Product Version ICADVM20.1 October 2020 © 2020 Cadence Design Systems, Inc. All rights reserved.

Portions © Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation. Used by permission.

Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Product AMS contains technology licensed from, and copyrighted by: Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, and other parties and is © 1989-1994 Regents of the University of California, 1984, the Australian National University, 1990-1999 Scriptics Corporation, and other parties. All rights reserved.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522. All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

- 1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
- 2. The publication may not be modified in any way.
- 3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
- 4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor

Contents

<u>1</u>
Setting Up the Migration Example
Features of the Migration Example
Testbench for the PLL Design
Inside the PLL_160MHZ Instance
2
<u>⊆</u> <u>Using the Spectre AMS Designer Simulator</u>
Changing the Simulator to AMS Designer11
Loading the State File for AMS Designer12
Selecting and Customizing Connect Rules for AMS Designer
Setting Netlister and Run Modes 21
<u>Viewing Options</u> 23
Viewing Analog (Spectre) Options
Viewing FastSPICE (UltraSim) Options
Viewing AMS Options
Netlisting and Running
<u> Viewing Waveforms</u>
<u> Displaying Partitions</u>
Understanding Connect Rules and Disciplines in AMS Designer
Simulating the Design Using the Spectre Solver

1

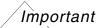
Setting Up the Migration Example

Important

SpectreVerilog and UltraSimVerilog are not available for 64-bit platforms. Hence, if you are using a 64-bit platform, and have set the CDS_AUTO_64BIT environment variable to ALL, unset the environment variable if you are using SpectreVerilog or UltraSimVerilog.

Before the Spectre AMS Designer simulator, the major mixed-signal solutions on the market were SpectreVerilog and UltraSimVerilog. The AMS Designer simulator provides faster simulation speed, increased capacity, and enhanced features for handling more complicated mixed-signal designs with new technologies. More and more chip designers use Spectre AMS Designer for mixed-signal simulation and verification.

This migration example shows you how you would simulate a PLL design using UltraSimVerilog, SpectreVerilog, and, finally, AMS Designer using the same <code>config</code> view. You will learn about the OSS netlister, the <code>ncverilog</code> flow, using <code>-v</code> and <code>-y</code> command-line options, compiled Verilog-A, fastcross, and other AMS Designer features.



Use IC 6.1.3, MMSIM 6.2 or later, and IUS 6.11 or later for this example.

To set up the migration example, do the following in a terminal window:

1. Make and change to a directory for the migration example:

```
mkdir migrateToAMS
cd migrateToAMS
```

2. Copy the migration example files to this directory:

```
cp -r $CDSHOME/tools/dfII/samples/tutorials/AMS/MigrateFromCBNToOSSN.tar.gz .
```

3. Decompress the archive file:

```
gunzip MigrateFromCBNToOSSN.tar.gz
tar xf MigrateFromCBNToOSSN.tar
```

4. Change to the following directory:

Setting Up the Migration Example

cd MigrateFromCBNToOSSN

5. Source the setup file:

source SETUP

The SETUP file sets the TUT_DIR environment variable to your current directory:

setenv TUT DIR `pwd`

6. Start Cadence software:

virtuoso &

See also:

- Features of the Migration Example, next
- Testbench for the PLL Design on page 4
- Inside the PLL_160MHZ Instance on page 5

Features of the Migration Example

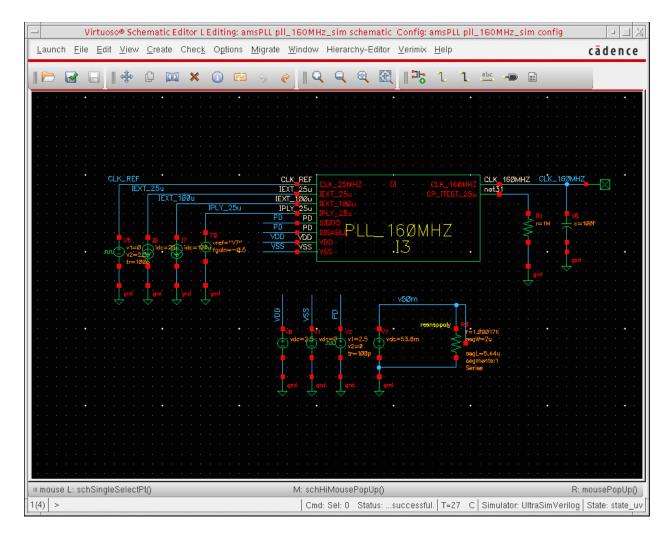
The migration example is a PLL design that has

- a 25 MHz input signal
- a 160 MHz output signal
- 305 MOSFETs
- 97 resistors
- 35 capacitors
- more than 30 behavioral modules

Testbench for the PLL Design

Here is the testbench schematic for the migration example (a PLL design).

