

Configuration File Example

[Figure 205](#) is a hierarchical diagram of a small design.

Figure 205 Design Hierarchy for Initializing from a Configuration File



The following are the example entries in a configuration file for the small design shown in [Figure 205](#).

```
instance top.d1 0
```

Initializes variables, registers, and memories in the `top.d1` instance to value 0.

```
tree top 0 0
tree top.d1 0 x
```

The first entry initializes all variables, registers, and memories in the design to value 0. The second entry excludes the variables, registers, and memories in the `top.d1` instance and all instances under `top.d1`, namely `top.d1.g1` and `top.d1.g2`, from the `initreg` feature.

```
module gizmo 1
```

Initializes variables, registers, and memories in all instances of the `gizmo` module to value 1, namely `top.d1.g1`, `top.d1.g2`, `top.d2.g3`, and `top.d2.g4`.

```
modtree dev 0 random
```

Initializes variables, registers, and memories in both instances of the `dev` module and all four instances beneath these instances with random values of 0 or 1. The `top` module is not initialized.

```
modtree dev 0 random
instance top.d1.g2 x
```

The first entry is described in the previous example. The second entry excludes the variables, registers, and memories in the `top.d1.g2` instance from initialization.

```
modtree dev 0 random 186
```

Initializes variables, registers, and memories in both instances of the `dev` module and all four instances beneath these instances with random values (0 or 1) set with seed 186. For different randomizations of variables, you can change the seed to a different value (for example, 999).

Options for Selecting Register or Memory Initialization

```
+vcs+initreg+random+nomem
```

Disables initialization of memories or multi-dimensional arrays (MDAs). This option allows initialization of variables that do not have a dimension. This option can only be used when the `+vcs+initreg+random` or `+vcs+initreg+config+config_file` option is specified at compile-time.

```
+vcs+initreg+random+noreg
```

Disables initialization of variables that do not have a dimension. This option allows initialization of memories or MDAs. This option can only be used When the `+vcs+initreg+random` or `+vcs+initreg+config+config_file` option is specified at compile-time.

Options for Using Radiant Technology

```
+rad
```

Performs Radiant Technology optimizations on your design.

These optimizations are also enabled for SystemVerilog part of the design.

```
+optconfigfile+filename
```

Specifies a configuration file that lists the parts of your design you want to optimize (or not optimize) and the level of optimization for these parts. You can also use the configuration file to specify ACC write capabilities. See [Compiling With Radiant Technology](#).

Options for Starting Simulation Right After Compilation

-R

Runs the executable file immediately after VCS links it together.

Note:

You cannot use this option with all compile options. This option is not recommended when extra options are expected at runtime.

Options for Specifying Delays and SDF Files

`-sdf min|typ|max:instance_name:file.sdf`

Enables SDF annotation. Minimum, typical, or maximum values specified in `file.sdf` are annotated on the instance, `instance_name`.

`+allmtm`

Specifies compiling separate files for minimum, typical, and maximum delays when there are `min:typ:max` delay triplets in SDF files. If you use this option, you can use the `+mindelays`, `+typdelays`, or `+maxdelays` options at runtime to specify which compiled SDF file VCS uses. Do not use this option with the `+maxdelays`, `+mindelays`, or `+typdelays` compile-time options.

`+charge_decay`

Enables charge decay in `trireq` nets. Charge decay does not work if you connect the `trireq` to a transistor (bi-directional pass) switch such as `tran`, `rtran`, `tranif1`, or `rtranif0`.

`+delay_mode_path`

Uses only delay specifications in module-path delays in `specify` blocks. Overrides all the delay specifications on all gates, switches, and continuous assignments to zero.

`+delay_mode_zero`

Removes delay specifications on all gates, switches, continuous assignments, and module paths.

`+delay_mode_unit`

Overrides all the delay specifications in module-path delays in specify blocks to zero delays. Overrides all the delay specifications on all gates, switches, and continuous assignments to the shortest time precision argument of all the '`'timescale`' compiler directives in the source code. The default time unit and time precision argument of the '`'timescale`' compiler directive is 1s.

`+delay_mode_distributed`

Overrides all the delay specifications in module-path delays in specify blocks to zero delays. Uses only the delay specifications on all gates, switches, and continuous assignments.

`-add_seq_delay`

Use the `-add_seq_delay` option to set the delays of the sequential User-Defined Primitives (UDPs) without delays. Its syntax is as follows:

`-add_seq_delay <n>`

Where, `n` is the delay specification argument. It can be a real number or a real number followed by a time unit. The time unit can be `fs`, `ps`, `ns`, `us`, `ms`, or `s`. If no time unit is specified, then the simulation `time_unit` is used. For example, if simulation `time_unit/time_precision` is `1ns/1ps`, then `-add_seq_delay 3` means `3ns`.

The delay specification argument is applied to all sequential UDPs (without delays) in the design.

Examples

For example, consider that the simulation `time_unit/time_precision` is `1ns/1ps`.

- The following option assigns a `1ns` delay to all sequential UDP paths.

`-add_seq_delay 1ns`

- The following option assigns a `0.7ns` delay to all sequential UDP paths.

`-add_seq_delay 0.7`

`-add_seq_delay 0.7ns`

`-add_seq_delay 700ps`

Key Points to Note

- If sequential UDPs already have delay specified (`#(delay)`, including `#0`), then `-add_seq_delay` is ignored. That is, `-add_seq_delay` only supports sequential UDPs without delays.

- The `-add_seq_delay` option does not affect IOPATH delay such as:

```
specify
  (posedge ck => q +: d) = (10,11);
endspecify
```

The above IOPATH delay (10,11) remains the same even when `-add_seq_delay <n>` is specified.

- If you use `+delay_mode_zero` and `-add_seq_delay` on the same command line, then the UDP is considered, as mentioned below:

With `+delay_mode_zero`: The `+delay_mode_zero` option takes precedence.

Without `+delay_mode_zero`: The `-add_seq_delay` option takes precedence if IOPATH delay is smaller than it.

- The `-add_seq_delay` option overrides `+delay_mode_unit`. For example, if you specify `-add_seq_delay 15ps +delay_mode_unit`, then still you see 15ps delay.
- The `-add_seq_delay` option overrides `+delay_mode_distributed`. For example, if you specify `-add_seq_delay 15ps +delay_mode_distributed`, then still you see 15ps delay.

`+maxdelays`

Specifies using the maximum timing delays in the min:typ:max delay triplets when compiling the SDF file. The `mtm_spec` argument to the `$sdf_annotate` system task overrides this option.

Note:

If you intend to include the `+maxdelays` option during elaboration, then it is mandatory to include this option during analysis as well. The syntax is as follows:

Analysis

```
% vlogan [vlogan_options] file2.v file3.v +maxdelays
```

Elaboration

```
% vcs -sdf min|typ|max:instance_name:file.sdf \
[elab_options] top_cfg/entity/module +maxdelays
```

Simulation

```
% simv [run_options]
+mindelays
```

Specifies using the minimum timing delays in the min:typ:max delay triplets when compiling the SDF file. The `mtm_spec` argument to the `$sdf_annotate` system task overrides this option.

If you intend to include the `+mindelays` option during elaboration, then it is mandatory to include this option during analysis as well. The syntax is as follows:

Analysis

```
% vlogan [vlogan_options] file2.v file3.v +mindelays
```

Elaboration

```
% vcs -sdf min|typ|max:instance_name:file.sdf \
[elab_options] top_cfg/entity/module mindelays
```

Simulation

```
% simv [run_options]
+typdelays
```

Specifies using the typical timing delays in min:typ:max delay triplets when compiling the SDF file. The `mtm_spec` argument to the `$sdf_annotate` system task overrides this option.

Note:

If you intend to include the `+typdelays` option during elaboration, then it is mandatory to include this option during analysis as well. The syntax is as follows:

```
% vlogan [vlogan_options] file2.v file3.v +typdelays
```

Elaboration

```
% vcs -sdf min|typ|max:instance_name:file.sdf \
[elab_options] top_cfg/entity/module +typdelays
```

Simulation

```
% simv [run_options]
```

Analysis

```
+multisource_int_delays
```

Enables the multisource INTERCONNECT feature, including transport delays with full pulse control.

```
+nbaopt
```

Removes all intra-assignment delays in all the non-blocking assignment statements in the design. Many users enter a #1 intra-assignment delay in non-blocking procedural assignment statements to make debugging in the Wave window easier. For example:

```
reg1 <= #1 reg2;
```

These delays impede the simulation performance of the design, so after debugging, you can remove these delays with this option.

Note:

The +nbaopt option removes all intra-assignment delays in all the non-blocking assignment statements in the design, not just the #1 delays.

```
+sdf_nocheck_celltype
```

For a module instance to which an SDF file back-annotates delay data, disables comparing the module identifier in the source code with the CELLTYPE entry in the SDF file.

```
+transport_int_delays
```

Enables transport delays for delays on nets with a delay back-annotated from an INTERCONNECT entry in an SDF file. The default is inertial delays.

```
+transport_path_delays
```

Enables transport delays for module-path delays.

```
-sdfretain
```

Enables timing annotation as specified by a RETAIN entry on IOPATH delays. By default, VCS ignores RETAIN entries with the following warning message:

```
Warning-[SDFCOM_RCI] RETAIN clause ignored
SDF_filename, line_number
module: module_name, "instance: hierarchical_name"
      SDF Warning: RETAIN clause ignored, but IOPATH
      annotated,
      Please use -sdfretain switch to consider RETAIN
```

The syntax for RETAIN entries is as follows:

```
(IOPATH port_spec port_instance (RETAIN delval_list)* delval_list)
```

For example:

```
(IOPATH RCLK DOUT[0] (RETAIN (40)) (100.1) (100.2))
```

```
-sdfretain=warning
```

If the RETAIN entry values are larger than the delay values, VCS displays the following warning message at runtime:

```
Warning-[SDFRT_IRV] RETAIN value ignored
    RETAIN value is ignored as it is greater than IOPATH
    delay
```

If you want to see a warning message at compile time, enter this option along with the `-sdfretain` option. The following is an example of this warning message:

```
Warning-[SDFCOM_RLTPD] RETAIN value larger than IOPATH delay
SDF_filename, line_number
module: module_name, "instance: hierarchical_name"
SDF Warning: RETAIN value (value) is larger than IOPATH delay, RETAIN
will be ignored at runtime

+ioopath+edge+sub-option
```

This option is used when edge sensitivity is used in IOPATH SDF file entries. The different sub-options used with `+ioopath+edge+option` and their descriptions are as follows:

`+ioopath+edge+strict`

This option is used for LRM compliance. When edge sensitivity is specified for the input port in the SDF file and corresponding arc is not found in Verilog model, VCS by default does not give the warning message, you should use the `+ioopath+edge+strict` switch to display the warning message. After the warning message is displayed, the data from SDF is not back-annotated to the Verilog model.

`+ioopath+edge+ignore`

This option can be used to make the annotation work by ignoring the edge in SDF.

`+ioopath+edge+max`

This option is used for annotating higher delays.

`+ioopath+edge+min`

This option is used for annotating smaller delays.

`+mp64`

This option is used for annotating only SDF IOPATH delays up to 64 bits. It supports both native delays in SDF and scaled delays due to a timescale difference.

Example

```
vcs test.v +mp64
```

Limitations

This option is not supported in the following scenarios:

- Specparam in specify blocks
- Retain delay
- Large delay, which is limited by IEEE754 double precision (52-bits fraction)

Options for Compiling an SDF File

`+csdf+precompile`

Precompiles your SDF file into a format that VCS can parse when it compiles your Verilog code. See [Precompiling an SDF File](#).

`-diag=sdf:icverbose`

Enables the `sdfAnnotateInfo` file to display the total number of INTERCONNECT statement in an SDF file and the number of successfully annotated INTERCONNECT statement in an SDF file.

Options for Specify Blocks and Timing Checks

`+pathpulse`

Enables the search for `PATHPULSE$ specparam` in specify blocks.

`+nospecify`

Suppresses module-path delays and timing checks in specify blocks. This option can significantly improve simulation performance.

`+notimingcheck`

Tells VCS to ignore timing check system tasks when it compiles your design. This option can moderately improve simulation performance. The extent of this improvement depends on the number of timing checks that VCS ignores. You can also use this option at runtime to disable these timing checks after VCS has compiled them into the executable. However, the executable simulates faster if you include this option at compile time so that the timing checks are not in the executable. If you need the delayed versions of the signals in negative timing checks but want faster performance, include this option at runtime. The delayed versions are not available if you use this option at compile time.

Note:

- VCS recognizes `+notimingchecks` to be the same as `+notimingcheck` when you enter it on the vcs or simv command line.
- The `+notimingcheck` option has higher precedence than any `tcheck` command in UCLI.

`+no_notifier`

Disables toggling of the notifier register that you specify in some timing check system tasks. This option does not disable the display of warning messages when VCS finds a timing violation that you specified in a timing check.

`+no_tchk_msg`

Disables display of timing violations, but does not disable the toggling of notifier registers in timing checks. This is also a runtime option.

Options for Pulse Filtering

`+pulse_e/number`

Displays an error message and propagates an x value for any path pulse whose width is less than or equal to the percentage of the module-path delay specified by the `number` argument, but is still greater than the percentage of the module-path delay specified by the `number` argument to the `+pulse_r/number` option.

`+pulse_r/number`

Rejects any pulse whose width is less than `number` percent of the module-path delay. The `number` argument is in the range of 0 to 100.

`+pulse_int_r`

Same as the existing `+pulse_r` option, except it applies only to INTERCONNECT delays.

`+pulse_int_e`

Same as the existing `+pulse_e` option, except it applies only to INTERCONNECT delays.

`+pulse_on_event`

Specifies that when VCS encounters a pulse shorter than the module-path delay, VCS waits until the module-path delay elapses and then drives an x value on the module output port and displays an error message. It drives that x value for a simulation time equal to the length of the short pulse or until another simulation event drives a value on the output port.

`+pulse_on_detect`

Specifies that when VCS encounters a pulse shorter than the module-path delay, VCS immediately drives an x value on the module output port, and displays an error message. It does not wait until the module-path delay elapses. It drives that x value until the short pulse propagates through the module or until another simulation event drives a value on the output port.

Options for Negative Timing Checks

`-negdelay`

Enables the use of negative values in IOPATH and INTERCONNECT entries in SDF files.

To consider a negative INTERCONNECT delay, one of the following should be true:

- Sum of INTERCONNECT and PORT delays should be greater than zero
- Sum of INTERCONNECT and IOPATH delays should be greater than zero
- Sum of INTERCONNECT and DEVICE delays should be greater than zero

Otherwise, the negative INTERCONNECT delay is ignored, and a warning message is generated for the same.

Similarly, to consider a negative IOPATH delay, the sum of IOPATH and DEVICE delays should be greater than zero. Otherwise, the negative IOPATH delay is ignored, and a warning message is generated for the same.

Limitations

This option is not supported in the following scenarios:

- RETAIN on negative IOPATH
- INCREMENT delay

`+neg_tchk`

Enables negative values in timing checks.

`+old_ntc`

Prevents the other timing checks from using delayed versions of the signals in the `$setuphold` and `$recrwm` timing checks.

`+NTC2`

In `$setuphold` and `$recrwm` timing checks, specifies checking the timestamp and timecheck conditions when the original data and reference signals change value instead of when their delayed versions change value.

Options for Profiling Your Design

`-simprofile time | mem`

Specifies the type of simulation profiling you want done, see [The Unified Simulation Profiler](#).

Options to Specify Source Files and Compilation/Elaboration Options in a File

`-f filename`

Specify a file that contains a list of source files and compile-time options, including C source files and object files.

The following are the features of `-f` option:

- You can use Verilog comment characters // and /* */ to comment out entries in the file. The comment character // used in between file path is treated as a file path only and not considered as a comment.

For example, consider `/abc/def//xyz.v`. The comment character // used here is not treated as a comment.

- You can use this option inside the `file` to point to another file.
- You can specify all compile-time options that begin with a plus (+) character. However, you can only specify the following compile-time options that begin with a minus (-) character:

<code>-f</code>	<code>-y</code>	<code>-l</code>	<code>-u</code>	<code>-v</code>	<code>-sverilog</code>
-----------------	-----------------	-----------------	-----------------	-----------------	------------------------

The `-f` option is not supported in the UUM flow.

`-file filename`

Specify a file that contains a list of source files and VCS compilation/elaboration options, including C source files and object files.

You can use this option to overcome the limitation of `-f` option related to compile-time options that begin with (-) character.

The `-file` option is supported in the UUM flow.

`-F filename`

This option is similar to the `-f` option, but you can also specify a file list and a path to search for the files. Following is the syntax:

```
%vcs top.v -F <path_to_file>/filelist
```

When you specify this option, the path to the file list gets added as a prefix to the content of the file list.

Consider that the `<filelist>` consisting of files `a.v` and `b.v` exists in the previous directory of the current working directory. With the following syntax, the path to the `<filelist>` is added as a prefix to the content of the `<filelist>`. When parsed, VCS searches for files `../a.v` and `../b.v`.

```
%vcs top.v <source_files> -F ../<filelist>
```

You can also specify an absolute path name using the following syntax:

```
%vcs top.v -F <absolute_path>/filelist
```

The syntax allows you to search for files `<absolute_path>/a.v` and `<absolute_path>/b.v`.

The following are the features of `-F` option:

- You can use Verilog comment characters such as `//` and `/* */` to comment out entries in the file.
- You can use this option inside the `file` to point to another file.
- You can specify all compile-time options that begin with a plus (+) character. However, you can only specify the following compile-time options that begin with a minus (-) character:

<code>-CC</code>	<code>-f</code>	<code>-F</code>	<code>-gen_asm</code>	<code>-gen_obj</code>	<code>-l</code>	<code>-line</code>
<code>-P</code>	<code>-u</code>	<code>-v</code>	<code>-y</code>			

The `-F` option is not supported in the UUM flow.

Limitations of `-f`, `-file` and `-F` options

- These options do not support the `-full64` option in the file. You must enter that option on the vcs command-line.
- You cannot specify escape characters in the file.
- You cannot use meta characters in the file, except `*` and `$`.
- At UUM flow, `-f` option is allowed only with `vlogan/vhdlan` command line. With `vcs` elaboration command, `-file` option should be used instead of the `-f` option.

Options for Compiling Runtime Options Into the Executable

`+plusarg_save`

Some runtime options must be preceded by the `+plusarg_save` option for VCS to compile them into the executable.

`+plusarg_ignore`

Tells VCS not to compile the following runtime options into the simv executable. This option is used to counter the `+plusarg_save` option on a previous line.

Options for PLI Applications

`+applylearn+filename`

Recompiles your design to enable only the ACC capabilities that you needed for the debugging operations you did during a previous simulation of the design.

`-e new_name_for_main`

Specifies the name of your `main()` routine. You write your own `main()` routine when you are writing a C++ application or when your application does some processing before starting the simv executable.

Note:

Do not use the `-e` option with the VCS/SystemC Cosimulation Interface.

`-P pli.tab`

Compiles a user-defined PLI definition table file.

`+vpi`

Enables the use of VPI PLI access routines.

`+vpi+1`

Allows you to reduce the runtime memory by reducing the information storage for VPI interface at runtime. This option limits the behavioral information at compile-time, but preserves the structural information.

This option allows you to do the following:

- Browse the design hierarchy and read the values of variables. This facilitates debugging.
- Write over or force values on variables using `vpi_put_value()`. This allows a foreign language testbench to drive a stimulus to a Verilog design.
- Register VPI callbacks. This facilitates the waveform dumping features. However, certain advance debugging features (such as Line stepping, Driver/Loads information, and so on) will not be available.

Limitations

- You cannot use this option to browse, enable, or disable SV and RT assertions.

Note:

The `+vpi+1+assertion` option allows you to browse, enable, and disable SV and RT assertions to the base features of `+vpi+1`.

- When `+vpi+1` is used with debug capabilities, the following capabilities are disabled:
 - Line breakpoints are disabled inside initial, always, task, and function blocks.
 - Statement, thread, and frame callbacks.
 - VPI handles to statements.

`+vpi+1+assertion`

Allows you to browse, enable, and disable SV and RT assertions to the base features of `+vpi+1`.

`-load shared_library:registration_routine`

Specifies the registration routine in a shared library for a VPI application.

Options to Enable the VCS DirectC Interface

`+vc+[abstract+allhdrs+list]`

The `+vc` option enables extern declarations of C/C++ functions and calling these functions in your source code. See the *VCS DirectC Interface User Guide*. The optional suffixes to this option are as follows:

`+abstract`

Enables abstract access through `vc_handles`.

`+allhdrs`

Writes the `vc_hdrs.h` file that contains external function declarations that you can use in your Verilog code.

`+list`

Displays all the C/C++ functions that you called in your Verilog source code.

Options for Flushing Certain Output Text File Buffers

When VCS creates a log, VCD, or text file specified with the `$fopen` system function, VCS writes the data for the file in a buffer and periodically dumps the data from the buffer to the file on disk. The frequency of these dumps varies depending on many factors including the amount of data that VCS has to write to the buffer as simulation or compilation progresses. If you need to see or use the latest information in these files more frequently than the rate at which VCS normally flushes this data, these options tell VCS to flush the data more often during compilation or simulation.

`+vcs+flush+log`

Increases the frequency of flushing both the compilation and simulation log file buffers.

`+vcs+flush+dump`

Increases the frequency of flushing all VCD file buffers.

`+vcs+flush+fopen`

Increases the frequency of flushing all the buffers for the files opened by the `$fopen` system function.

`+vcs+flush+all`

Shortcut option for entering all three of the `+vcs+flush+log`, `+vcs+flush+dump`, and `+vcs+flush+fopen` options.

These options do not increase the frequency of dumping other text files, including the VCDE files specified by the `$dumpports` system task.

These options can also be entered at runtime. Entering them at compile-time modifies the `simv` executable so that it runs as if these options were always entered at runtime.

Options for Simulating SWIFT VMC Models and SmartModels

`-lmc-swift`

Includes the LMC SWIFT interface.

`-lmc-swift-template`

Generates a Verilog template for a SWIFT Model.

Options for Controlling Messages

`-error`

Revises the `+lint` and `+warn` options, to control error and warning messages. With them you can:

- Disable the display of any lint, warning, or error messages
- Disable the display of specific messages
- Limit the display of specific messages to a maximum number that you specify

For more details, See [Error/Warning/Lint Message Control](#).

Note:

The `-error` option is also a runtime option. However, only the following feature is supported at runtime:

`-error=[no]message_ID[:max_number],...`

`-nc`

Suppresses the Synopsys copyright message.

`-suppress`

Disables the display of error and warning messages. For details, see [Error/Warning/Lint Message Control](#).

`+sdfverbose`

By default, VCS displays no more than ten warning and ten error messages about back-annotating delay information from SDF files. This option enables the display of all back-annotation warning and error messages.

This default limitation on back-annotation messages applies only to messages displayed on the screen and written in the simulation log file. If you specify an SDF log file in the `$sdf_annotation` system task, this log file receives all messages.

`+libverbose`

Tells VCS to display a message when it finds a module definition in a source file in a Verilog library directory that resolves a module instantiation statement that VCS read in your source files, a library file, or in another file in a library directory. The message is as follows:

`Resolving module "module_identifier"`

VCS does not display this message when it finds a module definition in a Verilog library file that resolves a module instantiation statement.

`+lint=[no] ID|none|all`

Enables messages that tell you when your Verilog code contains something that is bad style, but is often used in designs.

Here:

`no`

Specifies disabling lint messages that have the ID that follows. There is no space between the keyword `no` and the ID.

`none`

Specifies disabling all lint messages. IDs that follow in a comma separated list are exceptions.

`all`

Specifies enabling all lint messages. IDs that follow preceded by the keyword `no` in a comma separated list are exceptions.

The following examples show how to use this option:

- Enable all lint messages except the message with the GCWM ID: `+lint=all,noGCWM`
- Enable the lint message with the NCEID ID: `+lint=NCEID`
- Enable the lint messages with the GCWM and NCEID IDs: `+lint=GCWM,NCEID`
- Disable all lint messages. This is the default. `+lint=none`

The syntax of the `+lint` option is very similar to the syntax of the `+warn` option for enabling or disabling warning messages. Additionally, these options have in common that some of their messages have the same ID. This is because when there is a condition in your code that causes VCS to display both a warning and a lint message, the corresponding lint message contains more information than the warning message and can be considered more verbose.

The number of possible lint messages is not large. They are as follows:

`Lint-[IRIMW] Illegal range in memory word`

`Lint-[NCEID} Non-constant expression in delay`

`Lint-[GCWM] Gate connection width mismatch`

`Lint-[CAWM] Continuous Assignment width mismatch`

```

    Lint-[IGSFPG] Illegal gate strength for pull gate
    Lint-[TFIPC] Too few instance port connections
    Lint-[IPDP] Identifier previously declared as port
    Lint-[PCWM] Port connect width mismatch
    Lint-[VCDE] Verilog compiler directive encountered

```

`-no_error ID+ID`

Changes the error messages with the UPIMI and IOPCWM IDs to warning messages with the `-no_error` compile-time option. You include one or both IDs as arguments, for example:

`-no_error UPIMI+IOPCWM`

This option does not work with the ID for any other error message.

`-q`

Quiet mode; suppresses messages such as those about the C compiler VCS is using, the source files VCS is parsing, the top-level modules, or the specified timescale.

`-v`

Verbose mode; compiles verbosely. The compiler driver program prints the commands it executes as it runs the C compiler, assembler, and linker. If you include the `-R` option with the `-v` option, the `-v` option is also passed to runtime executable, just as if you had entered `simv -v`.

`-vt`

Verbose mode; provides CPU time information. Like `-v`, but also prints the amount of time used by each command. Use of the `-vt` option can cause the simulation to slow down.

`+warn=[no] ID|none|all`

Uses warning message IDs to enable or disable display of warning messages. In the following warning message:

`Warning-[TFIPC] Too few instance port connections`

The text string `TFIPC` is the message ID. The syntax of this option is as follows:

`+warn=[no] ID|none|all,...`

Where:

no	Specifies disabling warning messages with the ID that follows. There is no space between the keyword <code>no</code> and the ID.
----	----------------------------------------------------------------------------------------------------------------------------------

none	Specifies disabling all warning messages. IDs that follow, in a comma-separated list, specify exceptions. VCS treats all SDF error messages as warning messages so including <code>+warn=none</code> disables SDF error messages.
all	Specifies enabling all warning messages, IDs that follow preceded by the keyword <code>no</code> , in a comma separated list, specify exceptions.

The following are examples that show how to use this option:

<code>+warn=noIPDW</code>	Enables all warning messages except the warning with the IPDW ID.
<code>+warn=none,TFIPC</code>	Disables all warning messages except the warning with the TFIPC ID.
<code>+warn=noIPDW,noTFIPC</code>	Disables the warning messages with the IPDW and TFIPC IDs.
<code>+warn=all</code>	Enables all warning messages. This is the default.

In cases where both `-error` and `+warn` for the same ID are used on the command line in order to downgrade the error to warning and at the same time suppress warning, the order in which they are specified on the command line also impact the compilation. VCS processes the options based on the order they are specified in the command line. If the `+warn=no<ID>` option follows the `-error=no<ID>` option, then the `+warn=no<ID>` option can take effect because the `-error=no<ID>` has already downgraded it to warning. Otherwise, the option has no use. So, if `-error=no<ID>` follows the `+warn=no<ID>`, you might not see the error and the warning is suppressed.

To suppress an error, always use `-suppress=ID`, which actually combines the functionality of `-error=no<ID>` and `+warn=no<ID>` together.

`+error+count`

Enables you to increase the VCS elaboration error count limit. By default, VCS stops elaboration after reaching 10 errors.

When you use the `+error+count` compile-time option, VCS stops elaboration once the number of errors reaches the count limit with the following note:

```
Note-[MAX_ERROR_COUNT] Maximum error count reached
Current number of errors has reached the default maximum error count
(12).
Use +error+<count> to increase the limit.
```

Note:

This option is available only for Vlogan and VCS commands. Also note that, when VCS encounters a fatal error (for example, syntax error), it exits immediately irrespective of the count that you have specified in the `+error +count` option.

Option to Run VCS in Syntax Checking Mode

VCS runs in multiple stages, such as parsing, and compilation/elaboration stage.

To make sure VCS quits normally before all syntax and semantic issues are checked, you need to enable the parsing stage using the option `-parse_only`.

```
% vcs -parse_only <other options>
```

This enables the VCS to run only during the parsing stage to check for syntax errors. Any syntax errors found is reported. Regardless of whether any syntax error is reported or not, VCS stops at the end of the parsing stage. The link or elaboration stages are not run. Hence, no link or elaboration errors are reported when the `-parse_only` option is used. Also, `simv` executable is not generated.

For example, consider the following the testcase `test.v`:

```
module top;

task one(input string fldName, input int bus = 0,input string fldName2);
    $display("In one");
endtask

task two(input int bus = 0,input string fldName);
    $display("In two");
endtask

initial
begin
    one(.bus(1));
    one(,1);
    two(.bus(1));
end
endmodule
```

If there are errors, VCS exits with the following error message:

```
Parsing design file 'test.v'

Error-[TFAFTC] Too few arguments to function/task call
error.v, 9
"one(.bus(1));"
The above function/task call is not done with sufficient arguments.
```

If there are no errors, then VCS exits normally as follows:

```
Parsing design file 'test.v'
Top Level Modules:
top
No TimeScale specified
```

```
Note-[PARSE-ONLY] VCS Parse-Only Mode
No syntax or semantic error detected after parsing the complete design,
VCS exits normally without generating executable.
```

Limitations

The option has the following limitations:

- When `-error=<Warn_ID>` option is used, VCS quits prior to the normal quit point because VCS upgrades the warning to error. Also, when `-error=<Warn_ID>` option is used, VCS may still complete compilation and quit normally even if the specified error exists, as the error is downgraded to warning.
-

Options for Cell Definition

The `-debug_access` and `-debug_region` options internally handle the options for cell definition. If you compile your design with the `-debug_access` and `-debug_region` options along with any of the options for cell definition, a warning message is issued and the options for cell definition are ignored.

Following are the options for cell definition:

`+nolibcell`

Does not define as a cell modules defined in libraries unless they are under the `'celldefine` compiler directive.

`+nocelldefinepli+0`

Enables recording in VPD files, the transition times and values of nets and registers in all modules defined under the `'celldefine` compiler directive or defined in a library that you specify with the `-v` or `-y` options. This option also enables full PLI access to these modules.

`+nocelldefinepli+1`

Disables recording in VPD files, the transition times and values of nets and registers in all modules defined under the `'celldefine` compiler directive. This option also disables full PLI access to these modules. Modules in a library file or directory are not affected by this option unless they are defined under the `'celldefine` compiler directive.

```
+nocelldefinepli+2
```

In VPD files, disables recording the transition times and values of nets and registers in all modules defined under the `'celldefine` compiler directive or defined in a library that you specify with the `-v` or `-y` options, whether the modules in these libraries are defined under the `'celldefine` compiler directive or not. This option also disables PLI access to these modules.

Disabling recording of transition times and values of the nets and registers in library cells can significantly increase simulation performance.

Note:

Disabling recording transitions in library cells is intended for batch simulation only and not for interactive debugging with GUI.

```
+nocelldefinepli+1+ports
```

Removes the PLI capabilities from `'celldefine` modules but allows PLI access to port nodes and parameters.

```
+nocelldefinepli+2+ports
```

Removes the PLI capabilities from library and `'celldefine` modules and allows PLI access to port nodes and parameters.

Options for Licensing

```
-licwait timeout
```

Enables VCS to retry for a license until `timeout` expires, where `timeout` is the time in minutes.

You can set this license option as follows:

```
% vcs -licwait 10 <other compile options>
```

Here, VCS waits for the license for 10 minutes.

```
-licqueue
```

Tells VCS to try for the license till it finds the license. If there are multiple jobs asking for a license, then any one of those jobs get the license (similar to the older or the deprecated option `+vcs+lic+wait`).

You can set this license option as follows:

```
% vcs -licqueue <other compile options>
```

```
VCS_LICENSE_WAIT
```

You must set the `VCS_LICENSE_WAIT` variable to 1 and use the `-licqueue` option, which enables the license wait. Thus, the first job to enter the queue gets the license when the license is available. This option must be used along with the `-licqueue` option.

`-ID`

Returns useful information about a number of things: the version of VCS that you have set the `VCS_HOME` environment variable to, the name of your work station, your workstation's platform, the host ID of your workstation (used in licensing), the version of the VCS compiler (same as VCS) and the VCS build date.

Options for Controlling the Linker

`-ld linker`

Specifies an alternate front-end linker. Only applicable in incremental compile mode, which is the default.

`-LDFLAGS options`

Passes options to the linker.