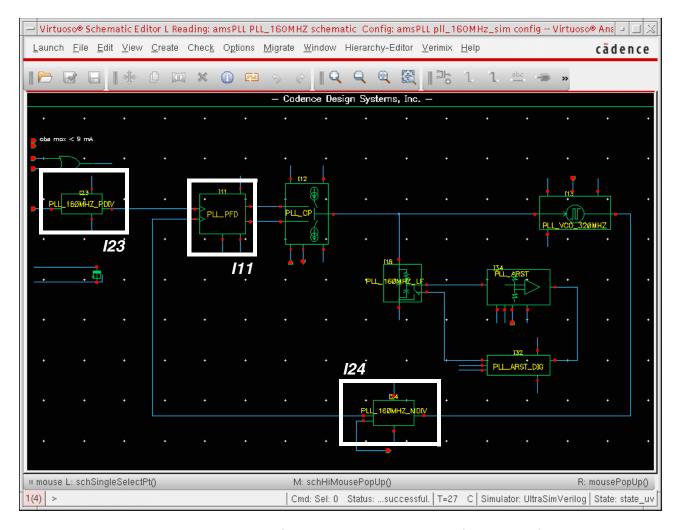
Figure 1-1 Testbench Schematic for PLL Design



Inside the PLL_160MHZ Instance

Inside the *I3* instance (*PLL_160MHZ*), the *I23* instance (*PLL_160MHZ_PDIV*) outputs a 5 MHz reference signal for the loop. The *I24* instance (*PLL_160MHZ_MDIV*) outputs a 160 MHz signal and a 5 MHz feedback signal for the *PLL_FPD* instance (*I11*).

Figure 1-2 Inside the I3 Instance



When the two *PD* input signals to *I3* (see <u>Figure 1-1</u> on page 5) are out of sync, the PFD (*PLL_PFD*) generates corrective pulses (*UP*, *DN*) to adjust the charge pump output voltage (*vCNTL*) which controls the frequency of the VCO (*PLL_VCO_320MHZ*).

Figure 1-3 PFD Corrective Pulses, Charge Pump Output, VCO Input



Whenever the PLL is locked, the FBCLK and $5MHZ_CLK$ signals are in phase and the VCO control signal (vCNTL) is stable.

SpectreVerilog/UltraSimVerilog Migration to AMS Designer Setting Up the Migration Example

2

Using the Spectre AMS Designer Simulator



Use IC 6.1.3, MMSIM 6.2 or later, and IUS 6.11 or later for this example.

Spectre AMS Designer has many advantages:

- Better performance (33% for this particular example)
- More powerful digital solver (NC-Sim versus Verilog-XL)
- Powerful connect rules (CRs)
- Flexible discipline definitions
- Bidirectional CR support
- More language support (Verilog-AMS, VHDL-AMS, SystemVerilog, SystemC)

If you use the Spectre AMS Designer simulator in the Virtuoso Analog Design Environment (ADE), there are two netlisters:

- The cell-based netlister, which is the original netlister for AMS, requires ams simInfo (which contains information such as a parameter list and how to netlist each component) when it generates the individual netlist.vams netlist files in the library/cell/view directory structure. For more information, see "Netlisting" in the Virtuoso AMS Environment User Guide.
- The open simulation system (OSS) netlister is available in IC 5.1.41 USR4 and later. You can use this netlister when you migrate to AMS Designer. The OSS netlister uses existing <code>spectre</code> views. The OSS netlister generates a single netlist file (netlist.vams) that includes all the modules that need to be compiled. (The final netlist is also one file for Spectre and UltraSim.) The OSS netlister works the same way for the AMS Designer simulator as it does for the Spectre and UltraSim simulators.

If you are using the OSS netlister, you can use the same config view to run AMS Designer.

Using the Spectre AMS Designer Simulator

This tutorial illustrates how to use the OSS netlister and neverilog so that you can benefit from the many advantages of AMS Designer. See the following topics for more information:

- Changing the Simulator to AMS Designer on page 11
- Loading the State File for AMS Designer on page 12
- Selecting and Customizing Connect Rules for AMS Designer on page 14
- Setting Netlister and Run Modes on page 21
- Viewing Options on page 23
- Netlisting and Running on page 27
- <u>Viewing Waveforms</u> on page 29
- <u>Displaying Partitions</u> on page 30
- <u>Understanding Connect Rules and Disciplines in AMS Designer</u> on page 32

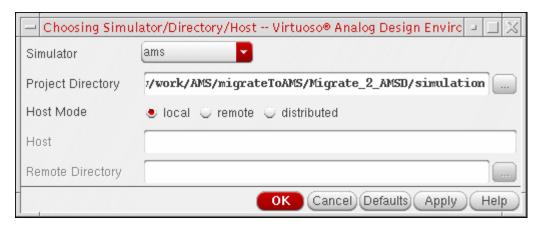
Changing the Simulator to AMS Designer

To change the simulator to AMS Designer, do the following:

1. In the Virtuoso[®] Analog Design Environment session window, choose Setup - Simulator/Directory/Host.

The Choosing Simulator/Directory/Host form appears.

2. In the Simulator drop-down combo box, select ams.



3. Click OK.

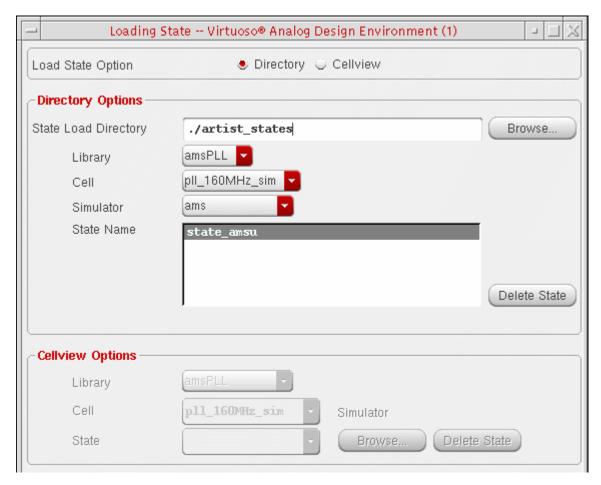
Simulator: ams appears on the status bar in the Virtuoso[®] Analog Design Environment session window. The name of the analog solver (*Spectre* or *UltraSim*) appears in parentheses after *ams*.



Loading the State File for AMS Designer

To load the state file for AMS Designer, do the following:

In the Virtuoso[®] Analog Design Environment window, choose Session – Load State.
 The Loading State form appears.



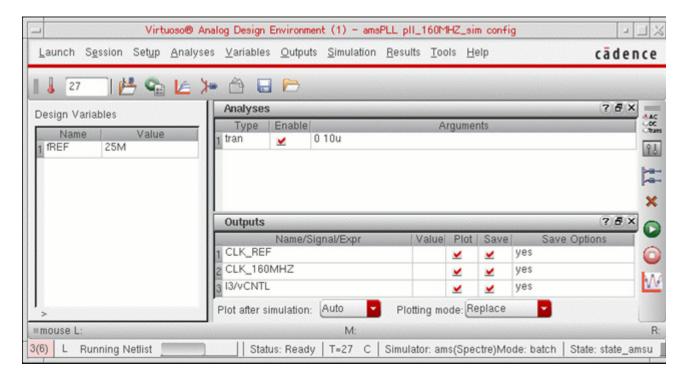
2. In the *State Name* area, select *state_amsu*.

This state uses the UltraSim solver.