Examples: `-LDFLAGS "-z muldefs", -LDFLAGS "-rdynamic", -LDFLAGS "-lpthread", -LDFLAGS "-L <path>", -LDFLAGS "-l name", -LDFLAGS "-load"`

`-c`

Tells VCS to compile the source files, generate the intermediate C, assembly, or object files, and compile or assemble the C or assembly code, but not to link them. Use this option if you want to link by hand.

`-lname`

Links the `name` library to the resulting executable. Usage is the letter `l` followed by a name (no space between `l` and `name`). For example: `-lm` (instructs VCS to include the math library).

`-Marchive=number_of_module_definitions`

By default, VCS compiles module definitions into individual object files and sends all the object files in a command line to the linker. Some platforms use a fixed-length buffer for the command line, and if VCS sends too long a list of object files, this buffer overflows and the link fails. A solution to this problem is to have the linker create temporary object files containing more than one module definition so there are fewer object files on the linker command line. With this option, you enable creating these temporary object files and specify how many module definitions are in these files.

Using this option briefly doubles the amount of disk space used by the linker because the object files containing more than one module definition are copies of the object files

for each module definition. After the linker creates the `simv` executable, it deletes the temporary object files.

`-picarchive`

VCS can fail during linking due to the following two reasons:

- Huge size of object files: VCS compiles the units of your design into object files, then calls the linker to combine them together. Sometimes the size of a design is large enough that the size of text section of these object files exceeds the limit allowed by the linker. If so, the linker fails and generates the following error:

```
relocation truncated to fit:....
```

- Large number of object files: By default, VCS compiles module or entity definitions into individual object files and sends this list of object files in a single command line to the linker. Some platforms use a fixed-length buffer for the command line. If VCS sends a long list of object files, this buffer overflows and the link fails, generating errors such as:

```
make: execvp: gcc: Argument list too long
```

```
make: execvp: g++: Argument list too long
```

You can use the `-picarchive` option to deal with the above linker errors. The `-picarchive` option does the following:

1. Enables Position Independent Code (PIC) object file generation along with linking the shared object version of VCS libraries.
2. Archives generated PIC code into multiple shared objects inside `simv.daidir` or `simv.db.dir` directory.
3. Links the Shared objects at runtime to the final executable, instead of linking all the objects statically into final executable in a single step at compile-time.

Options for Controlling the C Compiler

`-cc compiler`

Specifies an alternate C compiler.

`-CC options`

Passes options to the C compiler or assembler.

`-CFLAGS options`

Passes options to C compiler. Multiple `-CFLAGS` are allowed. Allows passing of C compiler optimization levels. For example, if your C code, `test.c`, calls a library file in your

VCS installation under `$VCS_HOME/include`, use any of the following `-CFLAGS` option arguments:

```
%vcs top.v test.c -CFLAGS "-I$VCS_HOME/include"
```

or

```
%setenv CWD `pwd'  
%vcs top.v test.c -CFLAGS "-I$CWD/include"
```

or

```
%vcs top.v test.c -CFLAGS "-I../include"
```

The reason to enter `"../include"` is because VCS creates a default `csrc` directory where it runs gcc commands. The `csrc` directory is under your current working directory. Therefore, you need to specify the relative path of the `include` directory to the `csrc` directory for gcc C compiler. Further, you cannot edit files in the `csrc` because VCS automatically creates this directory.

`-cpp`

Specifies the C++ compiler.

Note:

If you are entering a C++ file or an object file compiled from a C++ file on the `vcs` command line, you must tell VCS to use the standard C++ library for linking. To do this, enter the `-lstdc++` linker flag with the `-LDFLAGS` elaboration option.

For example:

```
vcs top.v source.cpp -P my.tab \  
-cpp /net/local/bin/c++ -LDFLAGS -lstdc++
```

`-jnumber_of_processes`

Specifies the number of processes that VCS forks for parallel compilation. There is no space between the `"j"` character and the number. You can use this option when generating intermediate C files (`-gen_c`) and their parallel compilation.

`-C`

Stops after generating the C code intermediate files.

`-OO`

Suppresses optimization for faster compilation (but slower simulation). Suppresses optimization for how VCS both writes intermediate C code files and compiles these files. This option is the uppercase letter "O" followed by a zero with no space between them.

`-Onumber`

Specifies an optimization level for how VCS both writes and compiles intermediate C code files. The number can be in the 0-4 range; 2 is the default, 0 and 1 decrease optimization, 3 and 4 increase optimization. This option is the uppercase letter “O” followed by 0, 1, 2, 3 or 4 with no space between them. See above for additional information regarding the `-OO` variant.

`-override-cflags`

Tells VCS not to pass its default options to the C compiler. By default, VCS has a number of C compiler options that it passes to the C compiler. The options it passes depends on the platform, whether it is a 64-bit compilation, whether it is a mixed HDL design, and other factors. VCS passes these options and then passes the options you specify with the `-CFLAGS` compile-time option.

Options for Source Protection

For information about source protection options, see chapter [Encrypting Source Files](#).

Options for Mixed Analog/Digital Simulation

Following are the options for mixed analog/digital simulation:

`-ad=[initfile]`

Enables the mixed-signal feature. If `-ad` is used alone, the mixed-signal control file name is `vcsAD.init`, by default. If the file name is different, it must be given with the `=initFile` option.

The mixed-signal simulation control file contains all the commands to configure mixed-signal simulation.

`-ams_discipline discipline_name`

Specifies the default discrete discipline in Verilog AMS.

`-ams_iereport`

If information on auto-inserted connect modules (AICMs) is available, displays this information on the screen and in the log file.

`+bidir+1`

Tells VCS to finish compilation when it finds a bidirectional registered mixed-signal net.

`+print+bidir+warn`

Tells VCS to display a list of bidirectional, registered, mixed signal nets.

```
+verilogamsext+vams
```

To avoid keyword conflicts between Verilog-AMS and SystemVerilog, it is preferable that Verilog-AMS and SystemVerilog code each get parsed separately using their own language parsers. Create all Verilog-AMS and SystemVerilog files with distinct extensions.

For example, "`*.vams`" can be used for Verilog-AMS files and "`*.v`", "`*.sv`", or "`*.svh`" for SystemVerilog. The following VCS options can be used to identify these file extensions as the differentiation between SystemVerilog and Verilog-AMS contexts:

```
% vcs -ams -ad +verilogamsext+vams \ +systemverilogext+sv+v+svh ...
```

Unified Option to Change Generic and Parameter Values

```
-gfile cmdfile
```

Overrides the default values for design generics and parameters by using values from the file `cmdfile`. The `cmdfile` file contains assign commands targeting design generics and parameters.

The syntax for a line in the file is as follows:

```
assign value path_to_parameter/generic
```

The path to the parameter or generic is similar to a hierarchical name except that you use the forward slash character (/) instead of a period as the delimiter.

Options for Changing Parameter Values

```
-pvalue+parameter_hierarchical_name=value
```

Changes the specified parameter to the specified value.

```
-parameters filename
```

Changes the parameters specified in the file to values specified in the file. The syntax for a line in the file is as follows:

```
assign value path_to_parameter
```

The path to the parameter is similar to a hierarchical name, except that you use the forward slash character (/) instead of a period as the delimiter.

Checking for x and z Values in Conditional Expressions

`-xzcheck [nofalseneg]`

Checks all the conditional expressions in the design and displays a warning message every time VCS evaluates a conditional expression to have an x or z value.

`nofalseneg`

Suppress the warning message when the value of a conditional expression transitions to an x or z value and then to 0 or 1 in the same simulation time step.

Options for Detecting Race Conditions

`-race`

Specifies that VCS generate a report of all the race conditions in the design and write this report in the `race.out` file during simulation. For more information, see [The Dynamic Race Detection Tool](#).

The `-race` compilation/elaboration option supports dynamic race detection for both pure Verilog and SystemVerilog data types.

`-racecd`

Specifies that during simulation, VCS generate a report of the race conditions in the design between the `'race` and `'endrace` compiler directives and write this report in the `race.out` file. For more information, see [The Dynamic Race Detection Tool](#).

The `-racecd` compilation/elaboration option supports dynamic race detection for both pure Verilog and SystemVerilog data types.

`+race=all`

Analyzes the source code during compilation to look for coding styles that cause race conditions. For more information, see [The Static Race Detection Tool](#).

The `+race=all` option supports only pure Verilog constructs.

Options to Specify the Time Scale

You can use the following options to specify the time scale:

`-timescale=time_unit/time_precision`

Occasionally, some source files contain the `'timescale` compiler directive and others do not. In this case, if you specify the source files that do not contain the `'timescale`

compiler directive on the command line before you specify the ones that do, this is an error condition and VCS halts compilation, by default. This option enables you to specify the timescale for the source files that do not contain this compiler directive and precede the source files that do. Do not include spaces when specifying the arguments to this option.

`-unit_timescale[=<default_timescale>]`

The `-unit_timescale` option enables you to specify the default time unit for the compilation-unit scope. You must not include spaces when specifying arguments to this option.

The IEEE Standard 1800-2005 SystemVerilog LRM explains the time unit declaration, as follows:

"The time unit of the compilation-unit scope can only be set by a time unit declaration, not a `'timescale` directive. If it is not specified, then the default time unit shall be used."

Since the `-timescale` option does not affect the compilation-unit scope, you must use the `-unit_timescale` option to specify the default time unit for the compilation-unit scope.

The `default_timescale` value should be in the same format as the `'timescale` directive. If the default timescale is not specified, then `1s/1s` is taken as the default timescale of the compilation-unit.

`-override_timescale=time_unit/time_precision`

Overrides the time unit and precision unit for all the `'timescale` compiler directives in the source code, and, similar to the `-timescale` option, provides a timescale for all module definitions that precede the first `'timescale` compiler directive. Do not include spaces when specifying the arguments to this option.

`-time base_time`

Sets the time base for the simulation. This option overrides the default `TIMEBASE` variable value in the `synopsys_sim.setup` file. The default value for `base_time` is `ns`.

`-time_res value`

Sets the time resolution for the simulation. This option overrides the default `TIME_RESOLUTION` variable value in the `synopsys_sim.setup` file.

`-noinherit_timescale`

This is a VCS or vlogan option that allows you to specify a global timescale within the compilation unit of any source file that does not have an explicit `'timescale` directive.

If you specify the `-noinherit_timescale` option in the command line and there is no time unit/time precision specification present inside the module, program, interface, or package definition, then the time unit/time precision is determined using the following rules:

- If the module or interface definition is nested, then the time unit/time precision shall be inherited from the enclosing module or interface.
- Else, if ``timescale` directive is previously specified within the file, then the time unit/time precision shall be set to that of the ``timescale` directive.
- Else, if the time unit/time precision is specified outside all other declarations within a compilation unit, then the time unit/time precision shall be set to that specification.
- Else, the global time unit/time precision specified by the `-noinherit_timescale` option shall be used.

Similarly in presence of other timescale options specified in this document, the following is the precedence:

`-override_timescale > -noinherit_timescale > -timescale/-unit_timescale`

Therefore, when the combination of options are used, the behavior is as follows:

- The `-override_timescale` option has the highest precedence. Therefore, the timescale specified by the `-noinherit_timescale` option is overridden by the `-override_timescale` option.
- If the `-timescale` (or the `-unit_timescale`) and the `-noinherit_timescale` options are used in the same command line, timescale from the `-noinherit_timescale` option shall be used and `-timescale` (or `-unit_timescale`) option is ignored.
- In case of three-step flow, if the `-timescale` (or `-unit_timescale`) and the `-noinherit_timescale` options are used in different command line, such as one is in `vlogan` and another is in `vcs` command line, timescale specified at `vlogan` stage cannot be overridden at `vcs` stage (except for the case when you use the `-override_timescale` option).

Figure 206 Example for -noinherit_timescale Option

a.v	b.v	c.v
<pre>module A; ... endmodule module B; ... endmodule</pre>	<pre>`timescale 1ns/1ns module X; ... endmodule `timescale 1ns/1ns module Y; ... endmodule</pre>	<pre>module P; timeunit 100ps; timeprecision 1ps; ... endmodule module Q; ... endmodule</pre>

You can use the following `vcs` and `vlogan` command lines:

```
% vcs -noinherit_timescale="10ps/1ps" a.v b.v c.v
```

Or

```
% vcs -noinherit_timescale="10ps/1ps" b.v c.v a.v
```

Or

```
% vlogan -noinherit_timescale="10ps/1ps" c.v a.v c.v
```

Any of the above commands result into the following timescale:

- module A - 10ps/1ps
- module B - 10ps/1ps
- module P - 100ps/1ps
- module Q - 10ps/1ps
- module X - 1ns/1ns
- module Y - 1ns/1ns

Key Points to Note

- The files that are read using a `include directive inherit the timescale of the file containing the `include directive. That is, the global timescale is only applied to files that are not read using an `include directive.
- Once the file is compiled or analyzed, the timescale of a module is not changed. This means that the timescale of a logic library is determined when the library is compiled, not when one of its module is incorporated using the -y option.

Option to Exclude Environment Variables During Timestamp Checks

`-vts_ignore_env=ENV1,ENV2,...`

You can use the `-vts_ignore_env=ENV1,ENV2,...` compile-time option to exclude certain environment variables from incremental compilation during VCS timestamp checks.

Consider the following testcase `test.v`:

```
module test
endmodule
```

Run the following commands:

```
%setenv myenv1 1
%setenv myenv2 2
% vcs test.v -vts_ignore_env=myenv1,myenv2

//Incremental compilation: There are no source code changes. Only the
environment variables "myenv1" and "myenv2" are changed

%unsetenv myenv1
%unsetenv myenv2

% vcs test.v -vts_ignore_env=myenv1,myenv2
```

Following is the output:

```
The design hasn't changed and need not be recompiled.
If you really want to, delete file simv.daidir/.vcs.timestamp and run VCS
again.
```

Options for Overriding Generics and Parameters

`-gfile`

You can use the `-gfile` compile-time option to override parameter and generic values through a file for both Verilog and VHDL respectively.

You must specify the file name, which contains the list of all generics and parameters that should be overridden, with the `-gfile` option.

The syntax for `-gfile` option is as follows:

```
% vcs top_level_entity_or_module -gfile parameters_or_generics_file
other_options
```

The syntax for the `parameters_or_generics_file` is as follows:

assign val path

Where,

`val`: Specifies the value that overrides the specified parameter/generic.

`path`: Specifies the absolute hierarchical path to the parameter/generic value which is to be overridden.

Note:

The `-gfile` supports only VHDL syntax for hierarchical path representation.

All escaped identifiers in the Verilog path must be converted into VHDL extended identifiers. If the escaped identifier contains '\' characters, they must be escaped with another '\' character.

For example, consider the following Verilog hierarchical path for the parameter 'P1'.

`top.dut.\inst1_\cpu .inst2.P1`

The corresponding `generics_file` entry is as follows:

```
assign 'hfffffffff /top/dut/\inst1_\\cpu\\inst2/P1
```

All 'for-generate' and 'instance-array' parentheses must be round parentheses, and the path delimiter must be '/'. All instance paths for VHDL-Top and Verilog-Top designs must start with '/'.

Example:

You can override the parameter and generic values using the `-gfile` option as follows:

```
% vcs vh_top -gfile overrides.txt
```

Where, `overrides.txt` contains the following entries:

```
assign 'hfffffffff /top/dut/\inst1_\\cpu\\inst2/P1
```

```
assign      "DUMMY"  /top/dut/\inst1_\\cpu\inst2/P2
assign      10.34   /top/dut/\inst1_\\cpu\inst2/P3
```

Supported Data Types:

The following data types are supported with the `-gfile` option:

- Integer
- Real
- String

The `-gfile` option ignores other data types with a suitable warning message.

`-pvalue`

You can use the `-pvalue` compile-time option for changing the parameter values from the `vcs` command line.

You specify a parameter with the `-pvalue` option. It has the following syntax:

```
vcs -pvalue+hierarchical_name_of_parameter= value
```

Example:

```
vcs source.v -pvalue+test.d1.param1=33
```

Note:

The `-pvalue` option does not work with a `localparam` or a `specparam`.

`-pvalue_nonlocal`

When `param_assignments` appear in a `module_parameter_port_list` and you specify the `-pvalue_nonlocal`, then any `param_assignments` that appear in the module become local parameters and shall not be overridden by any method (such as `-pvalue`).

Consider the following example (`test.v`):

```
module sub1 ();
    parameter dly = 2; // Non - Local Parameter
endmodule
module sub2 #(parameter delay = 42) ();
    parameter dly = 12; // Local Parameter As Per Verilog LRM
endmodule
module top;
    sub1 s1 ();
    sub2 #(delay(35)) s2 ();
endmodule
```

When you compile the example as follows:

```
% vcs -sverilog -pvalue+dly=20 test.v
```

VCS overrides the dly parameters from both sub1 and sub2 modules.

When you compile the example as follows:

```
% vcs -sverilog -pvalue+dly=20 -pvalue_nonlocal test.v
```

VCS overrides the dly parameters from sub1 module only.

```
-pvalue+pkg1::WIDTH=3 -pvalue+pkg2::WIDTH=4
```

You can override the package parameters during design elaboration using the -pvalue option and package scope resolution.

Consider the following example (test.v):

```
package pkg1;
    parameter WIDTH = 0;
endpackage

package pkg2;
    parameter WIDTH =1;
endpackage

module top;
    initial
        begin
            $display("pkg1 WIDTH = %0d",pkg1::WIDTH);
            $display("pkg2 WIDTH = %0d",pkg2::WIDTH);
        end
endmodule
```

When you compile the example as follows:

```
% vcs -sverilog -pvalue+pkg1::WIDTH=3 -pvalue+pkg2::WIDTH=4 test.v
./simv
```

VCS generates the following output:

```
pkg1 WIDTH = 3 pkg2 WIDTH = 4
-gv|-gvalue generic=value
```

Overrides the generic value defined in the source code with the value specified in the command line.

Example:

```
vcs work.top -gvalue /TOP/LEN=1
```

Note:

The `-gv|-gvalue` option overrides the generic value defined in the source code only if the generic is of type integer or real.

`-g|-generics cmdfile`

Overrides the default values for the design generics by using values from the file `cmdfile`. The file `cmdfile` is an include file that contains assign commands targeting design generics.

Suppressing Reporting of Generic Override Messages

By default, VCS captures the generics/parameters that are overridden during elaboration in standard output (stdout) and the compile log file.

You can use the `-param_override=dont_report` compile option to suppress printing of the generic override messages in stdout and the compile log file.

`-param_override=dont_report`

Suppress the message in stdout and compile log file.

Following is the syntax:

```
% vcs <design_files> -pvalue|-gfile|-gv|-g  
-param_override=dont_report
```

`-param_override=report`

Suppress the message in stdout and compile log file and add it in the `param_override.rpt` log file in the current working directory.

Following is the syntax:

```
% vcs <design_files> -pvalue|-gfile|-gv|-g  
-param_override=report
```

Example:

```
%vcs -sverilog t.v -pvalue P=10 -param_override=dont_report
```

Global -check_all Option

You can use the `-check_all` compile option to run all of the additional semantic checks that are traditionally run by `-check` option for VHDL and `-boundscheck` and `-ntb_opts check=all` options for SystemVerilog.

The `-check_all` option enables the available checks namely `-check` at runtime, `-boundscheck` at compile-time, and `-ntb_opts check=all` at compile-time. Therefore, warning messages are generated for the fixed-size arrays and error messages are generated for the variable-sized arrays.

Use Model

The option is supported in both VCS two-step and three-step flows.

For both two-step and three-step flows, specify the `-check_all` option in the `vcs` command line.

Consider the following test case:

Example 250 test.v

```
module test;
  logic arr[10];
  logic index,sel;
  logic [3:0] in1,in2;
  wire [3:0] out;
  initial
  begin
    in1 = 4'b1100; in2 = 4'b1000;
    #1;
    $display(arr[index]);
    $displayb(out);
  end
  DUT inst (sel,in1,in2,out);
endmodule : test

module DUT (sel,in1,in2,out);
  input sel;
  input [3:0] in1,in2;
  output logic [3:0] out;
  always @ (sel,in1,in2)
    if(sel)
      out = in1;
    else
      out = in2;
endmodule : DUT
```

If you run the test case with `-check_all` option, VCS generates the following warning message:

```
Warning-[AAII] Array access with indeterminate index
test.v, 10
Index value bits set to x or z. The array read "arr[1'bx]" has
indeterminate index value.
Simulation time = 1
```

This warning message can be upgraded to an error message only at runtime using the `-rt_f -f <file>` runtime option. This option allows you to upgrade the severity of messages. Following is the usage example:

```
%simv -rt_f -f rt_error_ctrl.f
%cat rt_error_ctrl.f
-error=AAII
```

Limitation

The feature has the following limitation:

- If you run the test case with the `-check_all` and `-xprop` options, the `-check_all` option is ignored and VCS generates the following warning message:

Warning-[XPROP-IGNORE-CHECK_ALL] `-check_all` is ignored in presence of `xprop` switches

Option to Enable Bounds Check at Compile-Time

`-boundscheck`

Enables the compile-time check for two-dimensional or three-dimensional arrays with packed dimensions. The following warning message is displayed during compile-time:

Warning-[SIOB] Select index out of bounds

This compile-time warning message is displayed in case of out-of-bounds condition.

Example

```
module tb();

reg [1:0][1:0] fifo_data[2:0]; // FIFO memory
reg some_bit; // temp signal

reg nibble[]; // Dynamic array

initial
begin
    //Packed dimension
    some_bit = fifo_data[1][1][2];

end
endmodule
```

The following warning message is displayed:

```
Warning-[SIOB] Select index out of bounds
test2.sv, 11 "fifo_data[1][1][2]"
The select index is out of declared bounds : [1:0] in module : tb.
```

Runtime Checks for Out-of-Bounds Condition

The `-boundscheck` compile option enables the runtime checks for the out-of-bounds and intermediate index access of fixed size and variable size unpacked arrays, dynamic arrays, and queues. Warning messages are generated for fixed size arrays, and error messages are generated for variable size arrays.

The following error messages or warning messages are displayed during runtime out-of-bounds index access:

- Error-[DT-OBAE] Out of bound access for queues
- Error-[DT-OBAE] Out of bound access for dynamic arrays
- Warning-[AOOBAW] Out of bound access for fixed size unpacked arrays
- Warning-[AOOBAW] Out of bound access for fixed size packed arrays

Error-[DT-OBAE] Out of Bounds Access for Queues

This runtime error message is displayed, if a queue element is accessed with an out-of-bound index.

Example

```
module tb();

int      temp;           // temp signal
int int_queue[$] = { 1, 2, 3}; //Queue

initial
begin
  //Queue
  temp = int_queue[9];
end
endmodule
```

The following error message is displayed:

```
Error-[DT-OBAE] Out of bound access
test2.sv, 9
Out of bound access on smart queue (size:3, index:9).
```

```
Simulation time = 0
Please make sure that the index is positive and less than size.
```

Error-[DT-OBAE] Out of Bounds Access for Dynamic Arrays

This runtime error message is displayed, if a dynamic array element is accessed with an out-of-bound index.

Example

```
module tb();

reg      some_bit;          // temp signal
reg nibble[]; // Dynamic array
int int_queue[$] = { 1, 2, 3}; //Queue

initial
begin
    //Dynamic array
    nibble = new[3];      // Create a 3-element array.
    some_bit = nibble[3];
end
endmodule
```

The following error message is displayed:

```
Error-[DT-OBAE] Out of bound access
test2.sv, 11
Out of bound access on dynamic array (size:3, index:3).
Simulation time = 0
Please make sure that the index is positive and less than size.
```

Warning-[AOOBAW] Array Out of Bounds Access for Fixed Size Unpacked Arrays

This runtime warning message is displayed, if a fixed size unpacked array element is accessed with an out-of-bound index.

Example

```
module tb();

reg [1:0] fifo_data[2:0]; // fifo memory
reg [1:0] some_signal;    // temp signal

initial
begin
    //Unpacked dimension
    some_signal = fifo_data[3];
```

```
    end
initial $monitor("some_signal = %b ",some_signal);
endmodule
```

The following warning message is displayed:

```
Warning-[AOOBAW] Array out of bounds access
test2.sv, 7
Array read "fifo_data[3]" is out of bounds.
Simulation time = 0
Please make sure index is within range. To disable this error message,
please remove '-boundscheck' at compile time.
```

Warning-[AOOBAW] Array Out of Bounds Access for Fixed Size Packed Arrays

This runtime warning message is displayed, if a fixed size packed array element is accessed with an out-of-bound index.

Example

```
module tb();
reg [7:0] rd_ptr;
reg [29:0] nibble;
reg some_signal3;
initial
begin
    rd_ptr = 40;
    #1 $finish();
end
assign some_signal3 = nibble[rd_ptr];
initial $monitor("some_signal3 = %b",some_signal3);
endmodule
```

The following warning message is displayed:

```
Warning-[AOOBAW] Array out of bounds access
test2.sv, 11
Array read "nibble[40]" is out of bounds.
Simulation time = 0
Please make sure index is within range. To disable this error message,
please remove '-boundscheck' at compile time.
```

The following error messages or warning messages are displayed during indeterminate index access, where array index has value X or Z:

- Error-[DT-OBAE] Intermediate access for dynamic arrays
- Warning-[AAIIW] Array access with intermediate index
- Warning-[AAIIW] Array access with intermediate index for fixed size packed arrays

Error-[DT-OBAE] Intermediate Access for Dynamic Arrays

This runtime error message is displayed, if a dynamic array element is accessed with an index value `x` or `z`.

Example

```
module tb();
reg [7:0] rd_ptr = 8'bxxxxxxxx;
reg nibble[]; // Dynamic array
reg some_signal3; // temp signal

initial
begin
    //Dynamic array
    nibble = new[3]; // Create a 3-element array.
    some_signal3 = nibble[rd_ptr];
end
endmodule
```

The following error message is displayed:

```
Error-[DT-OBAE] Out of bound access
test2.sv, 10
Out of bound access on dynamic array (size:3, index:255)
Simulation time = 0
Please make sure that the index is positive and less than size.
```

Warning-[AAIIW] Array Access with Intermediate Index

This runtime warning message is displayed, if a fixed size unpacked array element is accessed with an index value `x` or `z`.

Example

```
module tb();
reg [17:0] fifo_data[255:0]; // fifo memory
reg [7:0] rd_ptr;
wire [7:0] some_signal; // temp signal
initial begin
    rd_ptr = 8'dx;
    #1 $finish();
end

assign some_signal = fifo_data[rd_ptr];
initial $monitor("some_signal = %b",some_signal);

endmodule
```

The following warning message is displayed:

```
Warning-[AAIIW] Array access with indeterminate index
test2.sv, 9
Index value bits set to x or z. The array read "fifo_data[1'bx]" has
indeterminate index value.
Simulation time = 0
To disable this warning message, please remove '-boundscheck' at compile
time. To upgrade this warning to error, add "-error=AAII" to simv
runtime command.
```

Warning-[AAIIW] Array Access with Intermediate Index for Fixed Size Packed Arrays

This runtime warning message is displayed, if a fixed size packed array element is accessed with an index value x or z.

Example

```
module tb();
reg [7:0] rd_ptr;
reg [29:0] nibble;
reg some_signal3;
initial
begin
  rd_ptr = 8'bxxxxxxxxx;
#1 $finish();
end
assign some_signal3 = nibble[rd_ptr];
initial $monitor("some_signal3 = %b",some_signal3);
endmodule
```

The following warning message is displayed:

```
Warning-[AAIIW] Array access with indeterminate index
test2.sv, 10
Index value bits set to x or z. The array read "nibble[1'bx]" has
indeterminate index value.
Simulation time = 0
To disable this warning message, please remove '-boundscheck' at compile
time. To upgrade this warning to error, add "-error=AAII" to simv
runtime command.
```

Option to Enable Extra Runtime Checks in VHDL

You can use the following option to enable extra runtime checks in VHDL:

-check

This option enables the extra runtime checks in the VHDL code. The following error messages are displayed during runtime:

- Error-[SIMERR_FDIVZERO_SCOPE] Divide by Zero Error
- Error-[SIMERR_INCONSISTENTIC] Incorrect Binding Range
- Error-[SIMERR_INCONSISTENTIS] Subtype constraints inconsistencies

Error-[SIMERR_FDIVZERO_SCOPE] Divide by Zero Error

This runtime error message is displayed when the divisor is zero in floating point operation.

Example:

```
entity e is
end;

architecture a of e is
begin
process
variable v1,v2,v : real;
begin
-- real divide by Zero :
v1 := 1.1 ;
v2 := 0.0;
v := v1/v2;
end process;
end a;
```

The following error message is displayed:

```
Error-[SIMERR_FDIVZERO_SCOPE] Divide by Zero Error
test11.vhd, 11 v := v1/v2;
Error in floating point operation, divide by zero
in scope:/E/_P0 at 0 NS from process /E/_P0.
test11.vhd(11): v := v1/v2;
```

Error-[SIMERR_INCONSISTENTIC] Incorrect Binding Range

This runtime error message is displayed due to incorrect binding range.

Example

```
entity e is
end entity;
architecture a of e is
procedure p (l, r : integer) is
variable x : string(l to r);
```

```

begin
  end procedure;
begin
  c: process
  begin
    p(0, 3);
    wait;
  end process;
end architecture;

```

The following error message is displayed:

```

Error-[SIMERR_INCONSISTENTIC] Incorrect binding range
test13.vhd, 5 variable x : string(l to r);
  Binding range (0 to 3) is not consistent with declared range (1 to
2147483647) at 0 NS from process /E/C.
  test13.vhd(5):           variable x : string(l to r);

```

Error-[SIMERR_INCONSISTENTIS] Subtype Constraints Inconsistencies

This runtime error message is displayed, if the constraints of the two subtypes do not match exactly.

Example

```

entity e is
  generic (N : integer := 4);
end entity;
architecture a of e is
  subtype t is string(N downto 1);
  type a is access t;
begin
  P: process
    variable x : a;
  begin
    x := new string(1 to N);
  end process P;
end architecture;

```

The following error message is displayed:

```

Error-[SIMERR_INCONSISTENTIS] Subtype constraints inconsistencies
test12.vhd, 11
  x := new string(1 to N);
  Two subtypes whose constraints must match exactly differed. One is (1
to 4), the other one is (4 downto 1). The mismatch occurs at 0 NS from
process /E/P.
Please make sure that the constraints of the two subtypes match exactly.
  test12.vhd(11):           x := new string(1 to N);

```

Options to Detect Multiple Conflicts on Buses

VCS support options to detect multiple conflicts on buses. Buses are considered to be in conflict or floating state when:

- Any bus for which two or more active drivers have different strength values. Active drivers are those with Strong (strong0, strong1) or Supply (supply0, supply1) strength.
- Any external bus (that is inout or output port of DUT) for which two or more active drivers have the same values is considered to be in conflict state.
- Any bus is in floating state when all drivers are HiZ.

VCS reports an error if the state has been held for an interval greater than that provided in the command line/ \$busconfloat task argument.

In some cases, a driver can be a driver of multiple buses.

Note:

For external buses, multiple drivers with different values also creates a conflict scenario. However, if multiple drivers drive the same value, it is also considered a conflict.

Use Model

You can enable this feature by using one of the following option:

- Using compile time option: This option is as follows:

```
+busconfloat+IBC_interval+IBF_interval+EBC_interv al+EBF_interval
+DUT_instance_path
```

- Using system task. This option is as follows:

```
$busconfloat(IBC_interval, IBF_interval, EBC_interval, EBF_interval,
"DUT_instance_path")
```

where,

`IBC_interval` and `IBF_interval` are the internal bus conflict and floating intervals (the interval that conflict must be present before reporting a message).

`EBC_interval` and `EBF_interval` are the external bus conflict and floating state intervals respectively. If both are specified, then the parameters given in the compile time options are observed. VCS also detects buses based on the `+busdrive[+config_file]` compile-

time option where the configuration file specification is optional. In this case, the following scenarios are considered as buses:

- All multiply-driven nets that are highconns of output and inout ports of modules that have specparam `BUSCHECK$ = 1`. Such nets are considered internal or external using the same rules. Only highconns is considered because modules with this specparam are library cells (with no instantiations). For example,

```
module bar(output x);
  specparam BUSCHECK$=1;
  foo(x);
endmodule
module foo(inout y);
  bar(y);
endmodule
module goo;
  wire y;
  foo(y);
  assign y = ...
endmodule
```

Here, `bar.x` is identified as a driver because of the `BUSCHECK$` specparam. It's highconn `foo.y` is considered a bus of interest since it also has another `goo.y` driver.

- All multiply-driven nets identified through configuration file. These are highconns of either output or inout ports of specified instance or specified port of an instance. For example, consider the following configuration file (invoked with `+busconfig +config_file`):

```
module {AAAA} {BusDrive};
instance {top.BBB, top.DDD} {BusDrive};
port {top.CCC.D*.io} {BusDrive};
```

You can consider as buses the highconns of:

1. inout and output ports of all instances of module AAAA;
2. inout and output ports of instance `top.BBB` and `top.DDD`; and
3. `port {top.CCC.D*.io}` (where instance is given using wildcard).