3. Click OK.

Using the Spectre AMS Designer Simulator

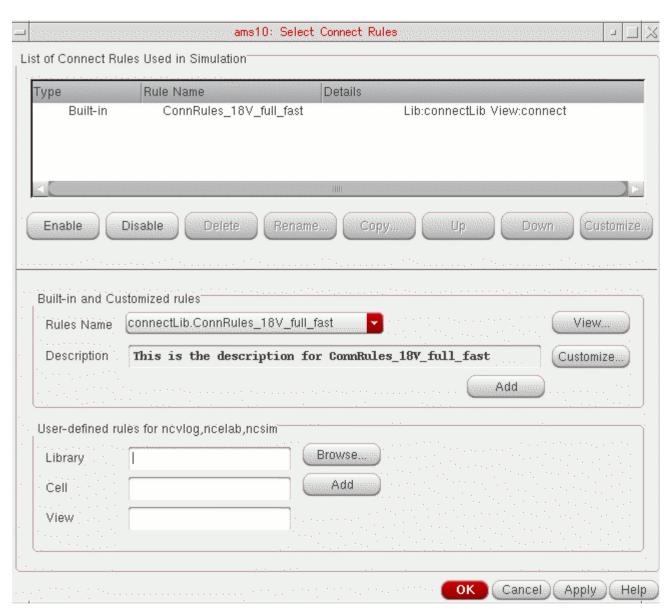
The state settings appear in the Virtuoso[®] Analog Design Environment session window, such as *tran ... 10u* in the *Analyses* area and nodes to plot in the *Outputs* area. *Simulator: ams(UltraSim)* appears on the status bar in the ADE window.



Selecting and Customizing Connect Rules for AMS Designer

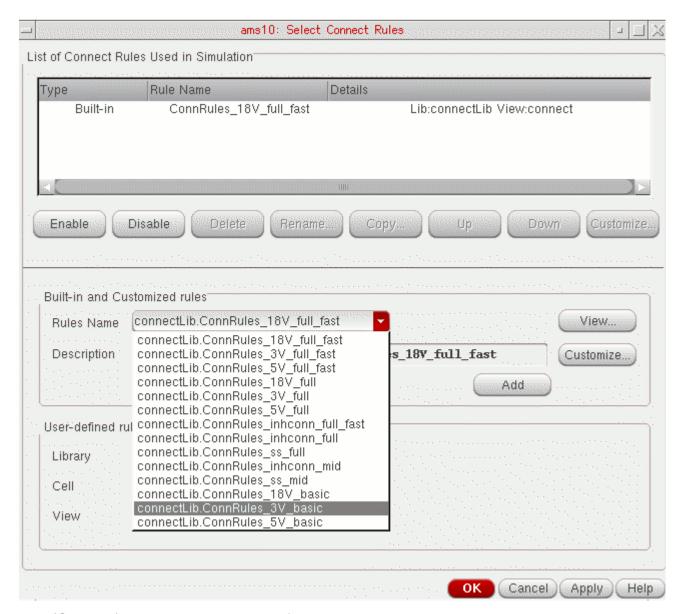
To specify and customize connect rules for the AMS Designer simulator, do the following:

In the Virtuoso[®] Analog Design Environment window, choose Setup – Connect Rules.
 The Select Connect Rules form appears.



Using the Spectre AMS Designer Simulator

2. In the *Rules Name* drop-down combo box, select *connectLib.ConnRules_3V_basic*.



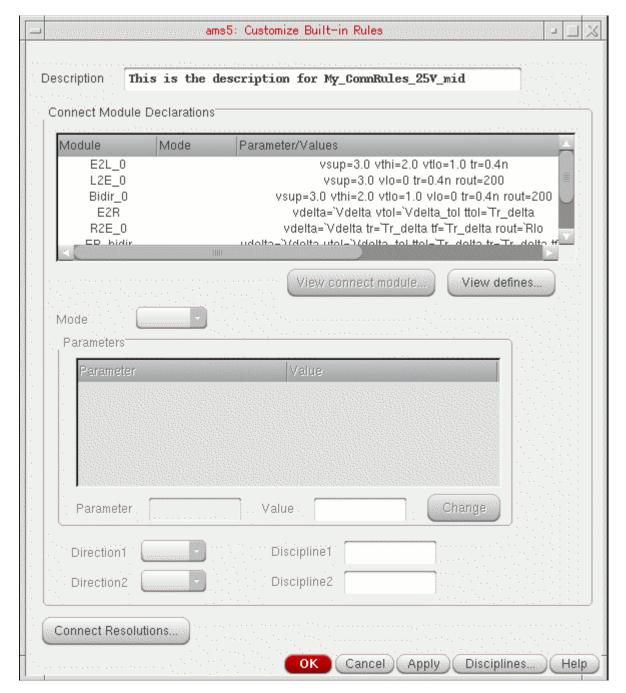
3. (Optional) To view the contents of the connect rule, click *View*.