The tracing of drivers to find buses is done through concats and select expressions in the port connections. Other arbitrary expressions in the port connection are not considered (for example, `.P(a&b)`). Also, cont assigns and gates are not used for tracing drivers. Note that only if a BUSCHECK driver can be traced to the

DUT, the node in the DUT is considered to be a bus.

The `+busdrive` option is interpreted as follows when either `$busconfloat` or `+busconfloat+*` is given:

1. no `+busdrive*`: All internal and external buses are detected.
2. `+busdrive`: Picks only buses in cells with BUSCHECK. Therefore, `+busdrive` supercedes `$busconfloat`.
3. `+busdrive+cfgfile`: Picks only buses driven by cells given in cfgfile. Therefore, cfgfile supercedes `$busconfloat`.
4. Both `+busdrive+cfgfile` and `+busdrive`: Picks buses driven by the cells specified in cfgfile and buses in cells with BUSCHECK.

The wild card specification in the configuration file can have the wild card character (asterisk) at the end of the module name or leaf level instance name. The following table shows an example of valid and incorrect syntax:

Table 78 Sample Wildcard Sample

Valid Syntax	Incorrect Syntax
<pre>module {AAAA*} {BusDrive}; instance {top.BBB*} {BusDrive}; port {top.CCC.D*.io} {BusDrive};</pre>	<pre>module {AA*AA} {BusDrive}; # module name doesn't end in star instance {top.B*BB} {BusDrive}; #instance name doesn't end in star instance {top.*.BBB} {BusDrive}; # not leaf-level instance instance {top.CCC*.D.io} {BusDrive}; # not leaflevel instance</pre>

Outputs

Different files are created for external, internal bus conflict and floating messages. Therefore, there are four output files. The outputs

are as follows:

- `conflict.external.rpt`

This report is created for conflicts in external buses.

- `conflict.internal.rpt`

This report is created for conflicts in internal buses.

- float.internal.rpt

This report is created for conflicts in float internal buses.

- float.external.rpt

This report is created for conflicts in float external buses.

The report contains the following information:

- Full path name of the bus and the time at the end of the conflict or floating state.
- Active drivers (full path names) and their values at the beginning of conflict/floating state.
- Active drivers and their values at the end of conflict/floating state.

Usage Example

Consider the following testcase:

Example 251 test.v

```
`timescale 1ns/1ps

module tb;

reg [0:3] in1, in2, in3;
reg [0:3] en1, en2;
wire [0:3] en3;
wire [0:3] out1, out2, out3;

dut dut1 (in1, in2, in3, en1, en2, en3, out1, out2, out3);

`ifdef MULT_TB_DRV
tb_mod1 m1(in1, in2, out1, out2);
// tb_mod1 m2(in2, in1, out1, out2);
`endif

initial begin
$monitor($time,, "en1=%v %b en2=%v %b en3=%v %b in1=%v %b in2=%v %b
in3=%v %b out1=%v %b out2=%v %b out3=%v %b tb.dut1.E4=%b tb.dut1.E5=%b",
en1, en1, en2, en2, en3, en3, in1, in1, in2, in2, in3, in3, out1, out1,
out2, out2, out3, out3, tb.dut1.E4, tb.dut1.E5);
#300 $finish;
end

initial begin
#20;
en1=4'b0000;
#20;
en2=4'b1111;
```

Appendix C: Compilation/Elaboration Options

Options to Detect Multiple Conflicts on Buses

```

#20;
in1=4'b1111;
in2=4'b0000;
in3=4'b1100;
#40;
en1=4'b0000;
#20;
en2=4'b1111;
#20;
in2=4'b0000;
#20;
en2=4'b1111;
in3=4'b0000;
#20;
en1=4'b0000;
in3=4'b1111;
#20;
en1=4'b0000;
en2=4'b1111;
in1=4'b0000;
in2=4'b1111;

#2;
$display($time,, " tb.dut1.C_O12.E3[3] = %b tb.dut1.E4[3] = %b",
tb.dut1.E3[3], tb.dut1.E4[3]);
#20;
en1=4'b0000;
end

`ifdef MULT_TB_DRV_FORCE
initial begin
  #40;
  force tb.dut1.E4=4'b1111;
  #15;
  release tb.dut1.E4;
  #75; force tb.dut1.O1=4'b1111;
  $display($time,, "After force ---tb.dut1.C_O11[3].out = %b,
tb.dut1.C_O12.out[3] = %b tb.dut1.O1[3] = %b", tb.dut1.C_O11[3].out,
tb.dut1.C_O12[3].out, tb.dut1.O1[3]);
  #20; release tb.dut1.O1;
  $display($time,, "After release ---tb.dut1.C_O11[3].out = %b,
tb.dut1.C_O12.out[3] = %b tb.dut1.O1[3] = %b", tb.dut1.C_O11[3].out,
tb.dut1.C_O12[3].out, tb.dut1.O1[3]);
  end
`endif

endmodule

module tb_mod1(in1, in2, out1, out2);
  input [0:3] in1;
  input [0:3] in2;
  output [0:3] out1;
  output [0:3] out2;

```

Appendix C: Compilation/Elaboration Options

Options to Detect Multiple Conflicts on Buses

```

`ifndef ONE_TB_DRV
`ifdef SUPPLY_ST
    assign (supply0, supply1) out1 = ~in1;
    assign (supply0, supply1) out2 = ~in2;
    assign (supply0, supply1) tb.dut1.E4 = in1;
`elsif WEAK_PULL_ST_TB
    assign (weak0, pull1) out1 = in1;
    assign (weak0, pull1) out2 = in2;
    assign (weak0, pull1) tb.dut1.E4 = in1;
`else
    assign out1 = in1;
    assign out2 = in2;
`endif
`else
    assign tb.dut1.E4 = in1;
`endif

endmodule

module dut(I1, I2, I3, E1, E2, E3, O1, O2, O3);
    input [0:3] I1;
    input [0:3] I2;
    input [0:3] I3;
    input [0:3] E1;
    input [0:3] E2;
    inout [0:3] E3;
    output [0:3] O1;
    output [0:3] O2;
    output [0:3] O3;

    tri [0:3] E4, E6;
    wand [0:3] E5;

`ifndef ONE_TB_DRV
`ifdef ONE_DUT_DRV
    child1 C_O11[0:3] (E3, E1, E2, I1, O1);
    child2 C_O12[0:3] (E3, , , I1, O1);
`else
    child1 C_O11[0:3] (E3, E1, E2, I1, O1);
    child1 C_O21[0:3] (E3, E1, E2, I1, O2);
    child2 C_O12[0:3] (E4, E1, E2, I2, O1);
    child2 C_O22[0:3] (E4, E1, E2, I2, O2);
    child3 C3_O13 (E4, I1, O2);
    child3 C3_O23 (E4, I2, O2);
`endif
`endif

endmodule

module child1(E_OUT, E_CTRL, E_IN, in, out);
    input E_IN;
    input in;

```

Appendix C: Compilation/Elaboration Options

Options to Detect Multiple Conflicts on Buses

```

inout E_CTRL;
inout E_OUT;
output out;

CELL1 c1(E_OUT, E_CTRL, E_IN, in, out);

endmodule

module child2(E_OUT, E_CTRL, E_IN, in, out);
    input E_IN;
    input in;
    inout E_CTRL;
    inout E_OUT;
    output out;

    CELL2 c2(E_OUT, E_CTRL, E_IN, in, out);

endmodule

module child3(E, in, out);
    inout [0:3] E;
    input [0:3] in;
    output [0:3] out;

    CELL3 c3_0(E[0], tb.dut1.E1[0], tb.dut1.E3[0], in[0], out[0]);
    CELL3 c3_1(E[1], tb.dut1.E1[1], tb.dut1.E3[1], in[1], out[1]);
    CELL3 c3_2(E[2], tb.dut1.E1[2], tb.dut1.E3[2], in[2], out[2]);
    CELL3 c3_3(E[3], tb.dut1.E1[3], tb.dut1.E3[1], in[3], out[3]);

endmodule

```

Example 252 LIB.v

```

`timescale 1ns/1ps

module CELL1(E_OUT, E_CTRL, E_IN, in, out);

    input E_IN;
    input in;
    input E_CTRL;
    inout E_OUT;
    output out;

`ifdef SUPPLY_ST
    bufif0 (supply0, supply1) (E_OUT, E_IN, E_CTRL);
    bufif0 (supply0, supply1) (out, in, E_OUT);
`elsif WEAK_PULL_ST
    bufif0 (weak0, pull1) (E_OUT, E_IN, E_CTRL);
    bufif0 (weak0, pull1) (out, in, E_OUT);
`else
    bufif0 (E_OUT, E_IN, E_CTRL);
    bufif0 (out, in, E_OUT);
`endif

```

Appendix C: Compilation/Elaboration Options

Options to Detect Multiple Conflicts on Buses

```

specify
  specparam BUSCHECK$=1;
  (E_CTRL => E_OUT) = (1, 1);
  // (in => out) = (2, 2);
endspecify

endmodule

module CELL2(E_OUT, E_CTRL, E_IN, in, out);

  input E_IN;
  input in;
  input E_CTRL;
  inout E_OUT;
  output out;

`ifdef SUPPLY_ST
  notif0 (supply0, supply1) (E_OUT, E_IN, E_CTRL);
  bufif1 (supply0, supply1) (out, in, E_OUT);
`elsif WEAK_PULL_ST
  notif0 (weak0, pull1) (E_OUT, E_IN, E_CTRL);
  bufif1 (weak0, pull1) (out, in, E_OUT);
`else
  notif0 (E_OUT, E_IN, E_CTRL);
  bufif1 (out, in, E_OUT);
`endif

specify
  specparam BUSCHECK$=1;
  (E_CTRL => E_OUT) = (1, 1);
  // (in => out) = (2, 2);
endspecify

endmodule

module CELL3(E_OUT, E_CTRL, E_IN, in, out);

  input E_IN;
  input in;
  input E_CTRL;
  inout E_OUT;
  output out;

`ifdef SUPPLY_ST
  bufif0 (supply0, supply1) (E_OUT, E_IN, E_CTRL);
  notif0 (supply0, supply1) (out, in, E_OUT);
`elsif WEAK_PULL_ST
  bufif0 (weak0, pull1) (E_OUT, E_IN, E_CTRL);
  notif0 (weak0, pull1) (out, in, E_OUT);
`else
  bufif0 (E_OUT, E_IN, E_CTRL);
  notif0 (out, in, E_OUT);
`endif

```

Appendix C: Compilation/Elaboration Options

Options to Detect Multiple Conflicts on Buses

```

`endif

specify
  specparam BUSCHECK$=1;
  (E_CTRL => E_OUT) = (1, 1);
 // (in => out) = (2, 2);
endspecify

endmodule

```

Use the following commands to run the example:

- vcs +vcstdumpvars -sverilog test.v -v LIB.v +busconffloat
+50000+10+10+10+tb.dut1 +busdrive +define+SUPPLY_ST+ONE_DUT_DRV
-full64
- ./simv

In this example, external conflict is observed. The output `conflict.external.rpt` is generated. The content of the `conflict.external.rpt` file is as follows:

External conflicts

```

Bus name: tb.dut1.O1[3]
0 SuX tb.dut1.C_O11[3].c1.out
0 SuX tb.dut1.C_O12[3].c2.out
60000 Low tb.dut1.C_O11[3].c1.out
60000 SuX tb.dut1.C_O12[3].c2.out

Bus name: tb.dut1.O1[2]
0 SuX tb.dut1.C_O11[2].c1.out
0 SuX tb.dut1.C_O12[2].c2.out
60000 Low tb.dut1.C_O11[2].c1.out
60000 SuX tb.dut1.C_O12[2].c2.out

Bus name: tb.dut1.O1[1]
0 SuX tb.dut1.C_O11[1].c1.out
0 SuX tb.dut1.C_O12[1].c2.out
60000 Low tb.dut1.C_O11[1].c1.out
60000 SuX tb.dut1.C_O12[1].c2.out

Bus name: tb.dut1.O1[0]
0 SuX tb.dut1.C_O11[0].c1.out
0 SuX tb.dut1.C_O12[0].c2.out
60000 Low tb.dut1.C_O11[0].c1.out
60000 SuX tb.dut1.C_O12[0].c2.out

Bus name: tb.dut1.E3[0]
0 StX tb.dut1.C_O11[0].c1.E_OUT
0 StX tb.dut1.C_O12[0].c2.E_OUT
300000 Su1 tb.dut1.C_O11[0].c1.E_OUT
300000 SuX tb.dut1.C_O12[0].c2.E_OUT

```

```

Bus name: tb.dut1.E3[3]
0 StX tb.dut1.C_O11[3].c1.E_OUT
0 StX tb.dut1.C_O12[3].c2.E_OUT
300000 Su1 tb.dut1.C_O11[3].c1.E_OUT
300000 SuX tb.dut1.C_O12[3].c2.E_OUT

Bus name: tb.dut1.E3[2]
0 StX tb.dut1.C_O11[2].c1.E_OUT
0 StX tb.dut1.C_O12[2].c2.E_OUT
300000 Su1 tb.dut1.C_O11[2].c1.E_OUT
300000 SuX tb.dut1.C_O12[2].c2.E_OUT

Bus name: tb.dut1.E3[1]
0 StX tb.dut1.C_O11[1].c1.E_OUT
0 StX tb.dut1.C_O12[1].c2.E_OUT
300000 Su1 tb.dut1.C_O11[1].c1.E_OUT
300000 SuX tb.dut1.C_O12[1].c2.E_OUT

```

For example, in the `tb.dut1.O1[0]` external bus there was a conflict between time 0 and 60000 and in the `tb.dut1.E3[0]` external bus there was a conflict between time 0 and 300000. Both the external buses have two drivers and the values are given for each at the start and end times of the conflict.

Also, all the observed external and internal buses are listed in `observed.external.rpt` and `observed.internal.rpt` reports respectively.

Limitations

The limitations of this feature are as follows:

- The selection mechanism using wildcard (`top.CCC.D*.io`) is not supported. For example,

```

module A(output q, output q3, ...)
specparam BUSCHECK$=1;
endmodule
module B(output q, output q3, ...)
wire q2;
assign q = q1 & q2;
A i1(q1, q3&q4);
endmodule
module DUT(...);
B i2(bus, bus1, ...);
endmodule

```

In the example, `A.q` is a driver and thus `B.q1` is considered for bus identification. However, you do not trace through to `B.q` because of the and gate in between `B.q1` and `B.q`. Similarly, `highconn q3&q4` is not traced up from `A.q3` because it is not a concat or select expression.

- Only wire, tri and supply (and their variations like wor, tri1) are supported.
- Port connections that lead from leaf ports to DUT must contain only simple name, concat or bit select. Multi-concat is not supported.
- Force is not considered for determining float or conflict state. A bus is considered floating if all of its drivers are HiZ even if the net has been forced to some value.
- FGP is not supported. Busconfloat is ignored in that case.
- MX designs are not supported.

Gate-Level Simulations

This section contains the following chapter:

- [Local Timescale Precision on Timing Check for Modules](#)

Local Timescale Precision on Timing Check for Modules

VCS used global time precision to scale delay or timing check limit values at compile time, when there were separate timescale or precision for each module.

VCS allows you to define local time precision for modules that uses different timescale units and precisions. To enable this feature, you can use a configuration file or the `-auto_tchk_local_precision` compile-time option.

Use Model

You can use either of the following use models to enable this feature:

- [Using configuration file](#)
- [Using the `-auto_tchk_local_precision` option](#)

Using configuration file

In the configuration file, you can specify the module name where you want to apply local precision. Following is the use model:

```
% vcs +optconfigfile+<config_filename> -full164
```

where, the format of the configuration file is as follows:

```
module {<module_name>} {tcLocalPrecision};
```

For example,

```
module {module3} {tcLocalPrecision};
```

```
module {module4} {tcLocalPrecision};
```

Using the **-auto_tchk_local_precision** option

This option automatically detects the overflows in the modules and applies local precision. It also creates a `tcLocalPrecision.txt` file which lists the modules where local precision is applied. Following is the use model:

```
% vcs <filename> -auto_tchk_local_precision -full164
```

Usage Example

Consider the following `test.v` file.

Example 253 test.v file

```
`timescale 1ns/1fs
module top();
reg CLK, D;
wire OP1;
dff f1(OP1, D, CLK);
//clock generator
always #5000000 CLK = ~CLK;
//print
initial begin
    D=0;    CLK=0;
    D, CLK, OP1);
end
//test
initial begin
    #43000000 D=1;
    #4000000 D=0;
    #1000000 D=1;
    #10000000 D=0;
    #20000000 $finish;
end
endmodule
//Simple D Flip Flop
`timescale 1ns/1ns
module dff (q, d, clk);
    output q;
    input d, clk;
    reg q, notifier;
    always @ (posedge (clk)) q=d;
    specify
        specparam t_setup=3000000;
        $setup(d, posedge clk, t_setup, notifier);
    endspecify
endmodule
```

If you are using the configuration file, the content of the `test.cfg` file is as follows:

```
module {dff} {tcLocalPrecision};
```

Run the example using the following commands:

- % vcs test.v +optconfigfile+test.cfg -full164
- % simv

If you are using the compile-time option, run the example using the following commands:

- % vcs test.v -auto_tchk_local_precision -full164
- % simv

The following violation is reported in the output:

```
"test.v", 40: Timing violation in top.f1
    $setup( d:430000000000000, posedge clk:450000000000000, limit:
3000000000000 );
```

Note:

The `-auto_tchk_local_precision` option, generates the `tcLocalPrecision.txt` file. The content of this file is as follows:

```
=====Modules require local precision=====
```

```
module dff TimeScale is: 1 ns/1 ns
```

Limitations

This features is not supported for the following:

- 32-bit build
- VHDL

Options for Additional Multiple Driver Checks

-varindex_drivers

As per *SystemVerilog LRM 1800TM-2017*, Section 6.5 Nets and variables, an error is issued when multiple continuous assignments or a mixture of procedural and continuous assignments writing to any term in the expansion of a written longest static prefix of a variable. VCS considers the longest static prefix for multiple driver analysis as defined in LRM under the `-varindex_drivers` compile-time option.

In VCS three-step flow, use the `-varindex_drivers` option at the VCS stage.

Usage Example

Consider the following example:

Example 254 test.sv

```
module test;
  logic[2:0] vec;
  assign vec[2]= 1;
  initial for(int i=0;i<2;i++) vec[i] = 0;
endmodule:test
```

Default behavior:

Run the design using the following commands:

```
vcs test.sv -sverilog -full64
```

The output generated is as follows:

The test compiles without any errors.

Behavior with `-varindex_drivers` option:

Run the design using the following commands ():

```
vcs test.sv -sverilog -full64 -varindex_drivers
```

The output generated is as follows:

```
Error-[ICPSD] Illegal combination of drivers
test.sv, 2
  Illegal combination of structural and procedural drivers.
  Variable "vec" is driven by an invalid combination of structural and
  procedural drivers. Variables driven by a structural driver cannot have
any
other drivers.
This variable is declared at "test.sv", 2: logic [2:0] vec;
The first driver is at "test.sv", 4: vec[i] = 0;
The second driver is at "test.sv", 3: assign vec[2] = 1;
```

`-ntb_opts multi_driver_no_source_info`

In the partition compile flow, VCS generates an error message in the presence of an invalid combination of structural and procedural drivers or an invalid combination of two structural drivers for the variables of a design.

VCS dumps the driver information of all the variables of the design during elaboration, which significantly increases the compile time. You can use the `-ntb_opts multi_driver_no_source_info` compile-time option to disable the dumping of the driver information of variables.

Usage Example

Consider the following example in which the `test.sv` test case is run with the `test.cfg` configuration file.

Example 255 test.sv

```
module top1;
int intVar;
assign intVar = 25;
endmodule:top1

module top2;
initial top1.intVar = 17;
endmodule:top2
```

Example 256 test.cfg

```
partition cell top1;
partition cell top2;
```

Default behavior:

Run the design using the following commands:

```
%vcs -sverilog -full64 -partcomp test.sv +optconfigfile+test.cfg
```

The output generated is as follows:

```
Error-[ICPSD] Illegal combination of drivers
test.sv, 2
Illegal combination of structural and procedural drivers.
Variable "intVar" is driven by an invalid combination of structural and
procedural drivers. Variables driven by a structural driver cannot have
any other drivers.
This variable is declared at "test.sv", 2: int intVar;
The first driver is at "test.sv", 7: top1.intVar = 17;
Hierarchical path: top2
The second driver is at "test.sv", 3: assign intVar = 25;

Hierarchical path: top1
```

Behavior with the `-ntb_opts multi_driver_no_source_info` option:

Run the design using the following commands:

```
%vcs -sverilog -full64 -partcomp test.sv +optconfigfile+test.cfg
-ntb_opts multi_driver_no_source_info
```

The output generated is as follows:

```
Error-[ICPSD] Illegal combination of drivers
test.sv, 2
```

```
Illegal combination of structural and procedural drivers.
Variable "intVar" is driven by an invalid combination of structural and
procedural drivers. Variables driven by a structural driver cannot have
any other drivers.
This variable is declared at "test.sv", 2: int intVar;
The first driver is at "test.sv", 7
Hierarchical path: top2
The second driver is at "test.sv", 3
Hierarchical path: top1
```

Option for Function Call Evaluation at Time Zero

SystemVerilog LRM semantics are not well-defined for continuous assignments at time zero with respect to the function call evaluation on the RHS. The behavior of the `always_comb` process is well-defined for a procedural assignment containing a function call on the RHS because `always_comb` is guaranteed to execute once at time zero.

If you want a continuous assignment containing a function call on the RHS to be executed at time zero, you can use the `func_eval_timezero` option at compile time.

The following example demonstrates the change in behavior with and without this option:

```
module top;
    function automatic [2: 0] foo;
        input rp;
        bit [2:0]OUT_RP_PRESENT = 3'b111;
        integer i;
        begin
            $display( "At time:%0t foo called. rp = %b", $time, rp);
            for (i=0; i<3; ++i)foo[i] = (rp == i[0]) & OUT_RP_PRESENT[i];
        end
    endfunction

    reg rp;
    reg [2:0] foo_checker;
    assign foo_checker = foo(rp);
endmodule
```

To compile the above design, run the following command:

```
% vcs -sverilog -full64 test.v +vcs+initreg+random -func_eval_timezero
```

Default Output (Without the `func_eval_timezero` Option)

At time:0 foo called. rp = 0

`reg foo_checker` in `assign foo_checker=foo(rp)` is not updated at time zero, when `rp=x`.

Output With the `func_eval_timezero` Option

```
At time:0 foo called. rp = x
At time:0 foo called. rp = 0
```

General Options

Specifying Directories for 'include' Searches

`+incdir+directory+`

Specifies the directory or directories in which VCS searches for include files used in the ``include` compiler directive.

Files to be included and specified with the `'include` compiler directive are called included files. VCS searches for included files in the following order:

1. In the current directory
2. In the directories specified with this `+incdir` compile-time option.

You can specify more than one directory separated by the plus (+) character. For example:

`+incdir+dir1+dir2`

In this example subdirectories `dir1` and `dir2` are in the current directory.

`+incdir+/file_sys/server/design_group/design_lib`

You can also specify an absolute path name.

Enable the VCS/SystemC Cosimulation Interface

`-sysc`

Enables SystemC cosimulation engine.

`-sysc=adjust_timeres`

Determines the finer time resolution of SystemC and HDL in case of a mismatch, and sets it as the simulator's timescale. VCS may be unable to adjust the time resolution if you elaborate your HDL with the `-timescale` option or use the `sc_set_time_resolution()` function call in your SystemC code. In such cases, VCS reports an error and does not create simv.

You must use this option along with the `-sysc` option.

The `-sysc=adjust_timeres` option is not supported in two-step flow. It is only supported in three-step (UUM) flow.

TetraMAX

`+tetramax`

Enables splitting of TetraMAX's large testbench to improve VCS capability and to reduce compile time.

Suppressing Port Coersion to inout

`+noportcoerce`

Prevents VCS from coercing ports to inout ports, which is the default condition. This option is the equivalent of the '`noportcoerce`' compiler directive.

Allow Inout Port Connection Width Mismatches

`+noerrorIOPCWM`

Changes the error condition, when a signal is wider or narrower than the inout port to which it is connected, to a warning condition, thus allowing VCS to create the simv executable after displaying the warning message.

Specifying a VCD File

`+vcs+dumpvars`

A substitute for entering the `$dumpvars` system task, without arguments, in your Verilog code.

Enabling Dumping

`+vcs+vcdpluson`

A compile-time substitute for the `$vcdpluson` system task, the `+vcs+vcdpluson` option enables recording in the VPD file transition times and values for the entire design (except SystemVerilog memories and multi-dimensional arrays (MDAs) that have unpacked dimensions). If `-debug_access` is not specified, then this option adds `-debug_access`.

Enabling Identifier Search

You can use the following elaboration options to enable and control the Search Identifiers feature:

- `-genid_db`
- `-nogenid_db`
- `-debug_access+idents_db`
- `-debug_access+all`

Use the `-genid_db` option in combination with a debug option, for example, as shown below, to enable Search Identifiers feature and prepare the internal search database.

```
% vcs -genid_db -debug_access+idents_db top.v
```

If you use `-genid_db` without a debug option, VCS issues a warning message saying that the feature is not enabled.

If you elaborate your design with `-debug_access+all`, but without `-genid_db`, then VCS creates the database during the first search query. This postpones most of the disk space and CPU overhead.

Specify `-nogenid_db`, if you want to completely avoid any disk space and CPU time overhead caused by Search Identifiers. You must use this option in combination with `-debug_access+all`.

Specifying a Log File

`-l filename`

Specifies a file where VCS records compilation messages. If you also enter the `-R` option, VCS records messages from both compilation and simulation in the same file.

`-a logfilename`

Captures simulation output and appends the log information in the existing log file. If the log file doesn't exist, then this option would create a log file.

Changing Source File Identifiers to Upper Case

`-u`

Changes all the characters in identifiers to uppercase. It does not change identifiers in quoted strings such as the first argument to the `$monitor` system task.

Defining a Text Macro

```
+define+macro=value+
```

Defines a text macro in your source code to a value or character string. You can test for this definition in your Verilog source code using the `'ifdef` compiler directive.

The `=value` argument is optional.

For example:

```
vcs design.v +define+USETHIS
```

The macro is used inside the source file using the `'ifdef` compiler directive. If this macro is not defined using the `+define` option, then the `else` portion in the code takes priority.

```
`ifdef USETHIS
    package p1;
    endpackage
`else
    package p2;
    Endpackage
`endif
```

Undefining a Text Macro

```
+undefine+<macroname>
```

Undefines a text macro in your source code that is already defined.

Note:

If the text macro is defined more than once in the source code, undefining does not happen because the `undef` flag is overwritten.

For example:

```
% vcs temp.v -sverilog -full164 +undefine+A
```

After compiling below example with the above vcs command, string "Fail" does not appear in simulation log, because macro A is no longer defined in the design once `+undefine+A` compile time option is given in the vcs command.

Example 257 temp.v

```
module top;

logic a;
`define A
```

```
`ifdef A
initial begin
$display ("Fail");
end
`endif
endmodule
```

Option for Macro Expansion

-p1800_macro_expansion

This option is used for LRM compliance to support macro expansion. This option produces results that are more LRM-compliant and accurate especially for SystemVerilog macros.

The syntax is:

```
% vcs [elab_options] test.sv -sverilog
-p1800_macro_expansion
```

For example, consider the following testcase:

```
module top;
logic [3:0] addr0_for_bank0='d10;
`define VAR(ANUM,BNUM) addr``ANUM``_for_bank``BNUM
`define NAME(STR) $display(`"``"STR````" is %d\n",STR);
`define ARG addr0_for_bank0

initial begin
`NAME(`VAR(0,0));
`NAME(`ARG)
end
endmodule
```

If you run the testcase without -p1800_macro_expansion option, VCS generates the following output:

```
"`VAR(0,0)" is 10
"addr0_for_bank0" is 10
```

If you run the testcase with -p1800_macro_expansion option, VCS generates the following output:

```
"addr0_for_bank0" is 10
"addr0_for_bank0" is 10
```

Specifying the Name of the Executable File

-o name

Specifies the name of the executable file. In UNIX, the default is `simv`.

Returning The Platform Directory Name

`-platform`

Returns the name of the `platform` directory in your VCS installation directory. For example, when you install VCS on a Solaris version 5.4 workstation, VCS creates a directory named, `sun_sparc_solaris_5.4`, in the directory where you install VCS. In this directory are subdirectories for licensing, executable libraries, utilities, and other important files and executables. You need to set your path to these subdirectories. You can do so by using this option:

```
set path=($VCS_HOME/bin\  
$VCS_HOME/'$VCS_HOME/bin/vcs -platform'/bin\$path)
```

Maximum Donut Layers for a Mixed HDL Design

`-maxLayers value`

Sets the maximum number of donut layers for a mixed HDL design. The default value is 8.

Enabling feature beyond VHDL LRM

`-xlrn`

Enables VHDL features beyond those described in *IEEE Standard VHDL 1076-2008 LRM*.

Enabling Loop Detect

`+vcs+loopreport+number`

It is mandatory to include the `+vcs+loopreport+number` option at compile-time, though the threshold number can be overridden at runtime.

When `+vcs+loopreport+number` is specified at compile time, VCS does the following based on the option specified at runtime:

- If `number` is not specified at runtime, VCS checks if the simulation event loops for 2,000,000 times (by default) in the same simulation time tick, and issues a runtime warning message. VCS also terminates the simulation and generates a report when a zero delay loop is detected.

- If `+vcs+loopreport+N` is specified at runtime, VCS checks if the simulation event loops for 'N' times instead of 2,000,000. VCS then issues a runtime warning message, and terminates the simulation.

For information about using the `+vcs+loopreport+number` option during runtime, see Section [Enabling Loop Detect](#) in Chapter "Simulation Options".

`+vcs+loopdetect+number`

When `+vcs+loopdetect+number` is specified at compile time, VCS does the following based on the option specified at runtime:

- If `number` is not specified at runtime, VCS checks if the simulation event loops for 2,000,000 times (by default) in the same simulation time tick, and issues a runtime error message. VCS also terminates the simulation.
- If `+vcs+loopdetect+N` is specified at runtime, VCS checks if the simulation event loops for 'N' times instead of 2,000,000. VCS then issues a runtime error message, and terminates the simulation.

For information about using the `+vcs+loopdetect+number` option during runtime, see Section [Enabling Loop Detect](#) in Chapter "Simulation Options".

Changing the Time Slot of Sequential UDP Output Evaluation

`-nonbaudpsched`

By default, VCS evaluates the output terminals of the sequential UDP (user-defined primitive) in the NBA region. If the design is compiled with this switch, the output of sequential UDPs is scheduled in the active region of the scheduler.

Gate-Level Performance

`-hsopt=gates`

Improves runtime performance on gate-level designs (both functional and timing simulations with SDF). You may see some compile-time degradation when you use this switch.

Limitation

The following is the limitation of the `-hsopt=gates` option:

- Line debug on combo gates is not supported.

Option to Omit Compilation of Code Between Pragmas

`-skip_translate_body`

Tells VCS to omit compilation of Verilog/SystemVerilog/VHDL code between the following:

`the //synopsys translate_off or /* synopsys translate_off */ pragma`

and

`the //synopsys translate_on or /* synopsys translate_on */ pragma`

Example of SystemVerilog Code with Translate off

The following SystemVerilog code example shows what this option can do:

```
module test;
initial begin
$display("\n before translate_off");
//synopsys translate_off
$display("\n after translate_off before translate_on");
//synopsys translate_on
$display("\n after translate_on before translate_off");
//synopsys translate_off
$display("\n 2nd after translate_off before translate_on");
//synopsys translate_on
$display("\n after translate_on\n");
end
endmodule
```

Without the `-skip_translate_body` option, VCS displays the following:

```
before translate_off

after translate_off before translate_on

after translate_on before translate_off

2nd after translate_off before translate_on

after translate_on
```

VCS compiles and executes all the `$display` system tasks.

With the `-skip_translate_body` option, VCS displays the following:

```
before translate_off

after translate_on before translate_off

after translate_on
```

VCS does not compile and execute the \$display system tasks between the //synopsys translate_off and //synopsys translate_on pragmas.

Example of VHDL code with Translate off

```
entity E1 is
  end entity E1;

architecture A1 of E1 is
begin
  assert false report "before translate off" severity note;
-- synopsys translate_off
  assert false report "after translate off before translate on"
severity note;
-- synopsys translate_on
  assert false report "after translate on before 2nd time
translate off" severity note;
-- synopsys translate_off
  assert false report "after translate off before translate on"
severity note;
-- synopsys translate_on
  assert false report "after 2nd time translate on" severity note;
end architecture A1;
```

Command Line

```
% vhdlan E1.vhdl -skip_translate_body
% vcs E1 -R
```

With the -skip_translate_body option, VCS displays the following message:

```
Assertion NOTE at 0 NS in design unit E1(A1) from
process /\\test/E1_inst\\_P0:
  "before translate off"
Assertion NOTE at 0 NS in design unit E1(A1) from
process /\\test/E1_inst\\_P1:
  "after translate on before 2nd time translate off"
Assertion NOTE at 0 NS in design unit E1(A1) from
process /\\test/E1_inst\\_P2:
  "after 2nd time translate on"
```

Known Issue

The /* pragma translate_off */ and /*pragma translate_on */ directives do not work with the -skip_translate_body option.

Generating a List of Source Files

```
-bom top-level_module -bfl filename
```

Generates a file that contains a list of absolute path names to the source files of all the module definitions in a design or IP block.

The `-bom` option must be accompanied by the `-bfl` option.

The argument to the `-bom` option is the module name of the top-level module in the design or IP block.

The argument to the `-bfl` option is the filename that contains the list. VCS adds the `.bfl` extension to the filename you specify.

If a module definition is in a Verilog source file in a Verilog library directory, the name of the directory and source file is included in the path names. If a module definition is in a Verilog library file, the pathname of the library file is included in the list.

The following is an example of the output pathname file:

```
/file_system/design_group/LIBDIR/dev.v
/file_system/user_name/design1/top.v
/file_system/design_group/libfile
```

Passing Options Starting With `"-"` to VCS in `-R` Flow

`+simargs`

Enables you to pass options starting with `"-"` to the VCS command line in the `-R` flow.

For example:

If you want to use VCS for invoking the Verdi interactive session from the compile command line, you can pass the `+simargs+-gui=verdi -R` option to the VCS command line as follows:

```
% vcs test.v -kdb -debug_access+all -lca +simargs+-gui=verdi -R
```

This command compiles the design and opens the Verdi interactive session in which you can perform various activities, such as running the interactive simulation, dumping FSDB, debugging using waveforms, and so on.

Option for Dumping Environment Variables

`-diag env`

Enables you to dump all environment variables that are set before starting the compilation and the simulation process. The list of environment variables that are set in the terminal is stored in the log file, which can be used to debug the environment related issues when the verification setup is complex and multiple and when nested scripts are used.

To dump all the environment variables, use the `-diag env` option with `vlogan/vcs` command line or `simv` command line.

Syntax

The following is the syntax for `-diag env` option:

```
% vlogan -diag env
```

Dumps all the environment variables in the `vlogan_env_diag_<pid>.log` log file that is generated in the `AN.DB` directory.

```
% vcs -diag env
```

Dumps all the environment variables in the `vcs_env_diag_<pid>.log` log file that is generated in the `simv.daidir` directory.

```
% simv -diag env
```

Dumps all the environment variables in the `simv_env_diag_<pid>.log` log file that is generated in the current working directory.

D

Simulation Options

This appendix describes the options and syntax associated with the `simv` executable. These runtime options are typically entered in the `simv` command line, however, some of them can be compiled into the `simv` executable at compile time.

This appendix describes the following runtime options:

- [Options for Simulating Native Testbenches](#)
- [Options for SystemVerilog Assertions](#)
- [Options to Control Termination of Simulation](#)
- [Options for Enabling and Disabling Specify Blocks](#)
- [Options for Specifying When Simulation Stops](#)
- [Options for Recording Output](#)
- [Options for Controlling Messages](#)
- [Options for VPD Files](#)
- [Options for VCD Files](#)
- [Options for Specifying Delays](#)
- [Options for Flushing Certain Output Text File Buffers](#)
- [Options for Licensing](#)
- [Option to Specify User-Defined Runtime Options in a File](#)
- [Option for the Support of Reading Gzipped Files](#)
- [Option for Initializing Verilog Variables, Registers and Memories at Runtime](#)
- [Option for Initializing Verilog Variables, Registers and Memories in Selective Parts of a Design at Runtime](#)
- [Options for Initializing Verilog Registers, Memories, and Sequential UDPs at Non-Zero Simulation Time](#)
- [General Options](#)

Options for Simulating Native Testbenches

`-cg_coverage_control`

Enables or disables the coverage data collection for all the coverage groups in your NTB-OV or SystemVerilog testbench.

The `$cg_coverage_control` system task takes precedence over this option.

Syntax: `-cg_coverage_control=value`

The valid values for `-cg_coverage_control` are 0 and 1. A value of 0 disables coverage collection and a value of 1 enables coverage collection.

You can also use this runtime option with the `coverage_control()` system task. The `coverage_control()` system task enables or disables data collection for one or more coverage groups at the program level. The runtime option takes precedence over the system task. For more information on this system task, see the *OpenVera Language Reference Manual: Native Testbench*.

`+ntb_cache_dir`

Specifies the directory location of the cache that VCS maintains as an internal disk cache for randomization.

`+ntb_delete_disk_cache=value`

Specifies whether VCS deletes the disk cache for randomization before simulation. The valid values are:

0 - do not delete

1 - delete the disk cache (the default condition)

`+ntb_disable_cnst_null_object_warning[=value]`

VCS produces the following warning when a null object handle is encountered in an object being randomized. Allowed values are 0 and 1.

0 - Do not disable null object warning (this is the default)

1 - Disable null object warning

Following is an example for the null object warning:

```
Warning-[CNST-PPRW] Constraint randomize NULL object warning test.sv,
<line number>. Null object found during randomization. Please make sure
all random variables/arrays/function calls being randomized are allocated
fully and properly.
```

The null handle might be intentional or the result of an oversight. If you want to randomize objects that contain null handles, you can use this switch to disable the runtime warning.

`+ntb_enable_checker_trace=0|1`

In-line constraint checker using `randomize(null)` returns 1 if all constraints are satisfied and 0 otherwise. This option controls whether the constraint checker trace is enabled or not. The valid arguments are as follows:

0	Does not display the constraint checker trace (default)
1	Displays the constraint checker trace

`+ntb_enable_checker_trace_on_failure[=value]`

Enables a mode that displays trace information only when the randomize returns 0. Allowed values are 0, 1, and 2.

0	Disables tracing
1	Enables tracing
2	Enables more verbose message in trace
3	In addition to the message in trace with option 2, the checker reports all the earlier solved constraints, which could have lead to the current failing constraint.

If `ntb_enable_checker_trace_on_failure` is specified without an argument, the default value is 1. If the `ntb_enable_checker_trace_on_failure` is not specified on the command line, then the default value is 0.

`+ntb_enable_solver_diagnostics [=1|2]`

Allows the solver to provide additional information about where the solver may be spending more time solving. This is commonly used in debugging performance issues in extracted testcases from a solver timeout or from `+ntb_solver_debug=extract`. The valid argument values are as follows:

1	Helps you to understand the variables that cause most of the slowdown or the timeout behavior
2	Helps you to understand the operators or constraint in the given problem definition that may result in a slowdown or timeout

Appendix D: Simulation Options

Options for Simulating Native Testbenches

`+ntb_enable_solver_trace[=0|1|2]`

Displays trace information for the VCS constraint solver. The valid argument values are as follows:

-
- | | |
|---|----------------------------------------|
| 0 | Disables displaying trace information |
| 1 | Enables displaying trace information |
| 2 | Enables more verbose trace information |
-

If `+ntb_enable_solver_trace` is specified without an argument, the default value is 1. If it is not specified, the default value is 0.

`+ntb_enable_solver_trace_on_failure[=0|1|2|3]`

Displays trace information when the VCS constraint solver fails to compute a solution. The valid argument values are as follows:

-
- | | |
|---|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 0 | Disables displaying trace information |
| 1 | Enables displaying trace information |
| 2 | Enables more verbose trace information |
| 3 | In addition to the more verbose trace information specified with 2, the solver reports all the earlier solved constraints, which could have lead to the current failing constraint. |
-

`+ntb_exit_on_error[=value]`

Causes VCS to exit when the value is less than 0. The value can be:

- 0 - continue
- 1 - exit on first error (default value)
- N - exit on nth error

When the value is 0, the simulation finishes regardless of the number of errors.

`+ntb_random_seed=value`

Sets the seed value to be used by the top-level random number generator at the start of simulation. The `srandom(seed)` system function call overrides this setting. The value can be any integer. The default random seed value is 1.

`+ntb_random_seed_automatic`

Picks a unique value to supply as the first seed used by a testbench. The value is determined by combining the time of day, host name and process id. This ensures that no two simulations have the same starting seed.

The `+ntb_random_seed_automatic` seed appears in both the simulation log and the coverage report. When you enter both `+ntb_random_seed_automatic` and `+ntb_random_seed` VCS displays a warning message and uses the `+ntb_random_seed` value.

`+ntb_random_reseed`

Enables the re-seeding of the value the top-level random number generator uses after a save and restore of the simulation.

You enter this option with the `+ntb_random_seed_automatic` or `+ntb_random_seed=value` options. The seed value after the restore is the same as the one specified or generated by these other options.

If you omit these other options, VCS ignores the `+ntb_random_reseed` option and displays the following informational message:

```
Info-[RNG-SEED-MISSING] New seed was not specified for reseeding.  
Please use runtime option +ntb_random_seed= or +ntb_random_automatic to  
specify new seed.
```

The `srandom(seed)` system function overrides this re-seeding.

`+ntb_solver_array_size_warn=value`

Specifies the array size warning limit (default is 10000) for constrained array sizes.

`+ntb_solver_cpu_limit`

Specifies the CPU limit (in seconds, default is 1000) for each partition before issuing a solver timeout.

`+ntb_solver_debug=keyword_argument`

Tells VCS to give you more information so you can debug the constraints for the `randomize()` calls in batch mode. The keyword arguments are as follows:

`extract`

Tells VCS to extract a standalone test case in SystemVerilog for the specified `randomize()` call(s). To use this keyword argument, also enter the `+ntb_solver_debug_filter` runtime option.

`profile`

Enables constraint profiling in VCS . You can view the constraint profile report in `simv.cst/html/profile.xml` using a web browser (`simv` is the default name of the VCS `simv` executable).

This keyword argument also writes a file with a listing of the top randomize calls in `simv.cst/serial2trace.txt` (`simv` is the default name of the VCS `simv` executable).

`serial`

Displays the randomize serial number at the end of each `randomize()` completion.

`trace`

Displays the solver trace to show how VCS solved the constraints for the random variables in specified `randomize()` call(s). To use this argument, also enter the `+ntb_solver_debug_filter runtime` option.

`trace_all`

Displays the solver trace for all `randomize()` calls. The `+ntb_solver_debug=trace_all` option is the equivalent of entering the following options and arguments together:

`+ntb_solver_debug=trace +ntb_solver_debug_filter=all`

You can enter multiple the keyword arguments using a plus (+) as a delimiter. For example:

```
vcs source.sv +ntb_solver_debug=serial+extract+profile \
+ntb_solver_debug_filter=12
```

However, you cannot enter multiple `+ntb_solver_debug` options.

`+ntb_solver_debug_dir=pathname`

Directs VCS to place profiles and extracted testcases in the specified directory. The default directory name is `simv.cst`, after the `simv` executable with the `.cst` extension.

```
+ntb_solver_debug_filter= serial_num [.partition_num] | file[:filename] | all
```

Specifies a list of `randomize()` calls that VCS displays debug information about. You can specify this list in the following ways:

- A comma separated list, for example:

```
+ntb_solver_debug_filter=1,5,4,20
```

This example specifies: the 5th partition of 1st call, and all partitions of the 4th and 20th call.

- In a file. The default filename is: `simv.cst/serial2trace.txt`. You need to enter the keyword argument file if the file is the default file name and location.
- The keyword `all` as in: `+ntb_solver_debug_filter=all`

Specifying `all` means you want debug information about all `randomize()` calls.

Note:

The `all` argument can result in a large amount of solver trace information or extracted test cases.

`+ntb_solver_mode=value`

Allows you to choose between one of two constraint solver modes. When set to 1, the solver spends more preprocessing time in analyzing the constraints during the first call to `randomize()` on each class. Therefore, subsequent calls to `randomize()` on that class are very fast. When set to 2, the solver does minimal preprocessing, and analyzes the constraint in each call to `randomize()`. The default mode is 3.

In the default mode, based on the test case, the solver picks up one of the above solver mode. In case of any difficulty to solve with the initially chosen solver mode, it tries to switch to the other mode automatically.

`+ntb_stop_on_constraint_solver_error=0|1`

Specifies whether VCS continues or exits after a constraint solver failure due to constraint inconsistency.

0 VCS to continues to run after a constraint solver failure (default).

1 VCS exits on the first constraint solver error

`+gc+high_threshold[+1|2|3|4|5|...]`

Sets a higher memory threshold for the purpose of initiating garbage collection (GC). It improves runtime performance by reducing the frequency of GC invocations, at the cost of additional memory consumption. Each increment in specified level raises GC threshold memory by about 25% of total live memory of all dynamic objects. When level value is not specified, it defaults to 2. The unified simulation profiler time and memory report shows the frequency of GC invocations and its cost in terms of time and memory overheads.

Options for SystemVerilog Assertions

`-assert keyword_argument`

Note:

- All the `-assert keyword_argument` runtime options, except the `-assert maxfail` and `-assert finish_maxfail` options are enabled only when the `-assert enable_diag` option is used at compile time.
- To enable the `-assert maxfail` and `-assert finish_maxfail` options at runtime, you must use the `-assert enable_hier` option at compile time.

The keyword arguments are as follows:

`dbgopt`

Enables the assertion optimization in the presence of higher debug caps. This compile time option is useful in the presence of `-debug_access` option and helps in generating optimal code for runtime performance. All the optimizations that are not part of the full debug flow are enabled under this switch. When this option is used at compile time, success callback is disabled and only failures are dumped. This option improves performance when the design is compiled with `-debug_access+f` or CLI level greater than 2 or `-debug_access+all`. You can enable success dumping by giving the switch `-assert dumpsuccess` at runtime, which can potentially degrade runtime performance.

`dbsopt=N`

Creates groups of `N` assertions with non-temporal expressions under the same clock, thereby localizing the evaluation of such assertions due to changes in a smaller set of signals.

`disable=<value>`

Disables the concurrent, immediate, and deferred assertions in a particular design. The options for `<value>` can be concurrent, deferred, and immediate.

`dumpoff`

Disables the dumping of SVA information in the VPD file during simulation.

`finish_maxfail=N`

Terminates the simulation if the number of failures for any assertion reaches `N`. You must supply `N`, otherwise no limit is set.

`global_finish_maxfail=N`

Terminates the simulation when the total number of failures from all SystemVerilog assertions reaches `N`.

`maxcover=N`

Disables the collection of coverage information for cover statements after the cover statements are covered `N` number of times. `N` must be a positive integer; it cannot be 0.

Appendix D: Simulation Options

Options for SystemVerilog Assertions

`maxfail=N`

Limits the number of failures for each assertion to `N`. When the limit is reached, VCS disables the assertion. You must supply `N`, otherwise no limit is set.

`maxsuccess=N`

Limits the total number of reported successes to `N`. You must supply `N`, otherwise no limit is set. VCS continues to monitor assertions even after the limit is reached.

`nocovdb`

Tells VCS not to write the `program_name.db` database file for assertion coverage.

`noDefSuccRpt`

Improves the performance of deferred assertions during runtime. When the `-debug_access` option or higher debug capabilities are used, VCS tracks both successes and failures, hence results in reduced runtime performance. When this switch is used, VCS tracks only failures of deferred assertions and drops the successes. This improves the performance because successes happen more frequently. This option has no effect when the design is compiled without the `-debug_access` switch.

`nopostproc`

Disables the display of the SystemVerilog `assert` and `cover` statement summary at the end of simulation.

This begins with the `assert` and `cover` statements that started but did not finish in the following format:

```
"source_filename.v", line_number:  
assert_or_cover_statement_hierarchical_name: started at simulation_time  
not finished
```

If the `assert` or `cover` statement does not start, this summary also reports about this in the following format:

```
**** Following assertions did not fire at all during  
simulation. ***** "source_filename.v", line_number:  
assert_or_cover_statement_hierarchical_name: No attempt started
```

This is followed by a `cover` statement summary in the following format:

```
"source_filename.v", line_number: cover_statement_hierarchical_name,  
number attempts, number match  
  
no_fatal_action
```

Excludes failures on SVA assertions with fail action blocks for computation of failure count in the `-assert [global_]finish_maxfail=N` runtime option.

`no_default_msg [=SVA | OVA | PSL]`

Disables the display of default failure messages for SVA assertions that contain a fail action block, and OVA and PSL assertions that contain user messages.

`novpi`

Disables all vpi based callback access for assertions and dumping of assertion. This compile time option is useful in the presence of `-debug_access` option. It helps to reduce assertion debug overhead, when the design is compiled with higher debug capabilities.

Note: If a combination of `-assert dbgopt+novpi` is given, you can have the access to perform the following:

Ability to iterate over all the assertions.

Mark the assertions for dynamically enabling or disabling them.

`quiet`

Disables the display of messages when assertions fail.

`quiet1`

Disables the display of messages when assertions fail, however, enables the display of summary information at the end of simulation. For example,

`Summary: 2 assertions, 2 with attempts, 2 with failures`

`report [=path/filename]`

- Generates a report file in addition to printing results on your screen. By default, the report file name and location is `./assert.report`, however, you can change it by entering the `path/filename` argument. The report file name can start with a number or letter.
- Generates a report of all assertions that are disabled using any one of the following mechanisms:

- - System tasks `$asserton/off/kill`
 - `assert hier` at compile time or runtime

The report is categorized based on:

- Disabled assertions on a module level (compile time)
- Assertions disabled through the `-assert hier` option
- Disabled assertions at the end-of-simulation

Note:

- If the file name is specified by the user, it is dumped as `<user_file>.disablelog`.
- If the file name is not specified by the user, it is dumped as `assert.report.disablelog`

The following special characters are acceptable in the file name: %, ^, and @. Using the following unacceptable special characters: #, &, *, [], \$, (), or ! has the following consequences:

- A file name containing # or & results in a file name truncation to the character before the # or &.
- A file name containing * or [] results in a `No match` message.
- A file name containing \$ results in an `Undefined variable` message.
- A file name containing () results in a `Badly placed ()'s` message.
- A file name containing ! results in an `Event not found` message.

`success`

Enables reporting of successful matches, and successes on `cover` and `assert` statements respectively, in addition to failures. The default is to report only failures.

`vacuous`

Enables reporting of vacuous successes on `assert` statements in addition to the failures. By default, VCS reports only failures.

`verbose`

Adds more information to the end of the report specified by the `report` keyword argument, and a summary with the number of assertions present, attempted, and failed.

`hier=file_name`

Specifies a file to enable and disable SystemVerilog assertions when you simulate your design. This feature enables you to control which assertions are active and VCS records in the coverage database, without having to recompile your design.

The types of entries you can make in the file are as follows:

`-assert <assertion_name> or -assert <assertion_hierarchical_name>`

If `<assertion_name>` is provided, VCS disables the assertions based on wildcard matching of the name in the complete design. If `<assertion_hierarchical_name>` is provided, VCS disables the assertions based on wildcard matching of the name in the particular hierarchy given.

Examples

```
-assert my_assert
```

Disables all assertions with name `my_assert` in the full design.

```
-assert A*
```

Disables all assertions whose name starts with `A` in the full design.

```
-assert *
```

Disables all assertions in the full design.

```
-assert top.INST2.A
```

Disables all assertions whose names start with `A` in the `top.INST2` hierarchy. If assertions whose name starts with `A` exists in inner scopes under `top.INST2`, they are not disabled. This command has affect on assertions only in scope `top.INST2`.

```
+assert <assertion_name> or +assert <assertion_hierarchical_name>
```

If `<assertion_name>` is provided, VCS enables the assertions based on wildcard matching of the name in the full design. If `<assertion_hierarchical_name>` is provided, then VCS enables the assertions based on wildcard matching of the name in the given hierarchy.

Examples

```
+assert my_assert
```

Enables all assertions with name `my_assert` in the full design.

```
+assert A*
```

Enables all assertions whose name starts with `A` in the full design.

```
+assert *
```

Enables all assertions in the full design.

```
+assert top.INST2.A
```

Enables assertion `A` in the hierarchy `top.INST2`.

```
+tree <module_instance_name> or +tree <assertion_hierarchical_name>
```

If `<module_instance_name>` is provided, VCS enables assertions in the specified module instance and all module instances hierarchically under that instance. If `<assertion_hierarchical_name>` is provided, VCS enables the specified SystemVerilog assertion. Wildcard characters can also be used for specifying the hierarchy.

Examples

```
+tree top.inst1
```

Enables the assertions in module instance `top.inst1` and all the assertions in the module instances under this instance.

```
+tree top.inst1.a1
```

Enables SystemVerilog assertion with the hierarchical name `top.inst1.a1`.

```
+tree top.INST*.A1
```

Enables assertion `A1` from all the instances whose names start with `INST` under module `top`.

```
-tree <module_instance_name> or -tree <assertion_hierarchical_name>
```

If `<module_instance_name>` is provided, VCS disables the assertions in the specified module instance and all module instances hierarchically under that instance. If `<assertion_hierarchical_name>` is provided, VCS disables the specified SystemVerilog assertion. Wildcard characters can also be used for specifying the hierarchy.

Examples

```
-tree top.inst1
```

Disables the assertions in module instance `top.inst1` and all the assertions in the module instances under this instance.

```
-tree top.inst1.a1
```

Disables the SystemVerilog assertion with the hierarchical name `top.inst1.a1`.

```
-tree top.INST*.A1
```

Disables assertion `A1` from all the instances whose names start with `INST` under module `top`.

The wildcard (*) replaces only one design element in the hierarchy path.

For example, `-tree top.INST*.A1` option disables assertion `A1` under `top.INST.blk1.A1` and `top.INST.blk2.A1`. This option does not disable assertion `A1` under `top.INST.blk1.blk2.A1`.

```
+module module_identifier
```

VCS enables all the assertions in all instances of the specified module.

For example, `+module dev`. VCS enables the assertions in all instances of module `dev`.

```
-module module_identifier
```

VCS disables all the assertions in all instances of the specified module.

For example, `-module dev`. VCS disables the assertions in all instances of module `dev`.

```
-assert assertion_block_identifier
```

VCS disables the assertion with the specified block identifier. You can use wildcard characters in specifying the block identifier to specify more than one assertion.

You can enter more than one keyword using the plus (+) separator. For example, `-assert maxfail=10+maxsuccess=20+success+filter`.

```
-cm assert
```

Specifies monitoring for SystemVerilog assertions coverage. When enabled, the `-cm assert` option does the following:

- Generates the number of attempts, pass, fail, and incomplete data.
- Generates vacuous and non-vacuous coverage.
- Irrespective of type of assert statement, reports coverage.
- Covers immediate and deferred assertions.
- Does not cover Expect statement.
- Affects SVA and OVA as well.

```
-uniqu_prior maxfail=integer
```

Specifies the maximum number of unique or priority violations (see `-error=UNIQUE` and `-error=PRIORITY` in [Options for SystemVerilog Assertions](#)) before VCS ends the simulation.

The types of error messages that this option controls are as follows:

```
RT Error: No condition matches in unique case statement
"dev.v", line 17, for top.dev, at time 0
```

```
RT Error: More than one conditions match in 'unique case' statement
"dev.v", line 18, for top.dev,
Line 19 & 20 are overlapping at time 0.
```

This runtime option is enabled by the `-error=UNIQUE`, `-error=PRIORITY`, or `-error=UNIQUE,PRIORITY` compile time option and keyword arguments.

Options to Control Termination of Simulation

```
-ova_enable_case_maxfail
```

Includes OVA case violations in computation of global failure count for the `-assert global_finish_maxfail=N` option.

Options for Enabling and Disabling Specify Blocks

`+no_notifier`

Suppresses the toggling of notifier registers that are optional arguments of system timing checks. The reporting of timing check violations is not affected. This is also a compile time option.

`+no_pulse_msg`

Suppresses pulse error messages, however, not the generation of `STE` values at module path outputs when a pulse error condition occurs.

`+no_tchk_msg`

Disables the display of timing violations, however, does not disable the toggling of notifier registers in timing checks. This is also a compile-time option.

`+notimingcheck`

Disables timing check system tasks in your design. Using this option at runtime can improve the simulation performance of your design, depending on the number of timing checks that this option disables.

You can also use this option at compile time. Using this option at compile time tells VCS to ignore timing checks when it compiles your design so that the timing checks are not compiled into the executable. This results in a faster simulating executable than one that includes timing checks, which are disabled by this option at runtime.

If you need the delayed versions of the signals in negative timing checks, but want faster performance, include this option at runtime.

Note:

The `+notimingcheck` option has higher precedence than any `tcheck` command in UCLI.

Options for Specifying When Simulation Stops

`+vcs+stop+time`

Stop simulation at the `time` value specified. The `time` value must be less than 232 or 4,294,967,296.

`+vcs+finish+time`

Ends simulation at the `time` value specified. The `time` value must be also less than 232. For example, you can specify the following:

```
+vcs+finish+9001us
```

For both of these options, there is a special procedure (See [Specifying Long Time Before Stopping the Simulation](#)) for specifying time values larger than 232.

Options for Recording Output

`-l filename`

Specifies writing all messages from simulation to the specified file as well as displaying these messages on the standard output.

Options for Controlling Messages

`-error`

Revises the `+lint` and `+warn` options, to control error and warning messages. With them you can:

- Disable the display of any lint, warning, or error messages
- Disable the display of specific messages
- Limit the display of specific messages to a maximum number that you specify

Only the following feature is supported at runtime.

```
-error=[no]message_ID[:max_number],...
```

For more information on the option, see [Error/Warning/Lint Message Control](#).

The `-error` option is also a compile time option.

`-q`

Quiet mode; suppresses display of VCS header and summary information. Suppresses the proprietary message at the beginning of simulation and suppresses the VCS Simulation Report at the end (time, CPU time, data structure size, and date). Suppresses SystemC BMI warnings and notes at the start of simulation.

`-V`

Verbose mode; displays VCS version and extended summary information. Displays VCS compile and runtime version numbers, and copyright information, at the start of simulation.

```
+no_pulse_msg
```

Suppresses pulse error messages, however, not the generation of `STE` values at module path outputs when a pulse error condition occurs.

You can enter this runtime option in the `simv` command line.

```
+vcs+nostdout
```

Disables all text output from VCS including messages and text from `$monitor` and `$display` and other system tasks for only the Verilog portion of the design. VCS still writes this output to the log file if you include the `-l` option.

Options for VPD Files

```
-vpd_bufsize number_of_megabytes
```

To gain efficiency, VPD uses an internal buffer to store value changes before saving them on disk. This option modifies the size of that internal buffer. The minimum size allowed is what is required to share two value changes per signal. The default size is the size required to store 15 value changes for each signal, however, not less than 2 megabytes.

VCS automatically increases the buffer size as needed to comply with this limit.

```
-vpd_file <file_name>
```

Specifies the name of the output VPD file (default is `vcplus.vpd`). You must include the full file name with the `.vpd` extension.

```
+vpdfilysize+number_of_megabytes
```

Creates a VPD file that has a moving window in time while never exceeding the file size specified by `number_of_megabytes`. When the VPD file size limit is reached, VPD continues saving simulation history by overwriting older history.

File size is a direct result of circuit size, circuit activity, and the data being saved. Test cases show that VPD file sizes will likely run from a few megabytes to a few hundred megabytes. Many users can share the same VPD history file, which might be a reason for saving all time value changes when you do simulation. You can save one history file for a design and overwrite it on each subsequent run.

```
-vpd_fileswitchsize <size_in_MB>
```

Specifies a size for the vpd file. When the vpd file reaches this size, VCS closes this file and opens a new file with the same hierarchy as the previous vpd file. There is a number suffix added to all new vpd file names to differentiate them.

For example, `simv -vpd_file test.vpd -vpd_fileswitchsize 10`. The first vpd file is named `test.vpd`. When its size reaches 10MB, VCS starts a new file `test_01.vpd`, the third vpd file is `test_02.vpd`, and so on.

+vpdignore

Tells VCS to ignore any `$vcdplusxx` system tasks and license checking. By default, VCS checks out a VPD PLI license if there is a `$vcdplusxx` system task in the Verilog source. In some cases, this statement is never executed and VPD PLI license checkout should be suppressed. The `+vpdignore` option performs the license suppression.

+vpdports

Causes VPD to store port information, which is then used by the Hierarchy Browser to show whether a signal is a port, and if so, its direction. This option to some extent affects simulation initialization time and memory usage for larger designs.

+vpdportsonly

Dumps only the port type information.

+vpdnoports

Dumps only the signal not the ports (input/output).

+vpddrivers

Stores data for changes on drivers of resolved nets.

+vpdupdate

Enables VPD file locking.

+vpdnocompress

Disables the default compression of data as it is written to the VPD file.

Options for VCD Files

+vcs+dumpfile+filename

Sets the name of the `$dumpvars` output file to `filename`. The default file name is `verilog.dump`. A `$dumpfile` system task in the Verilog source code overrides this option.

+vcs+dumpoff+t+ht

Turns off value change dumping (`$dumpvars`) at time `t`. `ht` is the high 32 bits of a time value greater than 32 bits.

+vcs+dumpon+t+ht

Suppresses the `$dumpvars` system task until time `t`. `ht` is the high 32 bits of a time value greater than 32 bits.

+vcs+dumparrays

Enables recording memory and multi-dimensional array values in the VCD file.

`+vcs+flush+dump`

Increases the frequency of dumping all VCD files.

Options for Specifying Delays

`-novitaltiming`

Enables functional-only simulation of VITAL components. All timing information is discarded for VITAL models during simulation. Timing information includes wire delays, path delays and timing checks. Any SDF information supplied on the command line is ignored when this switch is present.

`+maxdelays`

Specifies using the maximum delays in min:typ:max delay triplets in module path delays and timing checks, if you compiled your design with the `+allmtm` compile time option. Also specifies using the maximum timing delays in min:typ:max delay triplets in an uncompiled SDF file.

If you compiled the SDF file with the `+allmtm` compile-time option, the `+maxdelays` option specifies using the compiled SDF file with the maximum delays.

Another use for this runtime option is to specify timing for SWIFT VMC and SmartModels when you also include the `+override_model_delays` runtime option.

`+mindelays`

Specifies using the minimum delays in min:typ:max delay triplets in module path delays and timing checks, if you compiled your design with the `+allmtm` compile-time option. Also specifies using the minimum timing delays in min:typ:max delay triplets in an uncompiled SDF file.

If you compiled the SDF file with the `+allmtm` compile time option, the `+mindelays` option specifies using the compiled SDF file with the minimum delay.

Another use for this runtime option is to specify timing for SWIFT VMC and SmartModels when you also include the `+override_model_delays` runtime option.

`+typdelays`

Specifies using the typical delays in min:typ:max delay triplets in module path delays and timing checks, if you compiled your design with the `+allmtm` compile-time option. Also specifies using the typical timing delays in min:typ:max delay triplets in an uncompiled SDF file.

If you compiled the SDF file with the `+allmtm` compile-time option, the `+typdelays` option specifies using the compiled SDF file with the typical delays.

This is a default option. By default, VCS uses the typical delay in min:typ:max delay triplets in your source code and in uncompiled SDF files unless you specify otherwise with the `mtm_spec` argument to the `$sdf_annotation` system task. Also, by default, VCS uses the compiled SDF file with typical values.

Another use for this runtime option is to specify timing for SWIFT VMC and SmartModels when you also include the `+override_model_delays` runtime option.

```
-xlrn sdf_annotation_ignore_blank
```

Enables VCS to annotate a delay value in the timing arc of the library even when the SDF IOPATH statement contains a blank entry. By default, VCS uses the value that actually exists in the SDF. If SDF values are blank, then the values in the specify block are applied.

Example

```
specify // 01 10 0z z1 1z z0
( SEL *> Z ) = ( 0.001, 0.001, 0.001, 0.001, 0.001, 0.001 )
endspecify

[SDF]
(IOPATH SEL Z ) () () (0.3) (0.4) (0.5) (0.6) )
```

By default, the pin-to-pin delays from SEL to Z after SDF annotation are as follows:

0->1 0.001 from specify

1->0 0.001 from specify

0->z 0.3 SDF

z->1 0.4 SDF

1->z 0.5 SDF

z->0 0.6 SDF

With the `-xlrn sdf_annotation_ignore_blank` option, the delay value can be minimum, maximum, average, or specific, as shown in the following examples:

- **Minimum:** `-xlrn sdf_annotation_ignore_blank+min: min (0.3, 0.4, 0.5, 0.6) = 0.3`
- **Maximum:** `-xlrn sdf_annotation_ignore_blank+max: max (0.3, 0.4, 0.5, 0.6) = 0.6`
- **Average:** `-xlrn sdf_annotation_ignore_blank+ave: ave (0.3, 0.4, 0.5, 0.6) = 0.45`
- **Specific Value:** `-xlrn sdf_annotation_ignore_blank+<value>: -xlrn
sdf_annotation_ignore_blank+0.33 : 0.33.`

Note:

- Under this mode, the minimum, maximum, average, or specific delays are used for blanks regardless of the polarity of the delays (01, 10, 0z, z1, 1z, z0, and so on).
- Under this mode, min:typ:max delay triplets are distinguished.
- In case of conditional IOPATH, blanks are replaced with the specific minimum, maximum, or average value of all conditions of the same path.

Options for Flushing Certain Output Text File Buffers

When VCS creates a log file, VCD file, or a text file specified with the `$fopen` system function. VCS writes the data for the file in a buffer and periodically dumps the data from the buffer to the file on disk. The frequency of these dumps varies depending on many factors including the amount of data that VCS has to write to the buffer as simulation or compilation progresses. If you need to see or use the latest information in these files more frequently than the rate at which VCS normally dumps this data, these options tell VCS to dump the data more frequently. The frequency also depends on many factors; however the increased frequency will always be significant.

`+vcs+flush+log`

Increases the frequency of dumping both the compilation and simulation log files.

`+vcs+flush+dump`

Increases the frequency of dumping all VCD files.

`+vcs+flush+fopen`

Increases the frequency of dumping all files opened by the `$fopen` system function.

`+vcs+flush+all`

Increases the frequency of dumping all log files, VCD files, and all files opened by the `$fopen` system function.

These options do not increase the frequency of dumping other text files including the VCDE files specified by the `$dumpports` system task.

You can also enter these options at compile time. There is no performance gain to entering them at compile time.

Options for Licensing

`+vcs+lic+vcsi`

Checks out three VCSI licenses to run VCS.

```
+vcsi+lic+vcs
```

Checks out a VCS license to run VCSI when all VCSI licenses are in use.

```
+vcst+lic+wait
```

Waits for a network license if none is available when the job starts.

```
-licwait timeout
```

Enables license queuing, where `timeout` is the time in minutes that VCS waits for a license before finally exiting.

```
-licqueue
```

Tells VCS to wait for a network license if none is available.

Option to Specify User-Defined Runtime Options in a File

```
-f filename
```

You can use the `-f` runtime option to specify user-defined `plusargs` in a file. The user-defined `plusargs` are the plus arguments on the `simv` command line defined using `$test$plusargs` or `$value$plusargs` system tasks in RTL code as per *IEEE Standard 1364-2001 17.10 Command line input*. All other VCS runtime options should be specified on the `simv` command line.

Option for the Support of Reading Gzipped Files

VCS supports an option to read files that are compressed using the gzip.

To enable this option, use the `-io_gz` runtime switch.

Example

This example shows how the utility is used to read a gzipped file.

Example 258 Test.v

```
module m1;

    integer mcd,code,code1,mcd1;
    string str;

initial
begin
```

Appendix D: Simulation Options

Option for the Support of Reading Gzipped Files

```

mcd = $fopen("test_file.txt", "w");
$fwrite(mcd, "%s", "\nThis is VCS test file\n");
fclose(mcd);

$system("gzip test_file.txt");
mcd = $fopen("test_file.txt.gz", "r");
do
begin
    code = $fgets(str, mcd);
    $display("%s", str);
end
while($feof(mcd)==0);

fclose(mcd);

end
endmodule

```

You can use the following commands to run the test case.

```
vcs -sverilog test.v
simv -io_gz
```

Output

This is VCS test file

Supported APIs

Following are the APIs that are supported:

- \$fopen
- \$fgets
- \$fgetc
- \$fread
- \$fseek
- \$ftell
- \$rewind
- \$error
- \$feof
- \$fclose

- \$freadb
- \$freadh

Limitations

- You cannot perform a write operation on gzipped files.
- The `$fscanf` API is not supported.

Option for Initializing Verilog Variables, Registers and Memories at Runtime

`+vcs+initreg+0|1|random|seed_value`

Initializes all bits of the Verilog variables, registers defined in sequential UDPs, and memories including multi-dimensional arrays (MDAs) in your design to the specified values at time zero. The default seed is used when no random seed is specified. This option can only be used when the `+vcs+initreg+random` option is specified at compile time.

The supported data types are:

- reg
- bit
- integer
- int
- logic

The following table describes the initialization options at runtime:

Syntax of Runtime Option	Description
<code>+vcs+initreg+0</code>	Initializes all variables, registers and memories to value 0.
<code>+vcs+initreg+1</code>	Initializes all variables, registers and memories to value 1.
<code>+vcs+initreg+random</code>	Initializes all variables, registers and memories to random value 0 or 1, with the default seed.
<code>+vcs+initreg+100</code>	Initializes all variables, registers and memories to random value 0 or 1, with the user-defined seed 100. Note: The <code>seed_value</code> cannot be 1 or 0. Those values have special meanings.

The initialization options might cause potential race conditions due to the initialized values specified. For more information on race condition prevention, see [Option for Initializing Verilog Variables, Registers and Memories with Random Values](#).

Option for Initializing Verilog Variables, Registers and Memories in Selective Parts of a Design at Runtime

`+vcs+initreg+config+config_file`

Specifies a configuration file for initializing Verilog variables, registers defined in sequential UDPs, and memories including multi-dimensional arrays (MDAs) in your design at time zero. In the configuration file, you can define the parts of a design to apply the initialization and the initialization values of the variables.

This option can only be used at runtime when either the `+vcs+initreg+random` option or the `+vcs+initreg+config+config_file` option is specified at compile-time.

If the `+vcs+initreg+config+config_file` option is specified at both compile time and runtime, the configuration file specified at runtime overrides the configuration file at compile time.

If the `+vcs+initreg+config+config_file` option is specified at compile time and the `+ntb_random_seed=0|1` option is specified at runtime, the configuration file specified at compile time is overridden and the entire design is initialized.

The `+vcs+initreg+seed_value` option can be specified with the `+vcs+initreg+config+config_file` option at runtime to select a random seed for generating random initial values as defined in the configuration file.

If the `+vcs+initreg+0|1|random` and `+vcs+initreg+config+config_file` options are both specified at runtime, the `+vcs+initreg+0|1|random` option is ignored and a warning message is issued.

The following table describes the initialization options at runtime:

Syntax of Runtime Options	Description
<code>+vcs+initreg+config+config_file</code>	Specifies runtime configuration file <code>config_file</code> and overrides compile-time configuration file, if specified.
<code>+vcs+initreg+config+config_file+vcs+initreg+seed_value</code>	Uses specified seed for generating random initial values as defined in runtime configuration file.
<code>+vcs+initreg+config+config_file+vcs+initreg+random</code>	Issues a warning message, ignores <code>+vcs+initreg+random</code>

Syntax of Runtime Options	Description
+vcst+initreg+config+ <i>config_file</i> +vcst+init reg+0	Issues a warning message, ignores +vcst+initreg+0
+vcst+initreg+config+ <i>config_file</i> +vcst+init reg+1	Issues a warning message, ignores +vcst+initreg+1

For details on the configuration file entries, see [Option for Initializing Verilog Variables, Registers and Memories in Selective Parts of a Design](#).

Options for Initializing Verilog Registers, Memories, and Sequential UDPs at Non-Zero Simulation Time

The deferred initreg feature supports non-time zero (after time 0) initialization of Verilog registers, memories, and sequential UDPs. This feature initializes the output of sequential UDPs at the user-specified time and as per the user-specified configuration file.

Deferred initreg can be used for scenarios where the time 0 initialization is overridden and lost due to transitions on the UDP input signals. For example, when there is a transition from an X value to a stable value on a clock signal, and that transition is not covered by the UDP state table, the result is that the UDP output changes from the value deposited by initreg at time 0 to an X value. If you use the deferred initreg feature after the transition on the clock signal, the UDP is reinitialized.

Prerequisites for Deferred Initreg

To use this feature, specify the `+vcst+initreg+random` option at compile time.

Use Model of Deferred Initreg

You can use the deferred initreg feature using one of the following use models:

- [System Task](#)
- [Command Line](#)
- [Unified Command-Line Interface \(UCLI\)](#)

You can use the system task use model if you have planned the use of initreg for your simulation in advance. Adding the system task in your Verilog or SystemVerilog code triggers initreg initialization of the design based on the input arguments of the system task when it is invoked.

However, if you want to invoke initreg after the design creation phase, you can use the command-line or the Unified Command-Line Interface (UCLI) use model.

Note the following for the three use models:

- When you use the system task model, you cannot use any other use model. VCS generates an error message when you try to use the command line or the UCLI use model along with the system task use model.
- When you use the command line and the UCLI use models at the same time, the command line use model is preferred during simulation.
- When you want to invoke multiple deferred initreg initializations, use the UCLI use model. Multiple deferred initreg initializations are not supported in the command-line use model.
- Deferred initreg still happens at time 0 in addition to the initreg UCLI calls done for the UCLI script after time 0.
- Deferred initreg after time 0 does not change the elements that already have 0 or 1 values; It affects only the elements with X values.
- In case of a race between other design drivers and initreg, design drivers take precedence for the race condition unless X value is specified for the signal.

System Task

The syntax of the `$deferred_initreg` system task is as follows:

```
$deferred_initreg("module"|"tree"|"modtree"|"instance",
  <"module_or_hierarchy_name">,
  <"initialization-value"> [, <"depth_level">])
```

Where,

`"module"|"tree"|"modtree"|"instance"`

Specifies the hierarchy level.

- `"module"`: Used to specify a module.
- `"tree"`: Used to specify the complete hierarchy tree.
- `"modtree"`: Used to specify all instances of a module and all instances that are hierarchically beneath those instances.
- `"instance"`: Used to specify the fully rooted path name to a Verilog instance.

`<"module_or_hierarchy_name">`

Specifies the name of the module or hierarchy.

Example: top.m1.m2.m3.b

Here, m1, m2, m3, and b are names of module instances.

<"initialization-value">

It can be "0", "1", "random", "random <random- seed>", or <"integer" | random seed for random initialization>.

[<"depth_level">]

(Optional) Specifies the depth till which the output of signal should be initialized. The depth level is required only with the tree and modtree initialization types.

Example

```
+$deferred_initreg("modtree", "bot", "1", "0")
```

This example initializes all signals including and below the bot module with value 1. Here, depth = 0 implies infinite depth.

Command Line

Execute the following runtime option:

```
% simv
+vcst+initreg+deferred_time+deferred_time_value
+vcst+initreg+deferred_config+deferred_config_name
```

Example:

```
% simv
+vcst+initreg+deferred_time+210ns
+vcst+initreg+deferred_config+init_210.cfg
```

In this example, the signal is initialized at 210 ns based on the deferred configuration init_210.cfg. However, if the deferred configuration is not specified, the time zero initreg configuration is used.

Note the following for the command-line use model:

- +vcst+initreg+config+init_0.cfg +vcst+initreg+deferred_time+210ns: Initializes the signal at 210 ns based on the time zero configuration init_0.cfg. You can pass multiple configuration files at different times.
- +vcst+initreg+1 +vcst+initreg+deferred_time+210ns: It initializes the signal to 1 at 210 ns.
- +vcst+initreg+deferred_time+210ns: It randomly initializes the signal at 210 ns.
- Deferred initreg does not stop or alter the time zero initreg.

Note: The time zero initreg happens when you execute the following options:

+vcs+initreg+random compile-time option

+vcs+initreg+config+<config_file> compile-time or runtime option

- You can specify both 32-bit and 64-bit deferred time using the +vcs+initreg +deferred_time runtime option.
- If no time unit is specified, deferred initreg assumes the deferred time to be the resolved time unit.

Unified Command-Line Interface (UCLI)

To use this feature, specify the +vcs+initreg+random option at compile time.

1. Execute the -debug_access+r compile-time option.
2. Execute the +vcs+initreg+deferred runtime option
3. Specify the deferred time and the deferred configuration in the .tcl file as follows:

```
run deferred_time_value
initreg deferred_config_name
run
```

Consider the following .tcl file.

```
run 210ns
initreg init_210.cfg
run
```

In this example, the signal is initialized at 210 ns based on the deferred configuration init_210.cfg.

With UCLI, reinitialization with time zero initreg configuration cannot be done. For this, the required configuration must be passed using the .tcl file. To do this, execute the following command:

```
initreg <config-file-path>
```

Example of Deferred Initreg

The following example shows how you can use the deferred initreg feature. Consider the following test case:

Example 259 tmp.v

```
module top;
  reg q1,q2,q3;
  logic clk,d;
```

```

logic [1:0]p;
mid m1(clk,d);
udp u1(q1,clk,d);

always_comb $display("time:%3t q1:%0b m1.q2:%0b q3:%0b p:%0b
m1.r:%0p",$time,q1,m1.q2,q3,p,m1.r);

initial begin
#20 clk=1;

p = 2'bxx;
#20 d=1;clk=0;
q3 = 1'bx;
#10 clk=1;
#30 clk=1'b0;d=1'b0;
end
endmodule
module mid(input clk,d);
reg q2;
reg r [2:0][1:0];
udp u2(q2,clk,d);
initial begin
#20 r = '{'{1'bx,1'bx},{1'bx,1},{1,1'bx}};
end
endmodule
primitive udp (q, clk, d);
    output q;
    input clk, d;
    reg q;
    table
        // clk d    q    q+
        1  1 : ? : 1;
        1  0 : 0 : 0;
        1  0 : 1 : 1;
        1  0 : 0 : -;
        0  1 : ? : -;
        0  0 : ? : x;
    endtable
endprimitive

```

Example 260 tmp.cfg

```

defaultvalue 1

instance top.m1 0
module top 1

```

Run the design using the following commands:

```

% vcs -sverilog -full64 tmp.v +vcs+initreg+random
% simv +vcs+initreg+deferred_time+220
+vcs+initreg+deferred_config+tmp.cfg

```

The generated output is as follows:

```
time: 0 q1:0 m1.q2:0 q3:0 p:11 m1.r:'{{'{'h0, 'h0}, {'h1, 'h1}, {'h1,
'h1}}
time: 0 q1:1 m1.q2:1 q3:0 p:11 m1.r:'{{'{'h0, 'h0}, {'h1, 'h1}, {'h1,
'h1}}
time: 20 q1:1 m1.q2:1 q3:0 p:xx m1.r:'{{'hx, 'hx}, {'hx, 'h1}, {'h1,
'hx}}
time: 40 q1:1 m1.q2:1 q3:x p:xx m1.r:'{{'hx, 'hx}, {'hx, 'h1}, {'h1,
'hx}}
time: 80 q1:x m1.q2:xx q3:xx p:xx m1.r:'{{'hx, 'hx}, {'hx, 'h1}, {'h1,
'hx}}
time:220 q1:1 m1.q2:0 q3:1 p:11 m1.r:'{{'h0, 'h0}, {'h0, 'h1}, {'h1,
'h0}}
V C S      S i m u l a t i o n      R e p o r t
```

In this example, the signals are initialized as specified in the `tmp.cfg` file.

Limitations

The following are the limitations of this feature:

- This feature is not supported in Native Low Power (NLP).
- The `$deferred_initreg` system task is not compatible with other use models of the deferred initreg feature.
- You must explicitly call `$cm_stop()` and `$cm_start()` to guard coverage at the time of using the `$deferred_initreg()` system task.
- The two-state signal types like `bit/logic` and pre-initialized types like `int` are not initialized.
- This feature is not compatible with Logic2Wire optimization and partial-elaboration flow.

General Options

Viewing the Compile Time Options

`-sig program`

Starts the `program` that displays the compile time options that were on the `vcs` command line when you created the `simv` (or `simv.exe`) executable file.

For example, `% simv -sig echo`

You cannot use any other runtime option with the `-sig` option.

Recording Where ACC Capabilities are Used

+vcs+learn+pli

ACC capabilities enable debugging operations, however, they have a performance cost, so enable them where you need them. This option keeps track of where in your design you use them for debugging operations so that you can recompile your design, and in the next simulation, enable them only where you need them. When you use this option VCS writes the `pli_learn.tab` secondary PLI table file. You input this file with the `+applylearn` compile-time option when you recompile your design.

Suppressing the \$stop System Task

+vcs+ignorestop

Tells VCS to ignore the `$stop` system tasks in your source code.

Debugging Plusarg-Controlled Testbench Code

-Xplusargs=1

When this option is used, VCS generates a notification when the `$test$plusargs` and `$value$plusargs` system functions are evaluated. This is useful for the debug of plusarg-controlled testbench code.

VCS also prints out the return value of `$test$plusargs` and `$value$plusargs` that you can use to manually trace through the testbench code without having to run it interactively in Verdi.

Example

Example 261 test.v

```
module test();
    string val1,val2;

    initial begin
        if($value$plusargs("packet1=%s",val1)) begin
            $display("plusargs packet1=%s found.",val1);
        end

        if($value$plusargs("packet2=%s",val2)) begin
            $display("plusargs packet2=%s found.",val2);
        end

        if(!$test$plusargs("RESET")) begin
            $display("!RESET is TRUE");
        end
    end
endmodule
```

```

if($test$plusargs("SECONDARY")) begin
    $display("Go Secondary");
end
end
end

endmodule

```

To compile the above design, run the following command:

```
vcs test.sv -full64; simv -Xplusargs=1 +packet1=mypkt +SECONDARY
```

Output

VCS generates the following output on std.out:

```

PLUSARGS(Info): time:0 $value$plusargs(packet1)= 1 at xplus.v:5
  instance test.val1
plusargs packet1=mypkt found.
PLUSARGS(Info): time:0 $value$plusargs(packet2)= 0 at xplus.v:9
  instance test.val2
PLUSARGS(Info): time:0 $test$plusargs(RESET)= 0 at xplus.v:14 instance
  test
!RESET is TRUE
PLUSARGS(Info): time:0 $test$plusargs(SECONDARY)= 1 at xplus.v:16
  instance test
Go Secondary

```

Enabling User-defined Plusarg Options

```
+plus-options
```

User-defined runtime options to perform some operation when the option is on the `simv` command line. The `$test$plusargs` system task can check for such options.

Enabling Overriding the Timing of a SWIFT SmartModel

```
+override_model_delays
```

Instead of using the `DelayRange` parameter definition in the template file, this option enables the `+mindelays`, `+typdelays`, and `+maxdelays` runtime options to specify the timing used by SWIFT SmartModels.

Enabling feature beyond VHDL LRM

```
-xlrn
```

Enables VHDL features beyond those described in *IEEE Standard VHDL 1076-2008 LRM*.

Enabling Loop Detect

`+vcs+loopreport+number`

It is mandatory to include the `+vcs+loopreport+number` option at compile time, though you can override the threshold number at runtime.

When `+vcs+loopreport+number` is specified at compile time, VCS does the following based on the option specified at runtime:

- If `+vcs+loopreport` is specified at runtime, VCS checks if a simulation event loops for 2,000,000 times (by default) in the same simulation time tick, and issues a runtime warning message. VCS also terminates the simulation and generates a report when a zero delay loop is detected.
- If `+vcs+loopreport+N` is specified at runtime, VCS checks if the simulation event loops for 'N' times and issues a runtime warning message. VCS also terminates the simulation.

For information about using the `+vcs+loopreport+number` option during compile time, see [Enabling Loop Detect](#).

`+vcs+loopdetect+number`

When `+vcs+loopdetect+number` is not specified at compile time, VCS does the following based on the option specified at runtime:

- If `+vcs+loopdetect` is specified at runtime, VCS checks if a simulation event loops for 2,000,000 times (by default) in the same simulation time tick, and issues a runtime error message. VCS also terminates the simulation.
- If `+vcs+loopdetect+N` is specified at runtime, VCS checks if the simulation event loops for 'N' times and issues a runtime error message. VCS also terminates the simulation.

For information about using the `+vcs+loopdetect+number` option compile time, see Section [Enabling Loop Detect](#).

Note:

The `+vcs+loopdetect` and `+vcs+loopreport` options are mutually exclusive. It is recommended not to use both the options at the same time.

Specifying acc_handle_simulated_net PLI Routine

`+vcs+mipd+noalias`

For the `acc_handle_simulated_net` PLI routine, aliasing of a loconn net and a hiconn net across the port connection is disabled if MIPD delay annotation happens for the port. If

you specify ACC capability: mip or mipb in the `pli.tab` file, such aliasing is disabled only when actual MIPD annotation happens.

If during a simulation run, `acc_handle_simulated_net` is called before MIPD annotation happens, VCS issues a warning message. When this happens you can use this option to disable such aliasing for all ports whenever mip, mipb capabilities have been specified. This option works for reading an ASCII SDF file during simulation and not for compiled SDF files.

Loading DPI Libraries Dynamically at Runtime

```
-sv_lib library_path_name
-sv_root library_path_name
-sv_liblist library_path_name
```

The procedure for loading a DPI library at runtime is as follows:

1. Analyze and compile/elaborate the Verilog or SystemVerilog code, for example:

```
%vlogan -sverilog other_options test.v
% vcs top_level
%> vcs -sverilog other_options test.v
```

2. Compile the C code and create a shared object, for example:

```
%> gcc -fPIC -Wall ${CFLAGS} -I${VCS_HOME}/include \
other_libraries -c test.c
%> gcc -fPIC -shared ${CFLAGS} -o test.so test.o
```

3. Load the shared object at runtime using one of the following runtime options for this purpose:

```
-sv_lib -sv_root -sv_liblist
```

Loading PLI Libraries Dynamically at Runtime

```
-load library_path_name
```

Loads a PLI library dynamically at runtime. Enter the `-load` option for each library you are dynamically loading. For example,

```
% simv -load ./pli1.so -load ./pli2.so
```

To use this runtime option, when you compile the design include the PLI table file for the PLI libraries with the `-P` compile-time option:

```
% vcs -P pli.tab design_source_files
```

Independent Seeding Across Multiple Instances

```
-xlrn hier_inst_seed
```

Uses instance-specific initialization seeds for random number generators of modules, programs, interfaces, and packages. The default flow uses the same initialization seed across all instances. For example,

```
% simv -xlrn hier_inst_seed
```

Use the `-diag hier_seed` diagnostic option at run time to enable diagnostics. The diagnostics are reported in an ASCII text file named `HierInstanceSeed.txt`. This file is generated in the current working directory and cannot be renamed during simulation. You can rename the file after simulation. There is no VCS option to rename the file.

The diagnostics report file captures the seed values used for each module instance.

E

Verilog Compiler Directives and System Tasks

This appendix describes:

- [Compiler Directives](#)
 - [System Tasks and Functions](#)
-

Compiler Directives

Compiler directives are commands in the source code that specify how VCS compiles the source code that follows them, both in the source files that contain these compiler directives and in the remaining source files that VCS subsequently compiles.

Compiler directives are not effective down the design hierarchy. A compiler directive written above a module definition affects how VCS compiles that module definition, but does not necessarily affect how VCS compiles module definitions instantiated in that module definition. If VCS has already compiled these lower-level module definitions, it does not recompile them. If VCS has not yet compiled these module definitions, the compiler directive does affect how VCS compiles them.

Note:

Compile-time options override the compiler directives.

Compiler Directives for Cell Definition

``celldefine`

Specifies that the modules under this compiler directive be tagged as “cell” for delay annotation. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.10.

Syntax: ``celldefine`

``endcelldefine`

Disables ``celldefine`. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.10.

Syntax: ``endcelldefine`

Compiler Directives for Setting Defaults

``default_nettype`

Sets default net type for implicit nets. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.8.

Syntax: `'default_nettype wire | tri | tri0 | wand | triand | tril | wor | trior | trireg |none`

``resetall`

Resets all compiler directives. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.3.

Syntax: ``resetall`

Compiler Directives for Macros

``define`

Defines a text macro. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.5.1.

Syntax: ``define text_macro_name macro_text`

``else`

Specifies an alternative group of source code lines that VCS compiles if the text macro specified with an ``ifdef` compiler directive is not defined. It is used with the ``ifndef` compiler directive. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.6.

Syntax: ``else second_group_of_lines`

``elseif`

Specifies an alternative group of source code lines that VCS compiles if the text macro specified with an `'ifdef` compiler directive is not defined, but the text macro specified with this compiler directive is defined. It is used with the `'ifndef` compiler directive. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.6.

Syntax: ``elseif text_macro_name second_group_of_lines`

``endif`

Specifies the end of a group of lines specified by the `'ifdef` or `'else` compiler directives. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.6.

Syntax: ``endif`

```
`ifdef
```

Specifies compiling the source lines that follow if the specified text macro is defined by either the `\define` compiler directive or the `+define` compile-time option.

Syntax: `\ifdef text_macro_name group_of_lines 'endif`

```
'ifdef VCS
```

The character string `VCS` is a predefined text macro in VCS. The Verilog or SystemVerilog code that follows `\ifdef VCS` is the code that you want to compile by VCS. The code that follows a corresponding `\else` compiler directive is the source code that VCS ignores.

You can insert source code after the `\else` compiler directive that you intend for a third-party tool.

In the following source code, VCS compiles and executes the first block of code and ignores the second block, even when you do not include the `\define VCS` compiler directive or the `+define+VCS` compile-time option:

```
`ifdef VCS
    begin
        // Block of code for VCS      M
    end
`else
    begin
        // third party code
        M
    end
`endif
```

When you encrypt the source code, VCS inserts `\ifdef VCS` before all encrypted parts of the code.

You can use the option `-undef_vcs_macro` to cancel the VCS predefined text macro. As a result, the `\ifdef VCS ... \endif` block contents are skipped. You can use this option mainly for different VCS modes (other than simulation) where these block contents are not expected to pass through.

```
`ifndef
```

Specifies compiling the source code that follows if the specified text macro is not defined. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.6.

Syntax: `\ifndef text_macro_name group_of_lines`

```
`undef
```

Undefines a macro definition. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.5.2.

Syntax: `\undef text_macro_name`

Compiler Directives for Delays

``delay_mode_path`

Ignores the delay specifications on all gates and switches in all those modules under this compiler directive that contain the specify blocks. Uses only the module path delays and the delay specifications on continuous assignments.

Syntax: ``delay_mode_path`

``delay_mode_distributed`

Ignores the module path delays specified in the specify blocks in modules under this compiler directive and uses only the delay specifications on all gates, switches, and continuous assignments.

Syntax: ``delay_mode_distributed`

``delay_mode_unit`

Ignores the module path delays. Changes all the delay specifications on all gates, switches, and continuous assignments to the shortest time precision argument of all the `'timescale` compiler directives in the source code. The default time unit and the time precision argument of the `'timescale` compiler directive is 1 ns.

Syntax: ``delay_mode_unit`

``delay_mode_zero`

Changes all the delay specifications on all gates, switches, and continuous assignments to zero and changes all module path delays to zero.

Syntax: ``delay_mode_zero`

Compiler Directives for Back Annotating SDF Delay Values

``vcs_mipdexpand`

This compiler directive enables the runtime back-annotation of individual bits of a port declared in an ASCII text SDF file. This is done by entering the compiler directive over the port declarations for these ports. Similarly, entering this compiler directive over the port declarations enables a PLI application to pass delay values to the individual bits of a port.

As an alternative to using this compiler directive, you can use the `+vcs+mipdexpand` compile-time option, or you can enter the `mipb` ACC capability. For example:

```
$sdf_annotate call=sdf_annotate_call acc+=rw,mipb:top_level_mod+
```

When you compile the SDF file, which Synopsys recommends, you do not need to use this compiler directive to back-annotate the delay values for individual bits of a port.

``vcs_mipdnoexpand`

Turns off the enabling of back-annotating delay values on individual bits of a port as specified by a previous ``vcs_mipdexpand` compiler directive.

Compiler Directives for Source Protection

For information about compiler directives for source protection, see Chapter [Encrypting Source Files](#).

General Compiler Directives

Compiler Directive for Including a Source File

``include`

Includes (also compiles as part of the design) the specified source file. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.4.

Syntax: ``include "filename"`

Note:

If the included file is a different version of Verilog from the source file that contains the `'include` compiler directive, and you want VCS to compile the included file for the version specified by its filename extension, enter the `-extinclude` compile-time option, see [Options for Different Versions of Verilog](#).

Compiler Directive for Setting the Time Scale

``timescale`

Sets the timescale. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.4.

Syntax: ``timescale time_unit / time_precision`

In VCS, the default time unit is 1 s (a full second) and the default time precision is also 1 s.

Compiler Directive for Specifying a Library

`'uselib file | directory`

Searches the specified library for unresolved modules. You can specify either a library file or a library directory.

Syntax:

`'uselib file = filename`

or

```
`uselib dir = directory_name libext+.ext | libext=.ext
```

Enter path names if the library file or directory is not in the current directory. For example:

```
`uselib file = /sys/project/speclib.lib
```

If specifying a library directory, include the `libext+.ext` keyword and append to it the extensions of the source files in the library directory, similar to the `+libext+.ext` compile-time option. For example:

```
`uselib dir = /net/designlibs/project.lib libext+.v
```

To specify more than one search library, enter additional `dir` or `file` keywords, for example:

```
`uselib dir = /net/designlibs/library1.lib dir=/net/designlibs/
library2.lib libext+.v
```

Here, the `libext+.ext` keyword applies to both the libraries.

Compiler Directive for File Names and Line Numbers

```
`line line_number "filename" level
```

Maintains the file name and the line number. For more details, see IEEE SystemVerilog LRM Std 1800™-2012 Section 22.12.

Unimplemented Compiler Directives

The following compiler directives are IEEE SystemVerilog LRM Std 1800™-2012 compiler directives that are not yet implemented in VCS:

- ``unconnected_drive`
- ``nounconnected_drive`

System Tasks and Functions

This section describes the system tasks and functions that are supported by VCS and then lists the system tasks that it does not support.

System tasks are described in the IEEE SystemVerilog LRM Std 1800™-2012 for more information.

System Tasks for SystemVerilog Assertions Severity

`$fatal`

Generates a runtime fatal assertion error.

`$error`

Generates a runtime assertion error.

`$warning`

Generates a runtime warning message.

`$info`

Generates an information message.

System Tasks for SystemVerilog Assertions Control

`$assertoff`

Tells VCS to stop monitoring any of the specified assertions that start at a subsequent simulation time.

`$assertkill`

Tells VCS to stop monitoring any of the specified assertions that start at a subsequent simulation time, and stop the execution of any of these assertions that are now occurring.

`$asserton`

Tells VCS to resume the monitoring of assertions that it stopped monitoring due to a previous `$assertoff` or `$assertkill` system task.

These system tasks provide file name and line number from where these system tasks are called, which might otherwise be difficult to track in the absence of this information.

Note:

- The runtime option `-assert old_ctrl_msg` reverts the messaging to the old style for backward compatibility.
- It is recommended to use the `$assertoff` system task with arguments, as shown to turn off reporting of assertions globally for the entire design:

`$assertoff (0, "top_level_module")`

You may not be able to enable assertions on the desired hierarchies, if you use `$assertoff` without arguments to turn off assertions.

System Tasks for SystemVerilog Assertions

`$onehot`

Returns true if only one bit in the expression is true.

`$onehot0`

Returns true if, at the most, one bit of the expression is true (also returns true if none of the bits are true).

`$isunknown`

Returns true if one of the bits in the expression has an X value.

`$countx(expression)`

Returns the number of expression bits set to X.

`$countz(expression)`

Returns the number of expression bits set to Z.

`$countunknown(expression)`

Returns the number of expression bits set to either X or Z.

`$onedriven`

Returns true if only one bit of the expression is not Z, and its value is defined (not X).

`$onedriven0`

Returns true if at most one bit of the expression is not Z, and if such a bit exists, its value is defined (not X).

System Tasks for VCD Files

VCD files are ASCII files that contain a record of a net or register's transition times and values. There are number of third-party products that read VCD files to show you simulation results. VCS has the following system tasks for specifying the names and contents of these files. They require the `$dumpvars` system task.

`$dumpall`

Creates a checkpoint in the VCD file. When VCS executes this system task, VCS records the current values of all specified nets and registers them into the VCD file, irrespective of whether there is a value change at this time or not.

`$dumpoff`

Stops recording value change information in the VCD file.

`$dumpon`

Starts recording value change information in the VCD file.

`$dumpfile`

Specifies the name of the VCD file that you want VCS to record.

Syntax: `$dumpfile("filename");`

`$dumpflush`

Empties the VCD file buffer and writes all this data to the VCD file.

`$dumplimit`

Limits the size of a VCD file.

`$dumpvars`

Specifies the nets and variables whose transition times and values you want VCS to record in the VCD file.

Syntax: `$dumpvars(level_number,module_instance | net_or_var);`

You can specify individual nets or variables, or specify all the nets and variables in an instance.

The `$dumpvars` system task enables the other VCD system tasks, such as `$dumpon` and `$dumpfile`.

`$dumpchange`

Tells VCS to stop recording transition times and values in the current dump file and to start recording in the specified new file.

Syntax: `$dumpchange("filename");`

Code example: `$dumpchange("vcd16a.dmp");`

`$fflush`

VCS stores VCD data in the dump file buffer of the operating system. As simulation progresses, reads the date from this buffer to write to the VCD file on disk. If you need the latest information written to the VCD file at a specific time, use the `$fflush` system task.

Syntax: `$fflush("filename");`

Code example: `$fflush("vcdfile1.vcd");`

`$fflushall`

If you are writing more than one VCD file and need VCS to write the latest information to all these files at a particular time, use the `$fflushall` system task.

Syntax: `$fflushall;`

`$gr_waves`

Produces a VCD file with the name `grw.dump`. In this system task, you can specify a display label for a net or register whose transition times and values VCS records in the VCD file.

Syntax: `$gr_waves(["label",]net_or_reg,...);`

Code example: `$gr_waves("wire w1",w1, "reg r1",r1);`

System Tasks for LSI Certification VCD and EVCD Files

`$dumports`

Creates an EVCD file as specified in IEEE Verilog LRM Std. 1364-2005, Pages 338-339. For example, you can input a EVCD file into TetraMAX for fault simulation.

Syntax of the `$dumports` system task is now: `$dumports(module_instance,[module_instance,] "filename");`

You can specify more than one module instance.

Code example: `$dumports(top.middle1,top.middle2, "dumports.evcd");`

If your source code contains a `$dumports` system task and you want it to generate the simulation history files for LSI certification, include the `+dumports+lsi` runtime option.

`$dumpportsoff`

Suspends writing to files specified in `$dumports` system tasks. You can specify a file to which VCS suspends writing or specify no particular file, in which case VCS suspends writing to all files specified by `$dumports` system tasks. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 339-340.

Syntax: `$dumpportsoff("filename");`

`$dumpportson`

Resumes writing to the file after writing was suspended by a `$dumpportsoff` system task. You can specify the file to which you want VCS to resume writing or specify no particular file, in which case VCS resumes writing to all files to which writing was halted by any `$dumpportsoff` or `$dumports` system tasks. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 339-340.

Syntax: `$dumpportson("filename");`

```
$dumpportsall
```

By default, VCS writes to files only when a signal changes value. The `$dumpportsall` system task records the values of the ports in the module instances, which are specified by the `$dumpports` system task, irrespective of whether there is a value change on these ports or not. You can specify the file to which you want VCS to record the port values for the corresponding module instance or specify no particular file, in which case VCS writes port values in all files opened by the `$dumpports` system task. For more details, see IEEE Verilog LRM Std 1364-2005, Page 340.

Syntax: `$dumpportsall("filename");`

```
$dumpportsflush
```

VCS stores simulation data in a buffer during simulation from which it writes data to the file. If you want VCS to write all simulation data from the buffer to the file or files at a particular time, execute this `$dumpportsflush` system task. You can specify the file to which you want VCS to write from the buffer or specify no particular file, in which case VCS writes all data from the buffer to all files opened by the `$dumpports` system task. For more details, see IEEE Verilog LRM Std 1364-2005, Page 341.

Syntax: `$dumpportsflush("filename");`

```
$dumpportslimit
```

Specifies the maximum file size of the file specified by the `$dumpports` system task. You can specify the file size in bytes. When the file reaches this limit, VCS no longer writes to the file. You can specify the file whose size you want to limit or specify no particular file, in which case your specified size limit applies to all files opened by the `$dumpports` system task. For more details, see IEEE Verilog LRM Std 1364-2005, Page 340.

Syntax: `$dumpportslimit(filesize,"filename");`

System Tasks for VPD Files

VPD files are files that store the transition times and values for nets and registers but they differ from VCD files in the following ways:

- You can use the GUI to view the simulation results that VCS recorded in a VPD file. You cannot actually load a VCD file directly into GUI. When you load a VCD file, GUI translates the file to VPD and loads the VPD file.
- They are binary format and therefore take less disk space and load much faster.
- They can also record the order of statement execution so that you can use the source window in GUI to step through the execution of your code if you specify recording this information.

VPD files are commonly used in post-processing, where VCS writes the VPD file during batch simulation. After that you can review the simulation results using GUI.

There are system tasks that specify the information that VCS writes in the VPD file.

Note:

To use the system tasks for VPD files, you must compile your source code with the `-debug_access` option.

`$vcdplusautoflushoff`

Turns off the automatic flushing of simulation results to the VPD file whenever there is an interrupt, such as when VCS executes the `$stop` system task.

Syntax: `$vcdplusautoflushoff;`

`$vcdplusautoflushon`

Tells VCS to flush or write all the simulation results in memory to the VPD file whenever there is an interrupt, such as when VCS executes a `$stop` system task or when you halt VCS using the UCLI `stop` command, or using the **Stop** button on the GUI.

Syntax: `$vcdplusautoflushon;`

`$vcdplusclose`

Tells VCS to mark the current VPD file as completed and close the file.

Syntax: `$vcdplusclose;`

`$vcdplusdeltacycleon`

The `$vcdplusdeltacycleon` task enables reporting of delta cycle information from the Verilog source code. It must be followed by the appropriate `$vcdpluson`/`$vcdplusoff` task.

Glitch detection is automatically turned on when VCS executes `$vcdplusdeltacycleon` unless you have previously used `$vcdplusglitchon/off`. Once you use `$vcdplusglitchon/off`, GUI allows you to explicitly control the glitch detection.

Syntax: `$vcdplusdeltacycleon;`

Delta cycle collection can start only at the beginning of a time sample. The `$vcdplusdeltacycleon` task must precede the `$vcdpluson` command to ensure that delta cycle collection starts at the beginning of the time sample.

`$vcdplusevent`

The `$vcdplusevent` task allows you to record a unique event for a signal at the current simulation time unit.

Syntax

```
$vcdplusevent(net_or_reg, "event_name", "<E|W|I><S|T|D>");
```

A symbol is displayed in GUI on the signal's waveform and in the *Logic* browser. The `event_name` argument appears in the status bar when you click the symbol.

`E|W|I` — Specifies severity.

- `E` for error, displays a red symbol.
- `W` for warning, displays a yellow symbol.
- `I` for information, displays a green symbol.

`S|T|D` — Specifies the symbol shape.

- `S` for square.
- `T` for triangle.
- `D` for diamond.

Do not enter space between the arguments `E|W|I` and `S|T|D`. Do not include angle brackets `<>`. There is a limit of 244 unique events.

`$vcdplusfile`

Specifies the next VPD file that GUI opens during simulation after it executes the `$vcdplusclose` system task and when it executes the next `$vcdpluson` system task.

Syntax: `$vcdplusfile("filename");`

`$vcdplusglitchon`

Turns on the checking for zero delay glitches and other cases of multiple transitions for a signal at the same simulation time.

Syntax: `$vcdplusglitchon;`

`$vcdplusflush`

Tells VCS to flush or write all the simulation results in memory to the VPD file at the time VCS executes this system task. Use `$vcdplusautoflushon` to enable automatic flushing of simulation results to the file when the simulation stops.

Syntax: `$vcdplusflush;`

`$vcdplusmemon`

Records value changes and times for memories and multidimensional arrays (MDAs).

Syntax: `system_task(Mda [, dim1Lsb [, dim1Rsb [, dim2Lsb [, dim2Rsb [, ... dimNLsb [, dimNRsb]]]]]);`

Where,

`Mda`

Specifies the name of the MDA to be recorded. It must not be a part select. If no other arguments are given, then all elements of the MDA are recorded to the VPD file.

`dim1Lsb`

This is an optional argument that specifies the name of the variable that contains the left bound of the first dimension. If no other arguments are given, then all elements under this single index of this dimension are recorded.

`dim1Rsb`

This is an optional argument that specifies the name of variable that contains the right bound of the first dimension.

The `dim1Lsb` and `dim1Rsb` arguments specify the range of the first dimension to be recorded. If no other arguments are specified, then all elements under this range of addresses within the first dimension are recorded.

`dim2Lsb`

This is an optional argument with the same functionality as `dim1Lsb`, but refers to the second dimension.

`dim2Rsb`

This is an optional argument with the same functionality as `dim1Rsb`, but refers to the second dimension.

`dimNLsb`

This is an optional argument that specifies the left bound of the Nth dimension.

`dimNRsb`

This is an optional argument that specifies the right bound of the Nth dimension.

Note that MDA system tasks can take 0 or more arguments, with the following caveats:

- No arguments: The whole design is traversed and all memories and MDAs are recorded. Note that this process may cause significant memory usage and simulator drag.
- One argument: If the object is a scope instance, all memories/MDAs contained in that scope instance and its children are recorded. If the object is a memory/MDA, that object is recorded.

`$vcdplusmemoff`

Stops recording value changes and times for memories and multidimensional arrays.
 Syntax is same as the `$vcdplusmenon` system task.

`$vcdplusmemorydump`

Records (dumps) a snapshot of the values in a memory or multidimensional array into the VPD file. Syntax is same as the `$vcdplusmenon` system task.

`$vcdplusoff`

Stops recording the transition times and values for the nets and registers in the specified module instance or individual nets or registers in the VPD file.

Syntax: `$vcdplusoff[(level_number,module_instance | net_or_reg)];`

Where:

`level_number`

Specifies the number of hierarchy scope levels for which to stop recording the signal value changes (a zero value records all scope instances to the end of the hierarchy, which is the default value).

`module_instance`

Specifies the name of the scope for which to stop recording the signal value changes (default is all).

`net_or_reg`

Specifies the name of the signal for which to stop recording the signal value changes (default is all).

`$vcdpluson`

Starts recording the transition times and values for the nets and variables in the specified module instance or individual nets or variable in the VPD file. This system task does not enable recording memories or multidimensional arrays with an unpacked dimension.

Syntax: `$vcdpluson[(level_number,module_instance | net_or_variable)];`

where:

`level_number`

Specifies the number of hierarchy scope levels for which to record signal value changes (a zero value records all scope instances to the end of the hierarchy, which is the default value).

`module_instance`

Specifies the name of the scope for which to record the signal value changes (default is all).

`net_or_variable`

Specifies the name of the signal for which to record the signal value changes (default is all).

System Tasks for SystemVerilog Assertions

Note:

Enter these system tasks in an `initial` block. Do not enter them in an `always` block.

`$assert_monitor`

Analogous to the standard `$monitor` system task, it continually monitors the specified assertions and displays what is happening with them (you can only have it display on the next clock of the assertion). The syntax is as follows:

`$assert_monitor([0|1,] assertion_identifier...);`

Where:

`0`

Specifies reporting on the assertions if they are active (VCS checks for its properties). If the assertion is not active, assertions are reported whenever they start.

`1`

Specifies reporting on the assertions only once; the next time they start.

If you specify neither `0` or `1`, the default is `0`.

`assertion_identifier...`

A comma separated list of assertions. If one of these assertions is not declared in the module definition containing this system task, you can specify it by its hierarchical name.

`$assert_monitor_off`

Disables the display from the `$assert_monitor` system task.

`$assert_monitor_on`

Re-enables the display from the `$assert_monitor` system task.

System Tasks for Executing Operating System Commands

`$system`

Executes the operating system commands.

Syntax: `$system("command");`

Code example: `$system("mv -f savefile savefile.1");`

`$systemf`

Executes the operating system commands and accepts multiple-formatted string arguments.

Syntax: `$systemf("command %s ...", "string", ...);`

Code example: `int = $systemf("cp %s %s", "file1", "file2");`

The operating system copies the file named `file1` to a file named `file2`.

System Tasks for Log Files

`$log`

If a filename argument is included, this system task stops writing to the `vcs.log` file or the log file specified with the `-l` runtime option and starts writing to the specified file. If the file name argument is omitted, this system task tells VCS to resume writing to the log file after writing to the file was suspended by the `$nolog` system task.

Syntax: `$log[("filename")];`

Code example: `$log("reset.log");`

`$nolog`

Disables writing to the `vcs.log` file or the log file specified by either the `-l` runtime option or the `$log` system task.

Syntax: `$nolog;`

System Tasks for Data Type Conversions

`$bitstoreal[b]`

Converts a bit pattern to a real number. For more details, see IEEE Verilog LRM Std 1364-2005, Page 311.

`$itor[i]`

Converts integers to real numbers. For more details, see IEEE Verilog LRM Std 1364-2005, Page 310.

`$realtobits`

Passes bit patterns across module ports, converting a real number to a 64-bit representation. For more details, see IEEE Verilog LRM Std 1364-2005, Page 310.

`$rtoi`

Converts real numbers to integers. For more details, see IEEE std 1364-2001, page 310.

System Tasks for Displaying Information

`$display[b|h|0];`

Displays the arguments. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 278-285.

`$monitor[b|h|0]`

Displays the data when the arguments change the value. For more details, see IEEE Verilog LRM Std 1364-2005, Page 286.

`$monitoroff`

Disables the `$monitor` system task. For more details, see IEEE Verilog LRM Std 1364-2005, Page 286.

`$monitoron`

Re-enables the `$monitor` system task after it was disabled with the `$monitoroff` system task. For more details, see IEEE Verilog LRM Std 1364-2005, Page 286.

`$strobe[b|h|0];`

Displays simulation data on the selected time. For more details, see IEEE Verilog LRM Std 1364-2005, Page 285.

`$write[b|h|0]`

Displays the text. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 278-285.

System Tasks for File I/O

`$fclose`

Closes a file. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 287-289.

`$fdisplay[b|h|0]`

Writes to a file. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 288-289.

`$ferror`

Returns additional information about an error condition in file I/O operations. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 294-295.

`$fflush`

Writes buffered data to files. For more details, see IEEE Verilog LRM Std 1364-2005, Page 295.

`$fgetc`

Reads a character from a file. For more details, see IEEE Verilog LRM Std 1364-2005, Page 290.

`$fgets`

Reads a string from a file. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 290.

`$fmonitor [b|h|0]`

Writes to a file when an argument value changes. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 288-289.

`$fopen`

Opens the files. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 287-289.

`$fread`

Reads the binary data from a file. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 293.

`$fscanf`

Reads the characters in a file. For more details, see IEEE Verilog LRM Std 1364-2005, Page 291.

`$fseek`

Sets the position of the next read or write operation in a file. For more details, see IEEE Verilog LRM Std 1364-2005, Page 294.

`$fstrobe [b|h|0]`

Writes arguments to a file. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 288-289.

`$ftell`

Returns the offset of a file. For more details, see IEEE Verilog LRM Std 1364-2005, Page 294.

`$fwrite[b|h|0]`

Writes to a file. For more details, see IEEE Verilog LRM Std 1364-2005, Page 288.

`$rewind`

Sets the next read or write operation to the beginning of a file. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 294-295.

`$sformat`

Assigns a string value to a specified signal. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 289.

`$sscanf`

Reads characters from an input stream. For more details, see IEEE Verilog LRM Std 1364-2005, Page 291.

`$swrite`

Assigns a string value to a specified signal, similar to the `$sformat` system function. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 289.

`$ungetc`

Returns a character to the input stream. For more details, see IEEE Verilog LRM Std 1364-2005, Page 290.

System Tasks for Loading Memories

`$readmemb`

Loads binary values from a specified file into a specified memory. For more details, see IEEE Verilog LRM Std 1364-2005, Page 296.

`$readmemh`

Loads hexadecimal values from a specified file into a specified memory. For more details, see IEEE Verilog LRM Std 1364-2005, Page 296.

`$sreadmemb`

Loads specified binary string values into memories. For more details, see IEEE Verilog LRM Std 1364-2005, Page 517.

`$sreadmemh`

Loads specified string hexadecimal values into memories. For more details, see IEEE Verilog LRM Std 1364-2005, Page 517.

`$writememb`

Writes binary data from a specified memory to a specified file.

Syntax: `$writememb ("filename",memory [,start_address] [,end_address]);`

Code example: `$writememb ("testfile.txt",mem,0,255);`

`$writememh`

Writes hexadecimal data from a specified memory to a specified file.

Syntax: `$writememh ("filename",memory [,start_address] [,end_address]);`

System Tasks for Time Scale

`$printtimescale`

Displays the time unit and time precision from the last '`'timescale`' compiler directive that VCS has read before it reads the module definition containing this system task. For more details, see IEEE Verilog LRM Std 1364-2005, Page 299.

`$timeformat`

Specifies how the `%t` format specification reports time information. For more details, see IEEE Verilog LRM Std 1364-2005, Page 300.

System Tasks for Simulation Control

`$stop`

Causes a simulation to be suspended. For more details, see IEEE Verilog LRM Std 1364-2005, Page 302.

`$finish`

Causes a simulation to end. For more details, see IEEE Verilog LRM Std 1364-2005, Page 302.

System Tasks for Timing Checks

`$disable_warnings`

Disables the display of timing violations and toggling of notifier registers.

Syntax: `$disable_warnings[(module_instance,...)];`

An alternative syntax is:

```
$disable_warnings("timing"[,module_instance,...]);
```

If you specify a module instance, this system task disables timing violations for the specified instance and all instances hierarchically under this instance. If you omit module instances, this system task disables timing violations throughout the design.

Code example: \$disable_warnings(seqdev1);

```
$enable_warnings
```

Re-enables the display of timing violations and toggling of notifier registers after the execution of the `$disable_warnings` system task. This system task does not enable timing violations during simulation when you used the `+no_tchk_msg` compile-time option to disable them.

Syntax: `$enable_warnings[(module_instance,...)];`

An alternative syntax is:

```
$enable_warnings("timing"[,module_instance,...]);
```

If you specify a module instance, this system task enables timing violations for the specified instance and all instances hierarchically under this instance. If you omit module instances, this system task enables timing violations throughout the design.

Timing Checks for Clock and Control Signals

`$hold`

Reports a timing violation when a data event happens too soon after a reference event. For more details, see IEEE Verilog LRM Std 1364-2005, Page 242.

`$nochange`

Reports a timing violation if the data event occurs during the specified level of the control signal (the reference event). For more details, see IEEE Verilog LRM Std 1364-2005, Page 257.

`$period`

Reports a timing violation when an edge triggered event happens too soon after the previous matching edge triggered an event on a signal. For more details, see IEEE Verilog LRM Std 1364-2005, Page 256.

`$recovery`

Reports a timing violation when a data event happens too soon after a reference event. Unlike the `$setup` timing check, the reference event must include the `posedge` or `negedge`

keyword. Typically, the `$recovery` timing check has a control signal, such as clear, as the reference event and the clock signal as the data event. For more details, see IEEE Verilog LRM Std 1364-2005, Page 246.

`$recrrem`

Reports a timing violation if a data event occurs less than a specified time limit before or after a reference event. This timing check is identical to the `$setuphold` timing check except that typically the reference event is on a control signal and the data event is on a clock signal. You can specify negative values for the recovery and removal limits. The syntax is as follows: `$recrrem(reference_event, data_event, recovery_limit, removal_limit, notifier, timestamp_cond, timecheck_cond, delay_reference, delay_data);`

For more details, see IEEE Verilog LRM Std 1364-2005, Page 247.

`$removal`

Reports a timing violation if the reference event, typically an asynchronous control signal, happens too soon after the data event, the clock signal. For more details, see IEEE Verilog LRM Std 1364-2005, Page 245.

`$setup`

Reports a timing violation when the data event happens before and too close to the reference event. For more details, see IEEE Verilog LRM Std 1364-2005 Page 241. This timing check also has an extended syntax, such as the `$recrrem` timing check. This extended syntax is not described in IEEE Verilog LRM Std 1364-2005.

`$setuphold`

Combines the `$setup` and `$hold` system tasks. For the official description, see IEEE Verilog LRM Std 1364-2005, Page 238.

The syntax is as follows: `$setuphold(reference_event, data_event, setup_limit, hold_limit, notifier, timestamp_cond, timecheck_cond, delay_reference, delay_data);`

`$skew`

Reports a timing violation when a reference event happens too long after a data event. For more information, see IEEE Verilog LRM Std 1364-2005, Page 249.

`$width`

Reports a timing violation when a pulse is narrower than the specified limit. For more information, see IEEE Verilog LRM Std 1364-2005, Page 255. VCS ignores the threshold argument.

System Tasks for PLA Modeling

`$async$and$array to $sync$nor$plane`

For more details, see IEEE Verilog LRM Std 1364-2005, Page 303.

System Tasks for Stochastic Analysis

`$q_add`

Places an entry on a queue in stochastic analysis. For more details, see IEEE Verilog LRM Std 1364-2005, Page 307.

`$q_exam`

Provides statistical information about the activity that is in the queue. For more details, see IEEE Verilog LRM Std 1364-2005, Page 307.

`$q_full`

VCS returns 0 if the queue is not full. It returns a 1 if the queue is full. For more details, see IEEE Verilog LRM Std 1364-2005 Page, 308.

`$q_initialize`

Creates a new queue. For more details, see IEEE Verilog LRM Std 1364-2005, Page 307.

`$q_remove`

Receives an entry from a queue. For more details, see IEEE Verilog LRM Std 1364-2005, Page 307.

System Tasks for Simulation Time

`$realtime`

Returns a real number time. For more details, see IEEE Verilog LRM Std 1364-2005, Page 310.

`$stime`

Returns an unsigned integer that is a 32-bit time. For more details, see IEEE Verilog LRM Std 1364-2005, Page 309.

`$time`

Returns an integer that is a 64-bit time. For more details, see IEEE Verilog LRM Std 1364-2005, Page 309.

System Tasks for Probabilistic Distribution

`$dist_exponential`

Returns random numbers where the distribution function is exponential. For more details, see IEEE Verilog LRM Std 1364-2005, Page 312.

`$dist_normal`

Returns random numbers with a specified mean and standard deviation. For more details, see IEEE Verilog LRM Std 1364-2005, Page 312.

`$dist_poisson`

Returns random numbers with a specified mean. For more details, see IEEE Verilog LRM Std 1364-2005, Page 312.

`$dist_uniform`

Returns random numbers uniformly distributed between parameters. For more details, see IEEE Verilog LRM Std 1364-2005, Page 312.

`$random`

Provides a random number. For more details, see IEEE Verilog LRM Std 1364-2005 Page 311. Using this system function in certain kind of statements might cause simulation failure.

`$get_initial_random_seed`

Returns the integer number used as the seed for a simulation run, if the seed was set by +ntb_random_seed=value or by +ntb_random_seed_automatic or returns the default random seed value if the seed was not set using one of those two options. The default random seed value is 1.

System Tasks for Resetting VCS

`$reset`

Resets the simulation time to 0. For more details, see IEEE Verilog LRM Std 1364-2005, Page 514.

`$reset_count`

Keeps track of the number of times VCS executes the `$reset` system task in a simulation session. For more details, see IEEE Verilog LRM Std 1364-2005, Page 741-742.

`$reset_value`

System function that you can use to pass a value from, before or after VCS executes the `$reset` system task, that is, you can enter a `reset_value` integer argument to the `$reset` system task. After VCS resets the simulation, the `$reset_value` system function returns this integer argument. For more details, see IEEE Verilog LRM Std 1364-2005, Page 514.

General System Tasks and Functions

Checks for a Plusarg

`$test$plusargs`

Checks for the existence of a given plusarg on the runtime executable command line.

Syntax: `$test$plusargs ("plusarg_without_the_+");`

SDF Files

`$sdf_annotation`

Tells VCS to back-annotate delay values from an SDF file to your Verilog design.

Counting the Drivers on a Net

`$countdrivers`

Counts the number of drivers on a net. For more details, see IEEE Verilog LRM Std 1364-2005, Pages 511-512.

Depositing Values

`$deposit`

Deposits a value on a net, or variable, or cross-module references (XMRs). This deposited value overrides the value from any other driver of the net, or variable, or XMRs. The value propagates to all loads of the net, or variable, or XMRs. A subsequent simulation event can override the deposited value. You cannot use this system task to deposit values to part-selects.

Syntax: `$deposit(net_or_variable_or_xmr, value);`

The deposited value can be the value of another net, or variable, or XMRs. VCS also supports `$deposit` on array bit-select with non-constant index in behavioral context only.

Fast Processing Stimulus Patterns

`$getpattern`

Provides for fast processing of stimulus patterns. For more details, see IEEE Verilog LRM Std 1364-2005, Page 512.

Saving and Restarting The Simulation State

`$save`

Saves the current simulation state in a file. For more details, see IEEE Verilog LRM Std 1364-2005, Page 515.

`$restart`

Restores the simulation to the state that you had saved in the check file with the `$save` system task. For more details, see IEEE Verilog LRM Std 1364-2005, Page 515.

Checking for X and Z Values in Conditional Expressions

`$xzcheckon`

Displays a warning message every time VCS evaluates a conditional expression to have an `x` or `z` value.

Syntax: `$xzcheckon(level_number,hierarchical_name)`

`level_number` (Optional)

Specifies the number of hierarchy scope levels from the specified module instance to check for `x` and `z` values. If the number is 0 or not specified, VCS checks all scope instances to the end of the hierarchy.

`hierarchical_name` (Optional)

Hierarchical name of the module instance, that is, the top-level instance of the subhierarchy for which you want to enable checking.

`$xzcheckoff`

Suppress the warning message every time VCS evaluates a conditional expression to have an `x` or `z` value.

Syntax: `$xzcheckoff(level_number,hierarchical_name)`

`level_number` (Optional)

Specifies the number of hierarchy scope levels from the specified module instance, for which `x` and `z` value check is disabled. If the number is 0 or not specified, VCS disables the check on all scope instances to the end of the hierarchy.

`hierarchical_name` (Optional)

Hierarchical name of the module instance, that is, the top-level instance of the subhierarchy for which you want to disable checking.

Calculating Bus Widths

`$clog2`

Use this system function to calculate the bus widths, such as from parameters. The following illustrates its use:

```
integer result;
result = $clog2(n);
```

Note:

If the argument has `x` or `z` values, then that bit is considered as 1 or 0 respectively by VCS. The argument could be a vector with a few bits having `x` or `z` values.

For more information on this system function, see the *IEEE SystemVerilog LRM Std 1800-2012* Section 17.11.1 “Integer math functions”.

Displaying the Method Stack

`$stack()`

Displays stack information of lines in your code that trigger the execution of an entry of this system task. Multiple stacks are displayed for multiple entries of this system task. You can use this system task for debugging and back tracing.

You can enter this system task in modules and SystemVerilog programs, classes, packages, and interfaces. You can also enter this system task in user-defined tasks and functions and in `initial`, `always`, and `final` blocks (Synopsys recommends naming `begin-end` blocks in these `initial`, `always`, and `final` blocks).

The following code example illustrates an entry of this system task in the `test.sv` file:

```
program test;

    class C;
        static function f3();
            $stack(); // line 5
            endfunction
    endclass

    function f1();
        f2(); // line 10
        endfunction

    function f2();
        C::f3(); // line 14
        endfunction

    task t();
        f1(); // line 18
    endtask
endprogram
```

```

    endtask

    task t1();
        t(); // line 22
    endtask

    initial begin :B0
        t1(); // line 26
    end

endprogram

module top;
    test p();
endmodule

```

At runtime, VCS displays the following method stack information:

```
#0 in \C::f3 at test.sv:5
#1 in f2 at test.sv:14
#2 in f1 at test.sv:10
#3 in t at test.sv:18
#4 in t1 at test.sv:22
#5 in B0 at test.sv:26
#6 in top.p
```

In this method stack:

#0 is always the line containing the \$stack system task. In this example, it is in class C. The user-defined function f3, at line number 5 is test.sv.

#1 is a call of function f3 in the user-defined function f2 at line number 14. VCS executing function f2 causes VCS to execute the function f3.

#2 is a call of function f2 in user-defined function f1 at line number 10. VCS executing function f1 causes VCS to execute the function f2.

#3 is a call of function f1 in the user-defined task t at line number 18. VCS executing task t causes VCS to execute the function f1.

#4 is a task enabling statement for task t in the user-defined task t1 at line number 22. VCS executing task t1 causes VCS to execute the task t.

#5 is a task enabling statement for task t1 in the begin-end block named B0. VCS executing block B0 causes VCS to execute the task t1.

#6 is the instance of program test. VCS does not include the line number because this instantiation is in the top-level module.

You can use the `-frames` runtime option to control the number of lines printed in the stack as follows:

```
simv -frames <num>
```

Where `<num>` specifies the number of lines in the stack. When `<num>` is zero, VCS displays all the lines in the stack.

If debug mode is enabled, you can call the `$stack` system task from GUI.

For example:

```
ucli% step
in program p 3 1 4
mda_stack.v, 17 : $stack();

ucli% stack
0 : -line 14 -file mda_stack.v -scope
{test.P1.unnamed$$_4}
1 : -line 14 -file mda_stack.v -scope {test.P1.unnamed$$_4.unnamed$$_3}
2 : -line 17 -file mda_stack.v -scope
{test.P1.unnamed$$_4.unnamed$$_3.unnamed$$_1}
```

If the `$stack` system task is called inside a function that is exported to DPI, the “DPI function” name is displayed. The line number and details of the C code are not displayed.

For example:

```
#0 in int_from_sv at dpi_test.v:14
#1 in DPI function
#2 in int_test
```

In mixed-language simulations, the `$stack()` system task call displays only the information about the hierarchy. For example:

```
#0 in \top.r1.U1.d1.XOR2_INST at xor.v:6
#0 in top.xor_i at xor.v:6
#0 in \top.r1.U1 at t_ff_using_xor.v:11
```

In simulations with System-C at the top level, the `$stack()` system task call displays hierarchical information similar to the following:

```
#0 in \c::f at adder.v:5
#1 in \c::t at adder.v:9
#2 in unnamed$$_1 at adder.v:18
#3 in sYsTeMcToP.sc_top.adder_inst.p1
#0 in \c::f at adder.v:5
#1 in unnamed$$_1 at adder.v:19
#2 in sYsTeMcToP.sc_top.adder_inst.p1
```

In the OpenVera-SystemVerilog interoperability flow, the `$stack()` system task call displays both SystemVerilog and OpenVera information similar to the following:

```
#0 in f1 at fn_rt_sv.vr:14
#1 in \C::vera_method2 at fn_rt_sv.vr:7
#2 in unnamed$$_2 at fn_rt_sv.v:36
#3 in p

$psstack();
```

Returns a SystemVerilog string. The string provides hierarchical information of the scopes from where the system function is being called.

For example:

```
program test;

    function f();
        $display("psstack = %s", $psstack());
    endfunction

    task t2();
        f();
    endtask

    initial begin:psstack
        t2();
    end

endprogram
```

At runtime, VCS displays the following hierarchical information:

```
psstack = test.psstack.t2.f

$psstack();
```

Writes the output of `$stack` in the form of multi-line string to the variable argument of `$psstack`.

The following is an example for the `$psstack` usage:

```
class C;
    string str;
    function foo1;
        $psstack(str);
        $display(str);
    endfunction
    function foo2;
        foo1();
    endfunction
    task run;
        foo2();
    endtask
```

```
    endtask
endclass

program test;
  C obj;
  initial begin
    obj = new();
    obj.run();
  end
endprogram
```

The output is as follows:

```
#0 in \C:::foo1  at test.v:4
#1 in \C:::foo2  at test.v:8
#2 in \C:::run   at test.v:11
#3 in test at test.v:19
#4 in test
```

IEEE Standard System Tasks Not Yet Implemented

The following Verilog system tasks are included in the IEEE Verilog LRM Std 1364-2005, but are not yet implemented in VCS:

- \$dist_chi_square
- \$dist_erlang
- \$dist_t

F

PLI Access Routines

VCS includes a number of access routines. This appendix describes these access routines in the following sections:

Access Routines for Reading and Writing to Memories

VCS includes a number of access routines for reading and writing to a memory.

These access routines are as follows:

`acc_setmem_int`

Writes an integer value to specific bits in a Verilog memory word. See [Table 79](#) for details.

`acc_getmem_int`

Reads an integer value from specific bits in a Verilog memory word. See [Table 80](#) for details.

`acc_clearmem_int`

Clears a memory, that is, writes zeros to all bits. See [Table 81](#) for details.

`acc_setmem_hexstr`

Writes a hexadecimal string value to specific bits in a Verilog memory word. See [Table 82](#) for details.

`acc_getmem_hexstr`

Reads a hexadecimal string value from specific bits in a Verilog memory word. See [Table 83](#) for details.

`acc_setmem_bitstr`

Writes a string of binary bits (including x and z) to a Verilog memory word. See [Table 84](#) for details.

`acc_getmem_bitstr`

Reads a bit string from specific bits in a Verilog memory word. See [Table 85](#) for details.

`acc_handle_mem_by_fullname`

Returns the handle used by `acc_readmem`. See [Table 86](#) for details.

`acc_readmem`

Reads a data file and writes the contents to a memory. See [Table 87](#) for details.

`acc_getmem_range`

Returns the upper and lower limits of a memory. See [Table 88](#) for details.

`acc_getmem_size`

Returns the number of elements (or words or addresses) in a memory. See [Table 89](#) for details.

`acc_getmem_word_int`

Returns the integer of a memory element. See [Table 90](#) for details.

`acc_getmem_word_range`

Returns the least significant bit of a memory element and the length of the element. See [Table 91](#) for details.

acc_setmem_int

You use the `acc_setmem_int` access routine to write an integer value to specific bits in a Verilog memory word.

Table 79 acc_setmem_int

acc_setmem_int			
Synopsis:	Writes an integer value to specific bits in a memory word.		
Syntax:	<code>acc_setmem_int (memhand, value, row, start, length)</code>		
Type	Description		
Returns:	void		
Type	Name	Description	
Arguments:	handle	memhand	Handle to memory
	int	value	The integer value written in binary format to the bits in the word.
	int	row	The memory array index.

Table 79 acc_setmem_int (Continued)

	int	start	Bit number of the leftmost bit in the memory word where this routine starts writing the value.
	int	length	Starting with the start bit, specifies the total number of bits this routine writes to.
Related routines:	acc_getmem_int acc_setmem_hexstracc_getmem_hexstracc_setmem_bitstracc_getmem_bitstracc_clearmem_intacc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_intacc_getmem_word_range		

acc_getmem_int

You use the `acc_getmem_int` access routine to return an integer value for certain bits in a Verilog memory word.

Table 80 acc_getmem_int

acc_getmem_int			
Synopsis:	Returns an integer value for specific bits in a memory word.		
Syntax:	<code>acc_getmem_int (memhand, row, start, length)</code>		
	Type	Description	
Returns:	int	Integer value of the bits in the memory word.	
	Type	Name	Description
Arguments:	handle	memhand	Handle to memory
	int	row	The memory array index
	int	start	Bit number of the leftmost bit in the memory word where this routine starts reading the value.
	int	length	Specifies the total number of bits this routine reads starting with the start bit.
Related routines:	acc_setmem_intacc_setmem_hexstracc_getmem_hexstracc_setmem_bitstracc_getmem_bitstracc_clearmem_intacc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_intacc_getmem_word_range		

acc_clearmem_int

You use the `acc_clearmem_int` access routine to write zeros to all bits in a memory.

Table 81 acc_clearmem_int

acc_clearmem_int		
Synopsis:	Clears a memory word.	
Syntax:	<code>acc_clearmem_int (memhand)</code>	
	Type Description	
Returns:	<code>void</code>	
	Type Name Description	
Arguments:	<code>handle</code>	<code>memhand</code> Handle to memory
Related routines:	<code>acc_setmem_int</code> <code>acc_getmem_int</code> <code>acc_setmem_hexstracc_getmem_hexstracc_setmem_bitstracc_getmem_bitstracc_handle_mem_by_fullname</code> <code>acc_readmemacc_getmem_range</code> <code>acc_getmem_size</code> <code>acc_getmem_word_int</code> <code>acc_getmem_word_range</code>	

Examples

The following code examples illustrate how to use `acc_getmem_int`, `acc_setmem_int`, and `acc_clearmem_int`:

- [Example 262](#) shows C code that includes a number of functions to be associated with user-defined system tasks.
- [Example 263](#) shows the PLI table for associating these functions with these system tasks.
- [Example 264](#) shows the Verilog source code containing these system tasks.

Example 262 C Source Code for Functions Calling acc_getmem_int, acc_setmem_int, and acc_clearmem_int

```
#include <stdio.h>
#include "acc_user.h"
#include "vcs_acc_user.h"

void error_handle(char *msg)
{
    printf("%s", msg);
    fflush(stdout);
    exit(1);
}
```

Appendix F: PLI Access Routines

Access Routines for Reading and Writing to Memories

```

}

void set_mem()
{
    handle memhand = NULL;
    int value = -1;
    int row = -1;
    int start_bit = -1;
    int len = -1;

    memhand = acc_handle_tfarg(1);
    if(!memhand) error_handle("NULL MEM HANDLE\n");
    value = acc_fetch_tfarg_int(2);
    row = acc_fetch_tfarg_int(3);
    start_bit = acc_fetch_tfarg_int(4);
    len = acc_fetch_tfarg_int(5);

    acc_setmem_int(memhand, value, row, start_bit, len);
}

void get_mem()
{
    handle memhand = NULL;
    int row = -1;
    int start_bit = -1;
    int len = -1;
    int value = -1;

    memhand = acc_handle_tfarg(1);
    if(!memhand) error_handle("NULL MEM HANDLE\n");
    row = acc_fetch_tfarg_int(2);
    start_bit = acc_fetch_tfarg_int(3);
    len = acc_fetch_tfarg_int(4);
    value = acc_getmem_int(memhand, row, start_bit, len);
    printf("getmem: value of word %d is : %d\n", row, value);
    fflush(stdout);
}

void clear_mem()
{
    handle memhand = NULL;

    memhand = acc_handle_tfarg(1);
    if(!memhand) error_handle("NULL MEM HANDLE\n");

    acc_clearmem_int(memhand);
}

```

The function with the `set_mem` identifier calls the IEEE standard `acc_fetch_tfarg_int` routine to get the handles for arguments to the user-defined system task that you associate with this function in the PLI table file. It then assigns the handles to local

variables and calls `acc_setmem_int` to write to the specified memory in the specified word, start bit, for the specified length.

Similarly, the function with the `get_mem` identifier calls the `acc_fetch_tfarg_int` routine to get the handles for arguments to a user-defined system task and assign them to local variables. It then calls `acc_gtetmem_int` to read from the specified memory in the specified word, starting with the specified start bit for the specified length. It then displays the word index of the memory and its value.

The function with the `clear_mem` identifier likewise calls the `acc_fetch_tfarg_int` routine to get a handle and then calls `acc_clear_mem_int` with that handle.

Example 263 PLI Table File

```
$set_mem  call=set_mem acc+=rw:*
$get_mem  call=get_mem acc+=r:*
$clear_mem call=clear_mem acc+=rw:*
```

Here the `$set_mem` user-defined system task is associated with the `set_mem` function in the C code, as are the `$get_mem` and `$clear_mem` with their corresponding `get_mem` and `clear_mem` function identifiers.

Example 264 Verilog Source Code Using These System Tasks

```
module top;
// read and print out data of memory
parameter start = 0;
parameter finish = 9 ;
parameter bstart = 1 ;
parameter bfinish = 8 ;
parameter size = finish - start + 1;
reg [bfinish:bstart] mymem[start:finish];
integer i;
integer len;
integer value;

initial
begin
// $set_mem(mem_name, value, row, start_bit, len)
    $clear_mem(mymem);

// set values
#1      $set_mem(mymem, 8, 2, 1, 5);
#1      $set_mem(mymem, 32, 3, 1, 6);
#1      $set_mem(mymem, 144, 4, 1, 8);
#1      $set_mem(mymem, 29, 5, 1, 8);

// print values through acc_getmem_int
#1 len = bfinish - bstart + 1;
$display();
$display("Begin Memory Values");
for (i=start;i<=finish;i=i+1)
```

```

begin
    $get_mem(mymem,i,bstart,len);
end
$display("End Memory Values");
$display();

// display values
#1 $display();
$display("Begin Memory Display");
for (i=start;i<=finish;i=i+1)
begin
    $display("mymem word %d is %b",i,mymem[i]);
end
$display("End Memory Display");
$display();
end
endmodule

```

In this Verilog code, in the initial block, the following events occur:

1. The `$clear_mem` system task clears the memory.
2. Then the `$set_mem` system task deposits values in specified words, and in specified bits in the memory named `mymem`.
3. In a `for` loop, the `$get_mem` system task reads values from the memory and displays those values.

acc_setmem_hexstr

You use the `acc_setmem_hexstr` access routine for writing the corresponding binary representation of a hexadecimal string to a Verilog memory.

Table 82 acc_setmem_hexstr

acc_setmem_hexstr							
Synopsis:	Writes a hexadecimal string to a word in a Verilog memory.						
Syntax:	<code>acc_setmem_hexstr (memhand, hexStrValue, row, start)</code>						
	<table> <thead> <tr> <th>Type</th><th>Description</th></tr> </thead> </table>	Type	Description				
Type	Description						
Returns:	<table> <thead> <tr> <th>Type</th><th>Description</th></tr> </thead> </table>	Type	Description				
Type	Description						
Arguments:	<table> <thead> <tr> <th>Type</th><th>Name</th><th>Description</th></tr> </thead> </table>	Type	Name	Description			
Type	Name	Description					
	<table> <tbody> <tr> <td>handle</td><td>memhand</td><td>Handle to memory</td></tr> <tr> <td>char *</td><td>hexStrValue</td><td>Hexadecimal string</td></tr> </tbody> </table>	handle	memhand	Handle to memory	char *	hexStrValue	Hexadecimal string
handle	memhand	Handle to memory					
char *	hexStrValue	Hexadecimal string					

Table 82 acc_setmem_hexstr (Continued)

	int	row	The memory array index
	int	start	Bit number of the leftmost bit in the memory word where this routine starts writing the string.
Related routines:	acc_setmem_intacc_getmem_intacc_getmem_hexstracc_setmem_bitstracc_getmem_bitstracc_clearmem_intacc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_intacc_getmem_word_range		

This routine takes a value argument which is a hexadecimal string of any size and puts its corresponding binary representation into the memory word indexed by `row`, starting at the bit number `start`.

Examples

The following code examples illustrates the use of `acc_setmem_hexstr`:

- [Example 265](#) shows the C source code for an application that calls `acc_setmem_hexstr`.
- [Example 266](#) shows the contents of a data file read by the application.
- [Example 267](#) shows the PLI table file that associates the user-defined system task in the Verilog code with the application.
- [Example 268](#) shows the Verilog source that calls the application.

Example 265 C Source Code For an Application Calling acc_setmem_hexstr

```
#include <stdio.h>
#include "acc_user.h"
#include "vcsuser.h"
#define NAME_SIZE 256
#define len 100
pli()
{
    FILE *infile;
    char memory_name[NAME_SIZE] ;
    char value[len];
    handle memory_handle;
    int row,start;

    infile = fopen("initfile","r");
    while ( fscanf(infile,"%s %s %d %d ",
                  memory_name,value,&row,&start) != EOF )
    {
        printf("The mem= %s \n value= %s \n row= %d \n start= %d \n ",
               memory_name,value,row,start);
    }
}
```

```

        memory_handle=acc_handle_object(memory_name);
        acc_setmem_hexstr(memory_handle,value,row,start);
    }
}

```

[Example 265](#) shows the source code for a PLI application that:

1. Reads a data file named `initfile` to find the memory identifiers of the memories it writes to, the hexadecimal string to be converted to its bit representation when written to the memory, the index of the memory where it writes this value, and the starting bit for writing the binary value.
2. Displays where in the memory it is writing these values
3. Calls the access routine to write the values in the `initfile`.

Example 266 The Data File Read by the Application

```

testbench.U2.cmd_array 5 0 0
testbench.U2.cmd_array a5 1 4
testbench.U2.cmd_array a5a5 2 8
testbench.U1.slave_addr a073741824 0 4
testbench.U1.slave_addr 16f0612735 1 8
testbench.U1.slave_addr 2b52a90e15 2 12

```

Each line lists a Verilog memory, followed by a hex string, a memory index, and a start bit.

Example 267 PLI Table File

```
$pli call=pli acc=rw:*
```

Here the `$pli` system task is associated with the function with the `pli` identifier in the C source code.

Example 268 Verilog Source Calling the PLI Application

```

module testbench;
    monitor U1 ();
    master U2 ();
    initial begin
        $monitor($stime,,,
            "sladd[0]=%h sladd[1]=%h sladd[2]=%h load=%h
            cmd[0]=%h cmd[1]=%h cmd[2]=%h",
            testbench.U1.slave_addr[0],
            testbench.U1.slave_addr[1],
            testbench.U1.slave_addr[2],
            testbench.U1.load,
            testbench.U2.cmd_array[0],
            testbench.U2.cmd_array[1],
            testbench.U2.cmd_array[2] );
    #10;
    $pli();

```

```

    end
endmodule

module master;
    reg[31:0] cmd_array [0:2];
    integer i;
initial begin      //setup some default values
    for (i=0; i<3; i=i+1)
        cmd_array[i] = 32'h0000_0000;
end
endmodule

module monitor;
    reg load;
    reg[63:0] slave_addr [0:2];
    integer i;
initial begin //setup some default values
    for (i=0; i<3; i=i+1)
        slave_addr[i] = 64'h0000_0000_0000_0000;
    load = 1'b0;
end
endmodule

```

In Example 268 module `testbench` calls the application using the `$pli` user-defined system task for the application. The display string in the `$monitor` system task is on two lines to enhance readability.

acc_getmem_hexstr

You use the `acc_getmem_hexstr` access routine to get a hexadecimal string from a Verilog memory.

Table 83 `acc_getmem_hexstr`

acc_getmem_hexstr			
Synopsis:	Returns a hexadecimal string from a Verilog memory.		
Syntax:	<code>acc_getmem_hexstr (memhand, hexStrValue, row, start, len)</code>		
Type	Description		
Returns:	void		
Type	Name	Description	
Arguments:	handle	memhand	Handle to memory
	char *	hexStrValue	Pointer to a character array into which the string is written

Table 83 acc_getmem_hexstr (Continued)

	int	row	The memory array index
	int	start	Bit number of the leftmost bit in the memory word where this routine starts reading the string.
	int	length	Specifies the total number of bits this routine reads starting with the start bit.
Related routines:	<u>acc_setmem_intacc_getmem_intacc_setmem_hexstracc_setmem_bitstracc_getmem_bitstracc_clearmem_intacc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_intacc_getmem_word_range</u>		

acc_setmem_bitstr

You use the `acc_setmem_bitstr` access routine for writing a string of binary bits (including x and z) to a Verilog memory.

Table 84 acc_setmem_bitstr

acc_setmem_bitstr			
Synopsis:	Writes a string of binary bits to a word in a Verilog memory.		
Syntax:	<code>acc_setmem_bitstr (memhand, bitStrValue, row, start)</code>		
Type Description			
Returns:	void		
Type Name Description			
Arguments:	handle	memhand	Handle to memory
	char *	bitStrValue	Bit string
	int	row	The memory array index
	int	start	Bit number of the leftmost bit in the memory word where this routine starts writing the string.
Related routines:	<u>acc_setmem_intacc_getmem_intacc_setmem_hexstracc_getmem_hexstracc_getmem_bitstracc_clearmem_intacc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_intacc_getmem_word_range</u>		

This routine takes a value argument that is a bit string of any size, which can include the x and z values, and puts its corresponding binary representation into the memory word indexed by `row`, starting at the bit number `start`.

acc_getmem_bitstr

You use the `acc_getmem_bitstr` access routine to get a bit string, including x and z values, from a Verilog memory.

Table 85 *acc_getmem_bitstr*

acc_getmem_bitstr			
Synopsis:	Returns a hexadecimal string from a Verilog memory.		
Syntax:	<code>acc_getmem_bitstr (memhand,bitStrValue,row,start,len)</code>		
Type	Description		
Returns:	void		
Type	Name	Description	
Arguments:	handle	memhand	Handle to memory
	char *	hexStrValue	Pointer to a character array into which the string is written
	int	row	The memory array index
	int	start	Bit number of the leftmost bit in the memory word where this routine starts reading the string.
	int	length	Specifies the total number of bits this routine reads starting with the start bit.
Related routines:	<code>acc_setmem_intacc_getmem_intacc_setmem_hexstracc_getmem_hexstracc_setmem_bitstracc_clearmem_intacc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_intacc_getmem_word_range</code>		

acc_handle_mem_by_fullname

Returns a handle to a memory that can only be used as a parameter to `acc_readmem`.

Table 86 acc_handle_mem_by_fullname

acc_handle_mem_by_fullname			
Synopsis:	Returns a handle to be used as a parameter to acc_readmem only		
Syntax:	acc_handle_mem_by_fullname (fullMemInstName)		
	Type	Description	
Returns:	handle	Handle to the instance	
	Type	Name	Description
Arguments:	char*	fullMemInstName	Hierarchical name for a memory
Related routines:	acc_setmem_intacc_getmem_intacc_setmem_hexstracc_getmem_hexstracc_setmem_bitstracc_getmem_bitstracc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_intacc_getmem_word_range		

acc_readmem

You use the `acc_readmem` access routine to read a data file into a memory. It is similar to the `$readmemb` or `$readmemh` system tasks.

The `memhandle` argument must be the handle returned by `acc_handle_mem_by_fullname`.

Table 87 acc_readmem

acc_readmem			
Synopsis:	Reads a data file into a memory		
Syntax:	acc_readmem (memhandle, data_file, format)		
	Type	Description	
Returns:	void		
	Type	Name	Description
Arguments:	handle	memhandle	Handle returned by acc_handle_mem_fullname
	const char*	data_file	Data file this routine reads

Table 87 acc_readmem (Continued)

	int	format	Specify a character that is promoted to int. 'h' for hexadecimal data, 'b' for binary data.
Related routines:	acc_setmem_intacc_getmem_intacc_setmem_hexstracc_getmem_hexstracc_setmem_bitstracc_getmem_bitstracc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_intacc_getmem_word_range		

Examples

The following code examples illustrate the use of `acc_readmem` and `acc_handle_mem_by_fullname`.

Example 269 C Source Code Calling Tacc_readmem and acc_handle_mem_by_fullname

```
#include "acc_user.h"
#include "vcs_acc_user.h"
#include "vcsuser.h"

int test_acc_readmem(void)
{
    const char *memName = tf_getcstringp(1);
    const char *memFile = tf_getcstringp(2);
    handle mem = acc_handle_mem_by_fullname(memName);

    if (mem) {
        io_printf("test_acc_readmem: %s handle found\n", memName);
        acc_readmem(mem, memFile, 'h');
    }
    else {
        io_printf("test_acc_readmem: %s handle NOT found\n",
                  memName);
    }
}
```

Example 270 The PLI Table File

```
$test_acc_readmem call=test_acc_readmem
```

Example 271 The Verilog Source Code

```
module top;
reg [7:0] CORE[7:0];
initial $acc_readmem(CORE, "CORE");
initial $test_acc_readmem("top.CORE", "test_mem_file");
endmodule
```

acc_getmem_range

You use the `acc_getmem_range` access routine to access the upper and lower limits of a memory.

Table 88 acc_getmem_range

acc_getmem_range			
Synopsis:	Returns the upper and lower limits of a memory		
Syntax:	<code>acc_getmem_range (memhandle, p_left_index,p_right_index)</code>		
	Type	Description	
Returns:	void		
	Type	Name	Description
Arguments:	handle	memhandle	Handle to a memory
	int*	p_left_index	Pointer to int
	int	p_right_index	Pointer to int
Related routines:	<code>acc_setmem_int</code> <code>acc_getmem_int</code> <code>acc_setmem_hexstr</code> <code>acc_getmem_hexstr</code> <code>acc_setmem_bitstr</code> <code>acc_getmem_bitstr</code> <code>acc_handle_mem_by_fullname</code> <code>acc_readmem</code> <code>acc_getmem_size</code> <code>acc_getmem_word</code> <code>acc_getmem_word_range</code>		

acc_getmem_size

You use the `acc_getmem_size` access routine to access the number of elements in a memory.

Table 89 acc_getmem_size

acc_getmem_size			
Synopsis:	Returns the number of elements in a memory		
Syntax:	<code>acc_getmem_size (memhandle)</code>		
	Type	Description	
Returns:	int	The number of elements in a memory	
	Type	Name	Description

Table 89 acc_getmem_size (Continued)

Arguments:	handle	memhandle	Handle to a memory
Related routines:	acc_setmem_intacc_getmem_intacc_setmem_hexstracc_getmem_hexstracc_setmem_bitstracc_getmem_bitstracc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_word_intacc_getmem_word_range		

acc_getmem_word_int

You use the `acc_getmem_word_int` access routine to access the integer value of an element (or word, address, or row).

Table 90 acc_getmem_word_int

acc_getmem_word_int			
Synopsis:	Returns the integer value of an element		
Syntax:	<code>acc_getmem_word_int (memhandle, row)</code>		
	Type	Description	
Returns:	int	The integer value of a row	
	Type	Name	Description
Arguments:	handle	memhandle	Handle to a memory
	int	row	The element (word address, or row) in the memory
Related routines:	acc_setmem_intacc_getmem_intacc_setmem_hexstracc_getmem_hexstracc_setmem_bitstracc_getmem_bitstracc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_range		

acc_getmem_word_range

You use the `acc_getmem_word_range` access routine to access the least significant bit of an element (or word, address, or row) and the length of the element.

Table 91 acc_getmem_word_range

acc_getmem_word_range

Table 91 acc_getmem_word_range (Continued)

Synopsis:	Returns the least significant bit of an element and the length of the element		
Syntax:	acc_getmem_word_range (memhandle, lsb, len)		
	Type	Description	
Returns:	void		
	Type	Name	Description
Arguments:	handle	memhandle	Handle to a memory
	int*	lsb	Pointer to the least significant bit
	int*	len	Pointer to the length of the element
Related routines:	acc_setmem_intacc_getmem_intacc_setmem_hexstracc_getmem_hexstracc_setmem_bitstracc_getmem_bitstracc_handle_mem_by_fullnameacc_readmemacc_getmem_rangeacc_getmem_sizeacc_getmem_word_int		

Access Routines for Multidimensional Arrays

The type for multi-dimensional arrays is defined in the `vcs_acc_user.h` file. Its name is `accMda`.

You also have the following tf and access routines for accessing data in a multi-dimensional array:

`tf_mdanodeinfo` and `tf_imdanodeinfo`

Returns access parameter node information from a multi-dimensional array. See [tf_mdanodeinfo](#) and [tf_imdanodeinfo](#) for details.

`acc_get_mda_range`

Returns all the ranges of the multi-dimensional array. See [acc_get_mda_range](#) for details.

`acc_get_mda_word_range`

Returns the range of an element in a multi-dimensional array. See [acc_get_mda_word_range\(\)](#) for details.

`acc_getmda_bitstr`

Reads a bit string, including X and Z values, from an element in a multi-dimensional array. See [acc_getmda_bitstr\(\)](#) for details.

`acc_setmda_bitstr`

Writes a bit string, including X and Z values, from an element in a multi-dimensional array.
 See [acc_getmda_bitstr\(\)](#) for details.

tf_mdanodeinfo and tf_imdanodeinfo

You use these routines to access parameter node information from a multi-dimensional array.

Table 92 *tf_mdanodeinfo(), tf_imdanodeinfo()*

tf_mdanodeinfo(), tf_imdanodeinfo()			
Synopsis:	Returns access parameter node information from a multi-dimensional array.		
Syntax:	<code>tf_mdanodeinfo(npayload, mdanodeinfo_p) tf_imdanodeinfo(npayload, mdanodeinfo_p, instance_p)</code>		
	Type	Description	
Returns:	<code>mdanodeinfo_p *</code>	The value of the second argument if successful; 0 if an error occurs	
	Type	Name	Description
Arguments:	<code>int</code>	<code>npayload</code>	Index number of the multi-dimensional array parameter
	<code>struct t_tfmdanodeinfo *</code>	<code>mdanodeinfo_p</code>	Pointer to a variable declared as the <code>t_tfmdanodeinfo</code> structure type
	<code>char *</code>	<code>instance_p</code>	Pointer to a specific instance of a multi-dimensional array
Related routines:	<code>acc_get_mda_rangeacc_get_mda_word_rangeacc_getmda_bitstracc_setmda_bitstr</code>		

Structure `t_tfmdanodeinfo` is defined in the `vcsuser.h` file as follows:

```
typedef struct t_tfmdanodeinfo
{
    short node_type;
    short node_fulltype;
    char *memoryval_p;
    char *node_symbol;
    int node_ngroups;
    int node_vec_size;
    int node_sign;
```

```

int    node_ms_index;
int    node_ls_index;
int    node_mem_size;
int    *node_lhs_element;
int    *node_rhs_element;
int    node_dimension;
int    *node_handle;
int    node_vec_type;
} s_tfmdanodeinfo, *p_tfmdanodeinfo;

```

acc_get_mda_range

The `acc_get_mda_range` routine returns the ranges of a multi-dimensional array.

Table 93 acc_get_mda_range()

<code>acc_get_mda_range()</code>			
Synopsis:	Gets all the ranges of the multi-dimensional array.		
Syntax:	<code>acc_get_mda_range(mdaHandle, size, msb, lsb, dim, plndx, prindex)</code>		
	Type	Description	
Returns:	<code>void</code>		
	Type	Name	Description
Arguments:	handle	mdaHandle	Handle to the multi-dimensional array
	int *	size	Pointer to the size of the multi-dimensional array
	int *	msb	Pointer to the most significant bit of a range
	int *	lsb	Pointer to the least significant bit of a range
	int *	dim	Pointer to the number of dimensions in the multi-dimensional array
	int *	plndx	Pointer to the left index of a range
	int *	prndx	Pointer to the right index of a range
Related routines:	<code>tf_mdanodeinfo</code> and <code>tf_imdanodeinfo</code> <code>acc_get_mda_word_range</code> <code>acc_getmda_bitstr</code> <code>acc_setmda_bitstr</code>		

If you have a multi-dimensional array such as the following:

```
reg [7:0] my_mem[255:0][255:0][31:0];
```

And you call a routine, such as the following:

```
handle hN = acc_handle_by_name(my_mem);
acc_get_mda_range(hN, &size, &msb, &lsb, &dim, &plndx, &prndx);
```

It yields the following result:

```
size = 8;
msb = 7, lsb = 0;
dim = 4;
plndx[] = {255, 255, 31}
prndx[] = {0, 0, 0}
```

acc_get_mda_word_range()

The `acc_get_mda_word_range` routine returns the range of an element in a multi-dimensional array.

Table 94 acc_get_mda_word_range()

<code>acc_get_mda_word_range()</code>			
Synopsis:	Gets the range of an element in a multi-dimensional array.		
Syntax:	<code>acc_get_mda_range(mdaHandle, msb, lsb)</code>		
	Type	Description	
Returns:	<code>void</code>		
	Type	Name	Description
Arguments:	<code>handle</code>	<code>mdaHandle</code>	Handle to the multi-dimensional array
	<code>int *</code>	<code>msb</code>	Pointer to the most significant bit of a range
	<code>int *</code>	<code>lsb</code>	Pointer to the least significant bit of a range
Related routines:	<code>tf_mdanodeinfo</code> and <code>tf_imdanodeinfo</code> <code>acc_get_mda_range</code> <code>acc_getmda_bitstr</code> <code>acc_setmda_bitstr</code>		

If you have a multi-dimensional array such as the following:

```
reg [7:0] my_mem[255:0][255:0][31:0];
```

And you call a routine, such as the following:

```
handle hN = acc_handle_by_name(my_mem);
acc_get_mda_word_range(hN, &left, &right);
```

It yields the following result:

```
left = 7;
right = 0;
```

acc_getmda_bitstr()

You use the `acc_getmda_bitstr` access routine to read a bit string, including x and z values, from a multi-dimensional array.

Table 95 acc_getmda_bitstr()

acc_getmda_bitstr()			
Synopsis:	Gets a bit string from a multi-dimensional array.		
Syntax:	<code>acc_getmda_bitstr(mdaHandle, bitStr, dim, start, len)</code>		
	Type	Description	
Returns:	void		
	Type	Name	Description
Arguments:	handle	mdaHandle	Handle to the multi-dimensional array
	char *	bitStr	Pointer to the bit string
	int *	dim	Pointer to the dimension in the multi-dimensional array
	int *	start	Pointer to the start element in the dimension
	int *	len	Pointer to the length of the string
Related routines:	<code>tf_mdanodeinfo</code> and <code>tf_imdanodeinfoacc_get_mda_rangeacc_get_mda_word_rangeacc_setmda_bitstr</code>		

If you have a multi-dimensional array such as the following:

```
reg [7:0] my_mem[255:0][255:0][31:0];
```

And you call a routine, such as the following:

```
dim[]={5, 5, 10};  

handle hN = acc_handle_by_name(my_mem);  

acc_getmda_bitstr(hN, &bitStr, dim, 3, 3);
```

It yields the following string from *my_mem[5][5][10][3:5]*.

acc_setmda_bitstr()

You use the `acc_setmda_bitstr` access routine to write a bit string, including x and z values, into a multi-dimensional array.

Table 96 acc_setmda_bitstr()

acc_setmda_bitstr()			
Synopsis:	Sets a bit string in a multi-dimensional array.		
Syntax:	<code>acc_setmda_bitstr(mdaHandle, bitStr, dim, start, len)</code>		
	Type	Description	
Returns:	void		
	Type	Name	Description
Arguments:	handle	mdaHandle	Handle to the multi-dimensional array
	char *	bitStr	Pointer to the bit string
	int *	dim	Pointer to the dimension in the multi-dimensional array
	int *	start	Pointer to the start element in the dimension
	int *	len	Pointer to the length of the string
Related routines:	<code>tf_mdanodeinfo</code> and <code>tf_imdanodeinfoacc_get_mda_rangeacc_get_mda_word_rangeacc_getmda_bitstr</code>		

If you have a multi-dimensional array such as the following:

```
reg [7:0] my_mem[255:0][255:0][31:0];
```

And you call a routine, such as the following:

```
dim[]={5, 5, 10};  

bitstr="111";  

handle hN = acc_handle_by_name(my_mem);  

acc_setmda_bitstr(hN, &bitStr, dim, 3, 3);
```

It writes 111 in *my_mem[5][5][10][3:5]*.

Access Routines for Probabilistic Distribution

VCS includes the following API routines that duplicate the behavior of the Verilog system functions for probabilistic distribution:

`vcs_random`

Returns a random number and takes no argument. See [vcs_random](#) for details.

`vcs_random_const_seed`

Returns a random number and takes an integer argument. See [vcs_random_const_seed](#) for details.

`vcs_random_seed`

Returns a random number and takes a pointer to integer argument. See [vcs_random_seed](#) for details.

`vcs_dist_uniform`

Returns random numbers uniformly distributed between parameters. See [vcs_dist_uniform](#) for details.

`vcs_dist_normal`

Returns random numbers with a specified mean and standard deviation. See [vcs_dist_normal](#) for details.

`vcs_dist_exponential`

Returns random numbers where the distribution function is exponential. See [vcs_dist_exponential](#) for details.

`vcs_dist_poisson`

Returns random numbers with a specified mean. See [vcs_dist_poisson](#) for details.

These routines are declared in the `vcs_acc_user.h` file in the `$VCS_HOME/lib` directory.

vcs_random

You use this routine to obtain a random number.

Table 97 vcs_random()

vcs_random()			
Synopsis:	Returns a random number.		
Syntax:	<code>vcs_random()</code>		
	Type	Description	
Returns:	int	Random number	
	Type	Name	Description
Arguments:	None		
Related routines:	<code>vcs_random_const_seed</code> <code>vcs_random_seed</code> <code>vcs_dist_uniform</code> <code>vcs_dist_normal</code> <code>vcs_dist_exponential</code> <code>vcs_dist_poisson</code>		

vcs_random_const_seed

You use this routine to return a random number and you supply an integer constant argument as the seed for the random number.

Table 98 vcs_random_const_seed

vcs_random_const_seed			
Synopsis:	Returns a random number.		
Syntax:	<code>vcs_random_const_seed(integer)</code>		
	Type	Description	
Returns:	int	Random number	
	Type	Name	Description
Arguments:	int	integer	An integer constant.

Table 98 vcs_random_const_seed (Continued)

Related routines:	vcs_random vcs_random_seed vcs_dist_uniform vcs_dist_normal vcs_dist_exponential vcs_dist_poisson
-------------------	------------------------------------------------------------------------------------------------------

vcs_random_seed

You use this routine to return a random number and you supply a pointer argument

Table 99 vcs_random_seed()

vcs_random_seed()		
Synopsis:	Returns a random number.	
Syntax:	vcs_random_seed(seed)	
Type Description		
Returns:	int	Random number
Type Name Description		
Arguments:	int *	seed
Pointer to an int type.		
Related routines:	vcs_random vcs_random_const_seed vcs_dist_uniform vcs_dist_normal vcs_dist_exponential vcs_dist_poisson	

vcs_dist_uniform

You use this routine to return a random number uniformly distributed between parameters.

Table 100 vcs_dist_uniform

vcs_dist_uniform		
Synopsis:	Returns random numbers uniformly distributed between parameters.	
Syntax:	vcs_dist_uniform(seed, start, end)	
Type Description		
Returns:	int	Random number

Table 100 vcs_dist_uniform (Continued)

	Type	Name	Description
Arguments:	int *	seed	Pointer to a seed integer value.
	int	start	Starting parameter for distribution range.
	int	end	Ending parameter for distribution range.
Related routines:	vcs_random vcs_random_const_seed vcs_random_seed vcs_dist_normal vcs_dist_exponential vcs_dist_poisson		

vcs_dist_normal

You use this routine to return a random number with a specified mean and standard deviation.

Table 101 vcs_dist_normal

vcs_dist_normal			
Synopsis:	Returns random numbers with a specified mean and standard deviation.		
Syntax:	vcs_dist_normal(seed, mean, standard_deviation)		
	Type	Description	
Returns:	int	Random number	
	Type	Name	Description
Arguments:	int *	seed	Pointer to a seed integer value.
	int	mean	An integer that is the average value of the possible returned random numbers.
	int	standard_deviation	An integer that is the standard deviation from the mean for the normal distribution.
Related routines:	vcs_random vcs_random_const_seed vcs_random_seed vcs_dist_uniform vcs_dist_exponential vcs_dist_poisson		

vcs_dist_exponential

You use this routine to return a random number where the distribution function is exponential.

Table 102 vcs_dist_exponential

vcs_dist_exponential		
Synopsis:	Returns random numbers where the distribution function is exponential.	
Syntax:	<code>vcs_dist_exponential(seed, mean)</code>	
Type Description		
Returns:	int	Random number
Type Name Description		
Arguments:	int *	seed Pointer to a seed integer value.
	int	mean An integer that is the average value of the possible returned random numbers.
Related routines:	<code>vcs_random</code> <code>vcs_random_const_seed</code> <code>vcs_random_seed</code> <code>vcs_dist_uniform</code> <code>vcs_dist_normal</code> <code>vcs_dist_poisson</code>	

vcs_dist_poisson

You use this routine to return a random number with a specified mean.

Table 103 vcs_dist_poisson

vcs_dist_poisson		
Synopsis:	Returns random numbers with a specified mean.	
Syntax:	<code>vcs_dist_poisson(seed, mean)</code>	
Type Description		
Returns:	int	Random number
Type Name Description		
Arguments:	int *	seed Pointer to a seed integer value.

Table 103 vcs_dist_poisson (Continued)

	int	mean	An integer that is the average value of the possible returned random numbers.
Related routines:	vcs_random vcs_random_const_seed vcs_random_seed vcs_dist_uniform vcs_dist_normal vcs_dist_exponential		

Access Routines for Returning a Pointer to a Parameter Value

The 1364 Verilog standard states that for access routine `acc_fetch_paramval`, you can cast the return value to a character pointer using the C language cast operators (`char*`) (`int`). For example:

```
str_ptr=(char*) (int)acc_fetch_paramval(...);
```

In 64-bit simulation, you should use `long` instead of `int`:

```
str_ptr=(char*) (long)acc_fetch_paramval(...);
```

For your convenience, VCS provides the `acc_fetch_paramval_str` routine to directly return a string pointer.

acc_fetch_paramval_str

Returns the value of a string parameter directly as `char*`.

Table 104 acc_fetch_paramval_str

acc_fetch_paramval_str			
Synopsis:	Returns the value of a string parameter directly as <code>char*</code> .		
Syntax:	<code>acc_fetch_paramval_str(param_handle)</code>		
Type	Type	Description	
Returns:	<code>char*</code>	string pointer	
Type	Name	Description	
Arguments:	<code>handle</code>	<code>param_handle</code>	Handle to a module parameter or specparam.
Related routines:	<code>acc_fetch_paramval</code>		

Access Routines for Line Callbacks

VCS includes a number of access routines to monitor code execution. These access routines are as follows:

`acc_mod_lcb_add`

Registers a line callback routine with a module so that VCS calls the routine whenever VCS executes the specified module. See [acc_mod_lcb_add](#) for details.

`acc_mod_lcb_del`

Unregisters a line callback routine previously registered with the `acc_mod_lcb_add()` routine. See [acc_mod_lcb_del](#) for details.

`acc_mod_lcb_enabled`

Tests to see if line callbacks is enabled. See [acc_mod_lcb_enabled](#) for details.

`acc_mod_lcb_fetch`

Returns an array of breakable lines. See [acc_mod_lcb_fetch](#) for details.

`acc_mod_lcb_fetch2`

Returns an array of breakable lines. See [acc_mod_lcb_fetch2](#) for details.

`acc_mod_sfi_fetch`

Returns the source file composition for a module. See [acc_mod_sfi_fetch](#) for details.

acc_mod_lcb_add

Syntax

```
void acc_mod_lcb_add(handle handleModule,
                      void (*consumer)(), char *user_data)
```

Synopsis

Registers a line callback routine with a module so that VCS calls the routine whenever VCS executes the specified module.

The prototype for the callback routine is:

```
void consumer(char *filename, int lineno, char *user_data,
              int tag)
```

The tag field is a unique identifier that you use to distinguish between multiple 'include files.

Protected modules cannot be registered for callback. This routine will just ignore the request.

Returns

No return value.

Example 272 Example of acc_mod_lcb_add

```
#include <stdio.h>
#include "acc_user.h"
#include "vcs_acc_user.h"

/* VCS callback rtn */
void line_call_back(filename, lineno, userdata, tag)
char *filename;
int lineno;
char *userdata;
int tag;
{
    handle handle_mod = (handle)userdata;

    io_printf("Tag %2d, file %s, line %2d, module %s\n",
              tag, filename, lineno,
              acc_fetch_fullname(handle_mod));
}

/* register all modules for line callback (recursive) */
void register_lcb (parent_mod)
handle parent_mod;
{
    handle child = NULL;

    if (! acc_object_of_type(parent_mod, accModule)) return;

    io_printf("Registering %s\n",
              acc_fetch_fullname (parent_mod));

    acc_mod_lcb_add (parent_mod, line_call_back, parent_mod);

    while ((child = acc_next_child (parent_mod, child))) {
        register_lcb (child);
    }
}
```

acc_mod_lcb_del

Syntax

```
void acc_mod_lcb_del(handle handleModule,
                      void (*consumer)(), char *user_data)
```

Synopsis

Unregisters a line callback routine previously registered with the `acc_mod_lcb_add()` routine.

Returns

No return value.

Example 273 Example of acc_mod_lcb_del

```
#include <stdio.h>
#include "acc_user.h"
#include "vcs_acc_user.h"

/* VCS 4.x callback rtn */
void line_call_back(filename, lineno, userdata, tag)
char *filename;
int lineno;
char *userdata;
int tag;
{
    handle handle_mod = (handle)userdata;

    io_printf("Tag %2d, file %s, line %2d, module %s\n",
              tag, filename, lineno,
              acc_fetch_fullname(handle_mod));
}

/* unregister all line callbacks (recursive) */
void unregister_lcb (parent_mod)
handle parent_mod;
{
    handle child = NULL;

    if (! acc_object_of_type(parent_mod, accModule)) return;

    io_printf("Unregistering %s\n",
              acc_fetch_fullname (parent_mod));

    acc_mod_lcb_del (parent_mod, line_call_back, parent_mod);

    while ((child = acc_next_child (parent_mod, child))) {
        register_lcb (child);
    }
}
```

Caution

The module handle, consumer routine, and user data arguments must match those supplied to the `acc_mod_lcb_add()` routine for a successful delete.

For example, using the result of a call such as `acc_fetch_name()` as the user data will fail, because that routine returns a different pointer each time it is called.

acc_mod_lcb_enabled

Syntax

```
int acc_mod_lcb_enabled()
```

Synopsis

Test to see if line callbacks is enabled.

By default, the extra code required to support line callbacks is not added to a simulation executable. You can use this routine to determine if line callbacks have been enabled.

Returns

Non-zero if line callbacks are enabled; 0 if not enabled.

Example 274 Example of acc_mod_lcb_enabled

```
if (! acc_mod_lcb_enable) {
    tf_warning("Line callbacks not enabled. Please recompile with
    -line.");
}
else {
    acc_mod_lcb_add ( ... );
    ...
}
```

acc_mod_lcb_fetch

Syntax

```
p_location acc_mod_lcb_fetch(handle handleModule)
```

Synopsis

Returns an array of breakable lines.

This routine returns all the lines in a module that you can set breakpoints on.

Returns

The return value is an array of line number, file name pairs. Termination of the array is indicated by a NULL file name field. The calling routine is responsible for freeing the returned array.

```
typedef struct t_location {
    int line_no;
```

```
char *filename;
} s_location, *p_location;
```

Returns NULL if the module has no breakable lines or is source protected.

Example 275 Example of acc_mod_lcb_fetch

```
#include <stdio.h>
#include "acc_user.h"
#include "vcs_acc_user.h"

void ShowLines(handleModule)
handle handleModule;
{
    p_location plocation;

    if ((plocation = acc_mod_lcb_fetch(handleModule)) != NULL) {
        int i;

        io_printf("%s:\n", acc_fetch_fullname(handleModule));

        for (i = 0; plocation[i].filename; i++) {
            io_printf(" [%s:%d]\n",
                      plocation[i].filename,
                      plocation[i].line_no);
        }
        acc_free(plocation);
    }
}
```

acc_mod_lcb_fetch2

Syntax

```
p_location2 acc_mod_lcb_fetch2(handle handleModule)
```

Synopsis

Returns an array of breakable lines.

This routine returns all the lines in a module that you can set breakpoints on.

The tag field is a unique identifier used to distinguish 'include files. For example, in the following Verilog module, the breakable lines in the first 'include of the file sequential.code have a different tag than the breakable lines in the second 'include. (The tag numbers will match the vcs_srcfile_info_t->SourceFileTag field. See the acc_mod_sfi_fetch() routine for details.)

```
module x;
initial begin
    'include sequential.code
    'include sequential.code
```

```
    end
endmodule
```

Returns

The return value is an array of location structures. Termination of the array is indicated by a NULL filename field. The calling routine is responsible for freeing the returned array.

```
typedef struct t_location2 {
    int line_no;
    char *filename;
    int tag;
} s_location2, *p_location2;
```

Returns NULL if the module has no breakable lines or is source protected.

Example 276 Example of acc_mod_lcb_fetch2

```
#include <stdio.h>
#include "acc_user.h"
#include "vcs_acc_user.h"

void ShowLines2(handleModule)
handle handleModule;
{
    p_location2 plocation;

    if ((plocation = acc_mod_lcb_fetch2(handleModule)) != NULL) {
        int i;

        io_printf("%s:\n", acc_fetch_fullname(handleModule));

        for (i = 0; plocation[i].filename; i++) {
            io_printf(" file %s, line %d, tag %d\n",
                     plocation[i].filename,
                     plocation[i].line_no,
                     plocation[i].tag);
        }
        acc_free(plocation);
    }
}
```

acc_mod_sfi_fetch

Syntax

```
vcs_srcfile_info_p acc_mod_sfi_fetch(handle handleModule)
```

Synopsis

Returns the source file composition for a module. This composition is a file name with line numbers, or, if a module definition is in more than one file, it is an array of

`vcs_srcfile_info_s` struct entries specifying all the file names and line numbers for the module definition.

Returns

The returned array is terminated by a NULL `SourceFileName` field. The calling routine is responsible for freeing the returned array.

```
typedef struct vcs_srcfile_info_t {
    char *SourceFileName;
    int SourceFileTag;
    int StartLineNum;
    int EndLineNum;
} vcs_srcfile_info_s, *vcs_srcfile_info_p;
```

Returns NULL if the module is source protected.

Example 277 Example of acc_mod_sfi_fetch

```
#include <stdio.h>
#include "acc_user.h"
#include "vcs_acc_user.h"

void print_info (mod)
handle mod;
{
    vcs_srcfile_info_p infoa;

    io_printf("Source Info for Module %s:\n",
              acc_fetch_fullname(mod));

    if ((infoa = acc_mod_sfi_fetch(mod)) != NULL) {
        int i;
        for (i = 0; infoa[i].SourceFileName != NULL; i++) {
            io_printf("  Tag %2d, StartLine %2d, ",
                      infoa[i].SourceFileTag,
                      infoa[i].StartLineNum);
            io_printf("EndLine %2d, SrcFile %s\n",
                      infoa[i].EndLineNum,
                      infoa[i].SourceFileName);
        }
        acc_free(infoa);
    }
}
```

Access Routines for Source Protection

The `enclib.o` file provides a set of access routines that you can use to create applications which directly produce encrypted Verilog source code. Encrypted code can only be decoded by the VCS compiler. There is no user-accessible decode routine.

Note that both Verilog and SDF code can be protected. VCS knows how to automatically decrypt both.

VCS provides the following routines to monitor the port activity of a device:

`vcsSpClose`

This routine frees the memory allocated by `vcsSpInitialize()`. See [vcsSpClose](#) for details.

`vcsSpEncodeOff`

This routine inserts a trailer section containing the '`endprotected`' compiler directive into the output file. It also toggles the encryption flag to false so that subsequent calls to `vcsSpWriteString()` and `vcsSpWriteChar()` will NOT cause their data to be encrypted. See [vcsSpEncodeOff](#) for details.

`vcsSpEncodeOn`

This routine inserts a trailer section containing the '`protected`' compiler directive into the output file. It also toggles the encryption flag to false so that subsequent calls to `vcsSpWriteString()` and `vcsSpWriteChar()` will have their data encrypted. See [vcsSpEncodeOn](#) for details.

`vcsSpEncoding`

This routine gets the current state of encoding. See [vcsSpEncoding](#) for details.

`vcsSpGetFilePtr`

This routine just returns the value previously passed to the `vcsSpSetFilePtr()` routine. See [vcsSpGetFilePtr](#) for details.

`vcsSpInitialize`

Allocates a source protect object. See [vcsSpInitialize](#) for details.

`vcsSpOvaDecodeLine`

Decrypts one line. See [vcsSpOvaDecodeLine](#) for details.

`vcsSpOvaDisable`

Switches to regular encryption. See [vcsSpOvaDisable](#) for details.

`vcsSpOvaEnable`

Enables the OpenVera assertions (OVA) encryption algorithm. Tells VCS's encrypter to use the OVA IP algorithm. See [vcsSpOvaEnable](#) for details.

`vcsSpSetDisplayMsgFlag`

Sets the DisplayMsg flag. See [vcsSpSetDisplayMsgFlag](#) for details.

`vcsspSetFilePtr`

Specifies the output file stream. See [vcsspSetFilePtr](#) for details.

`vcsspSetLibLicenseCode`

Sets the OEM license code. See [vcsspSetLibLicenseCode](#) for details.

`vcsspSetPliProtectionFlag`

Sets the PLI protection flag. See [vcsspSetPliProtectionFlag](#) for details.

`vcsspWriteChar`

Writes one character to the protected file. See [vcsspWriteChar](#) for details.

`vcsspWriteString`

Writes a character string to the protected file. See [vcsspWriteString](#) for details.

[Example 278](#) outlines the basic use of the source protection routines.

Example 278 Using the Source Protection Routines

```
#include <stdio.h>
#include "enclib.h"
void demo_routine()
{
    char *filename = "protected.vp";
    int write_error = 0;
    vcsspStateID esp;
    FILE *fp;

    /* Initialization */

    if ((fp = fopen(filename, "w")) == NULL) {
        printf("Error: opening file %s\n", filename);
        exit(1);
    }

    if ((esp = vcsspInitialize()) == NULL) {
        printf("Error: Initializing src protection routines.\n");
        printf("          Out Of Memory.\n");
        fclose(fp);
        exit(1);
    }

    vcsspSetFilePtr(esp, fp); /* tell rtns where to write */

    /* Write output */

    write_error += vcsspWriteString(esp,
                                   "This text will *not* be encrypted.\n");
}
```

```

write_error += vcsSpEncodeOn(esp);
write_error += vcsSpWriteString(esp,
                               "This text *will* be encrypted.");
write_error += vcsSpWriteChar(esp, '\n');

write_error += vcsSpEncodeOff(esp);
write_error += vcsSpWriteString(esp,
                               "This text will *not* be encrypted.\n");

/* Clean up */

write_error += fclose(fp);
vcsSpClose(esp);

if (write_error) {
    printf("Error while writing to '%s'\n", filename);
}
}

```

Caution

If you are encrypting SDF or Verilog code that contains include directives, you must switch off encryption (`vcsSpEncodeOff`), output the include directive and then switch encryption back on. This ensures that when the parser begins reading the included file, it is in a known (non-decode) state.

If the file being included has proprietary data it can be encrypted separately. (Don't forget to change the '`include`' compiler directive to point to the new encrypted name.)

vcsSpClose

Syntax

```
void vcsSpClose(vcsSpStateID esp)
```

Synopsis

This routine frees the memory allocated by `vcsSpInitialize()`. Call it when source encryption is finished on the specified stream.

Returns

No return value.

Example 279 Example of vcsSpClose

```
vcsSpStateID esp = vcsSpInitialize();
...
vcsSpClose(esp);
```

vcsSpEncodeOff

Syntax

```
int vcsSpEncodeOff(vcsSpStateID esp)
```

Synopsis

This function performs two operations:

1. It inserts a trailer section that contains some closing information used by the decryption algorithm into the output file. It also inserts the `endprotected compiler directive in the trailer section.
2. It toggles the encryption flag to false so that subsequent calls to `vcsSpWriteString()` and `vcsSpWriteChar()` will NOT cause their data to be encrypted.

Returns

Non-zero if there was an error writing to the output file, 0 if successful.

Example 280 Example of vcsSpEncodeOff

```
vcsSpStateID esp = vcsSpInitialize();
FILE *fp = fopen("protected.file", "w");
int write_error = 0;
if (fp == NULL) exit(1);

vcsSpSetFilePtr(esp, fp);

if (vcsSpWriteString(esp, "This text will *not* be encrypted.

        ++write_error;

if (vcsSpEncodeOn(esp)) ++write_error;
if (vcsSpWriteString(esp, "This text *will* be encrypted.

        ++write_error;

if (vcsSpEncodeOff(esp)) ++write_error;
if (vcsSpWriteString(esp, "This text will *not* be encrypted.

        ++write_error;

fclose(fp);
vcsSpClose(esp);
```

Caution

You must call `vcsSpInitialize()` and `vcsSpSetFilePtr()` before calling this routine.

vcsSpEncodeOn

Syntax

```
int vcsSpEncodeOn(vcsSpStateID esp)
```

Synopsis

This function performs two operations:

1. It inserts a header section which contains the 'protected' compiler directive into the output file. It also inserts some initial header information used by the decryption algorithm.
2. It toggles the encryption flag to true so that subsequent calls to `vcsSpWriteString()` and `vcsSpWriteChar()` will have their data encrypted.

Returns

Non-zero if there was an error writing to the output file, 0 if successful.

Example 281 Example of vcsSpEncodeOn

```
vcsSpStateID esp = vcsSpInitialize();
FILE *fp = fopen("protected.file", "w");
int write_error = 0;

if (fp == NULL) exit(1);

vcsSpSetFilePtr(esp, fp);

if (vcsSpWriteString(esp, "This text will *not* be encrypted.\n"))
    ++write_error;

if (vcsSpEncodeOn(esp)) ++write_error;
if (vcsSpWriteString(esp, "This text *will* be encrypted.\n"))
    ++write_error;

if (vcsSpEncodeOff(esp)) ++write_error;
if (vcsSpWriteString(esp, "This text will *not* be encrypted.\n"))
    ++write_error;
fclose(fp);
vcsSpClose(esp);
```

Caution

You must call `vcsSpInitialize()` and `vcsSpSetFilePtr()` before calling this routine.

vcsSpEncoding

Syntax

```
int vcsSpEncoding(vcsSpStateID esp)
```

Synopsis

Calling `vcsSpEncodeOn()` and `vcsSpEncodeOff()` turns encoding on and off. Use this function to get the current state of encoding.

Returns

1 for on, 0 for off.

Example 282 Example of vcsSpEncoding

```
vcsSpStateID esp = vcsSpInitialize();
FILE *fp = fopen("protected.file", "w");

if (fp == NULL) { printf("ERROR: file ..."); exit(1); }

vcsSpSetFilePtr(esp, fp);
...

if (! vcsSpEncoding(esp))
    vcsSpEncodeOn(esp)
...
if (vcsSpEncoding(esp))
    vcsSpEncodeOff(esp);

fclose(fp);
vcsSpClose(esp);
```

vcsSpGetFilePtr

Syntax

```
FILE *vcsSpGetFilePtr(vcsSpStateID esp)
```

Synopsis

This routine just returns the value previously passed to the `vcsSpSetFilePtr()` routine.

Returns

File pointer or NULL if not set.

Example 283 Example of vcsSpGetFilePtr

```
vcsSpStateID esp = vcsSpInitialize();
FILE *fp = fopen("protected.file", "w");
if (fp != NULL)
    vcsSpSetFilePtr(esp, fp);
else
    /* doh! */

...
if ((gfp = vcsSpGetFilePtr(esp)) != NULL) {
    /* Add comment before starting encryption */
    fprintf(gfp, "\n// TechStuff Version 2.2\n");
    vcsSpEncodeOn(esp);
}
```

Caution

Don't use non-vcsSp* routines (like `fprintf`) in conjunction with vcsSp* routines, while encoding is enabled.

vcsSpInitialize

Syntax

```
vcsSpStateID vcsSpInitialize(void)
```

Synopsis

This routine allocates a source protect object.

Returns a handle to a malloc'd object which must be passed to all the other source protection routines.

This object stores the state of the encryption in progress. When the encryption is complete, this object should be passed to `vcsSpClose()` to free the allocated memory.

If you need to write to multiple streams at the same time (perhaps you're creating include or SDF files in parallel with model files), you can make multiple calls to this routine and assign a different file pointer to each handle returned.

Each call mallocs less than 100 bytes of memory.

Returns

The `vcsSpStateID` pointer or NULL if memory could not be malloc'd.

Example 284 Example of vcsSpStateID

```
vcsSpStateID esp = vcsSpInitialize();
if (esp == NULL) {
```

```

        fprintf(stderr, "out of memory\n");
        ...
    }

```

Caution

This routine must be called before any other source protection routine.

A NULL return value means the call to `malloc()` failed. Your program should test for this.

vcsSpOvaDecodeLine

Syntax

```
vcsspstateid vcsSpOvaDecodeLine(vcsspstateid esp, char *line)
```

Synopsis

This routine decrypts one line.

Use this routine to decrypt one line of protected IP code such as OVA code. Pass in a null `vcsspstateid` handle with the first line of code and a non-null handle with subsequent lines.

Returns

Returns NULL when the last line has been decrypted.

Example 285 Example of vcsSpOvaDecodeLine

```
#include "enclib.h"

if (strcmp(linebuf, "'protected_ip synopsys\n") == 0) {
    /* start IP decryption */
    vcsspstateid esp = NULL;
    while (fgets(linebuf, sizeof(linebuf), infile)) {
        /* linebuf contains encrypted source */
        esp = vcsSpOvaDecodeLine(esp, linebuf);
        if (linebuf[0]) {
            /* linebuf contains decrypted source */
            ...
        }
        if (!esp) break; /* done */
    }
    /* next line should be 'endprotected_ip */
    fgets(linebuf, sizeof(linebuf), infile);
    if (strcmp(linebuf, "'endprotected_ip\n") != 0) {
        printf("warning - expected 'endprotected_ip\n");
    }
}
```

vcsSpOvaDisable

Syntax

```
void vcsSpOvaDisable(vcsSpStateID esp)
```

Synopsis

This routine switches to regular encryption. It tells VCS's encrypter to use the standard algorithm. This is the default mode.

Returns

No return value.

Example 286 Example of vcsSpOvaDisable

```
#include "enclib.h"
#include "encint.h"

int write_error = 0;
vcsSpStateID esp;

if ((esp = vcsSpInitialize()) == NULL) printf("Out Of Memory");

vcsSpSetFilePtr(esp, fp); /* previously opened FILE* pointer */

/* Configure for OVA IP encryption */
vcsSpOvaEnable(esp, "synopsys");

if (vcsSpWriteString(esp, "This text will NOT be encrypted.\n"))
    ++write_error;

if (vcsSpEncodeOn(esp)) ++write_error;

if (vcsSpWriteString(esp, "This text WILL be encrypted.\n"))
    ++write_error;

if (vcsSpEncodeOff(esp)) ++write_error;

if (vcsSpWriteString(esp, "This text will NOT be encrypted.\n"))
    ++write_error;
/* Switch back to regular encryption */
vcsSpOvaDisable(esp);

if (vcsSpEncodeOn(esp)) ++write_error;

if (vcsSpWriteString(esp, "This text WILL be encrypted.\n"))
    ++write_error;

if (vcsSpEncodeOff(esp)) ++write_error;
```

```
vcsspClose(esp);
```

vcsSpOvaEnable

Syntax

```
void vcsSpOvaEnable(vcsSpStateID esp, char *vendor_id)
```

Synopsis

Enables the OpenVera assertions (OVA) encryption algorithm. Tells VCS's encrypter to use the OVA IP algorithm.

Returns

No return value.

Example 287 Example of vcsSpOvaEnable

```
#include "enclib.h"
#include "encint.h"

int write_error = 0;
vcsSpStateID esp;

if ((esp = vcsSpInitialize()) == NULL) printf("Out Of Memory");

vcsSpSetFilePtr(esp, fp); /* previously opened FILE* pointer */

/* Configure for OVA IP encryption */
vcsSpOvaEnable(esp, "synopsys");

if (vcsSpWriteString(esp, "This text will NOT be encrypted.\n"))
    ++write_error;

if (vcsSpEncodeOn(esp)) ++write_error;

if (vcsSpWriteString(esp, "This text WILL be encrypted.\n"))
    ++write_error;

if (vcsSpEncodeOff(esp)) ++write_error;

if (vcsSpWriteString(esp, "This text will NOT be encrypted.\n"))
    ++write_error;
/* Switch back to regular encryption */
vcsSpOvaDisable(esp);

if (vcsSpEncodeOn(esp)) ++write_error;

if (vcsSpWriteString(esp, "This text WILL be encrypted.\n"))
    ++write_error;
```

```
if (vcsSpEncodeOff(esp)) ++write_error;
vcsSpClose(esp);
```

vcsSpSetDisplayMsgFlag

Syntax

```
void vcsSpSetDisplayMsgFlag(vcsSpStateID esp, int enable)
```

Synopsis

This routine sets the DisplayMsg flag. By default, the VCS compiler does not display decrypted source code in its error or warning messages. Use this routine to enable this display.

Returns

No return value.

Example 288 Example of vcsSpSetDisplayMsgFlag

```
vcsSpStateID esp = vcsSpInitialize();
vcsSpSetDisplayMsgFlag(esp, 0);
```

vcsSpSetFilePtr

Syntax

```
void vcsSpSetFilePtr(vcsSpStateID esp, FILE *fp)
```

Synopsis

This routine specifies the output file stream. Before using the `vcsSpWriteChar()` or `vcsSpWriteString()` routines, you must specify the output file stream.

Returns

No return value.

Example 289 Example of vcsSpSetFilePtr

```
vcsSpStateID esp = vcsSpInitialize();
FILE *fp = fopen("protected.file", "w");
if (fp != NULL)
    vcsSpSetFilePtr(esp, fp);
else
    /* abort */
```

vcsSpSetLibLicenseCode

Syntax

```
void vcsSpSetLibLicenseCode(vcsSpStateID esp, unsigned int code)
```

Synopsis

This routine sets the OEM library license code that will be added to each protected region started by `vcsSpEncodeOn()`.

This code can be used to protect library models from unauthorized use.

When the VCS parser decrypts the protected region, it verifies that the end user has the specified license. If the license does not exist or has expired, VCS exits.

Returns

No return value.

Example 290 Example of vcsSpSetLibLicenseCode

```
unsigned int lic_code = MY_LICENSE_CODE;
vcsSpStateID esp = vcsSpInitialize();
...
/* The following text will be encrypted and licensed */
vcsSpSetLibLicenseCode(esp, code); /* set license code */
vcsSpEncodeOn(esp); /* start protected region */
vcsSpWriteString(esp, "this text will be encrypted and licensed");
vcsSpEncodeOff(esp); /* end protected region */

/* The following text will be encrypted but unlicensed */
vcsSpSetLibLicenseCode(esp, 0); /* clear license code */
vcsSpEncodeOn(esp); /* start protected region */
vcsSpWriteString(esp, "this text encrypted but not licensed");
vcsSpEncodeOff(esp); /* end protected region */
```

Caution

The rules for mixing licensed and unlicensed code is determined by your OEM licensing agreement with Synopsys.

The code segment in [Example 290](#) shows how to enable and disable the addition of the license code to the protected regions. Normally you would call this routine once, that is, after calling `vcsSpInitialize()` and before the first call to `vcsSpEncodeOn()`.

vcsSpSetPliProtectionFlag

Syntax

```
void vcsSpSetPliProtectionFlag(vcsSpStateID esp, int enable)
```

Synopsis

This routine sets the PLI protection flag. You can use it to disable the normal PLI protection that is placed on encrypted modules. The output files will still be encrypted, but CLI and PLI users will not be prevented from accessing data in the modules.

This routine only affects encrypted Verilog files. Encrypted SDF files, for example, are not affected.

Returns

No return value.

Example 291 Example of vcsSpSetPliProtectionFlag

```
vcsSpStateID esp = vcsSpInitialize();
vcsSpSetPliProtectionFlag(esp, 0); /* disable PLI protection */
```

Caution

Turning off PLI protection will allow users of your modules to access object names, values, etc. In essence, the source code for your module could be substantially reconstructed using the CLI commands and ACC routines.

vcsSpWriteChar

Syntax

```
void vcsSpSetPliProtectionFlag(vcsSpStateID esp, int enable)
```

Synopsis

This routine writes one character to the protected file.

If encoding is enabled (see [vcsSpEncodeOn](#)) the specified character is encrypted as it is written to the output file.

If encoding is disabled (see [vcsSpEncodeOff](#)) the specified character is written as-is to the output file (no encryption.)

Returns

Non-zero if the file pointer has not been set (see [vcsSpSetFilePtr](#)) or if there was an error writing to the output file (out-of-disk-space, and so on.)

Returns 0 if the write was successful.

Example 292 Example of vcsSpWriteChar

```
vcsSpStateID esp = vcsSpInitialize();
FILE *fp = fopen("protected.file", "w");
int write_error = 0;

if (fp == NULL) exit(1);

vcsSpSetFilePtr(esp, fp);

if (vcsSpWriteChar(esp, 'a')) /* This char will *not* be encrypted.*/
    ++write_error;

if (vcsSpEncodeOn(esp))
    ++write_error;

if (vcsSpWriteChar(esp, 'b')) /* This char *will* be encrypted. */
    ++write_error;
if (vcsSpEncodeOff(esp))
    ++write_error;

fclose(fp);
vcsSpClose(esp);
```

Caution

`vcsSpInitialize()` and `vcsSpSetFilePtr()` must be called prior to calling this routine.

vcsSpWriteString

Syntax

```
int vcsSpWriteString(vcsSpStateID esp, char *s)
```

Synopsis

This routine writes a character string to the protected file.

If encoding is enabled (see [vcsSpEncodeOn](#)) the specified string is encrypted as it is written to the output file.

If encoding is disabled (see [vcsSpEncodeOff](#)) the specified string will be written as-is to the output file (no encryption.)

Returns

Non-zero if the file pointer has not been set (see [vcsSpSetFilePtr](#)) or if there was an error writing to the output file (out-of-disk-space, etc.)

Returns 0 if the write was successful.

Example 293 Example of vcsSpWriteString

```
vcsSpStateID esp = vcsSpInitialize();
FILE *fp = fopen("protected.file", "w");
int write_error = 0;

if (fp == NULL) exit(1);

vcsSpSetFilePtr(esp, fp);

if (vcsSpWriteString(esp, "This text will *not* be encrypted.\n"))
    ++write_error;

if (vcsSpEncodeOn(esp)) ++write_error;
if (vcsSpWriteString(esp, "This text *will* be encrypted.\n"))
    ++write_error;

if (vcsSpEncodeOff(esp)) ++write_error;
if (vcsSpWriteString(esp, "This text will *not* be encrypted.\n"))
    ++write_error;

fclose(fp);
vcsSpClose(esp);
```

Caution

`vcsSpInitialize()` and `vcsSpSetFilePtr()` must be called prior to calling this routine.

Access Routine for Signal in a Generate Block

There is only one access routine for signals in generate blocks.

acc_object_of_type

Syntax

```
bool acc_object_of_type(accGenerated, sigHandle)
```

Synopsis

This routine returns true if the signal is in a generate block.

Returns

- 1 - if the signal is in a generate block.
- 0 - if the signal is not in a generate block.

VCS API Routines

Typically VCS controls the PLI application. If you write your application so that it controls VCS, you need these API routines.

Vcsinit()

When VCS is run in slave mode, you can call this function to elaborate the design and to initialize various data structures, scheduling queues, etc. that VCS uses. After this routine executes, all the initial time 0 events, such as the execution of initial blocks, are scheduled.

Call the `vmc_main(int argc, char *argv)` routine to pass runtime flags to VCS before you call `VcsInit()`.

VcsSimUntil()

This routine tells VCS to schedule a stop event at the specified simulation time and execute all scheduled simulation events until it executes the stop event. The syntax for this routine is as follows:

```
VcsSimUntil (unsigned int* t)
```

Argument `t` is for specifying the simulation time. It needs two words. The first [0] is for simulation times from 0 to 232 -1, the second is for simulation times that follow.

If any events are scheduled to occur after time `t`, their execution must wait for another call to `VcsSimUntil`.

If `t` is less than the current simulation time, VCS returns control to the calling routine.