The connect rule file appears in a window. When you are finished viewing the file, you can choose *File - Close Window*.

4. To customize this connect rule, click *Customize*.

Using the Spectre AMS Designer Simulator

The Customize Built-in Rules form appears.



5. In the *Description* field, change the name of the rule to My_ConnRules_25V_mid:

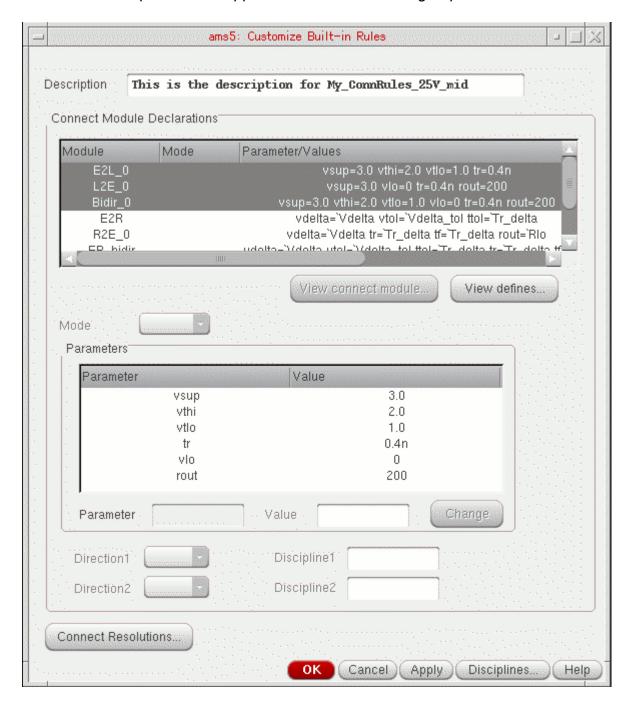
This is the description for $My_ConnRules_25V_mid$

Description This is the description for My_ConnRules_25V_mid

Using the Spectre AMS Designer Simulator

6. In the *Connect Module Declarations* group box, hilight the top three lines containing information for modules *E2L_0*, *L2E_0*, and *Bidir_0*.

The shared parameters appear in the *Parameters* group box.



Using the Spectre AMS Designer Simulator

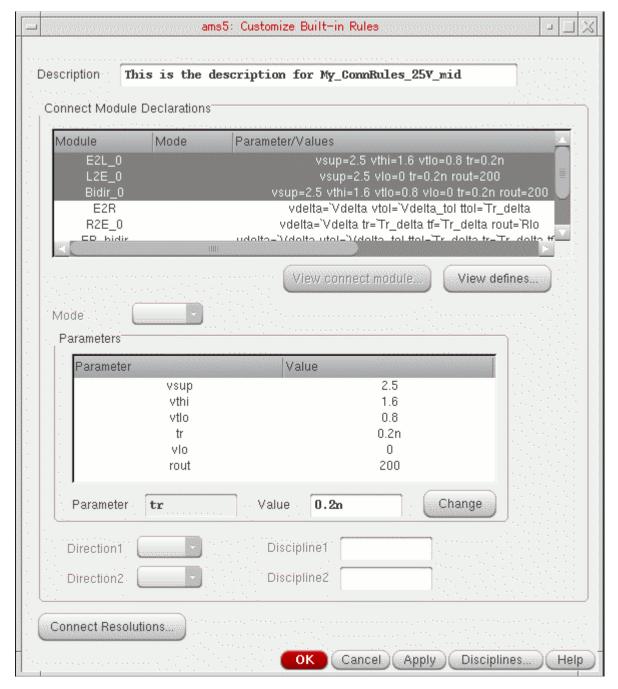
7. Change these values as follows:

Parameter	Value	Change it to
vsup	3.0	2.5
vthi	2.0	1.6
vtlo	1.0	0.8
tr	0.4n	0.2n

- **a.** Select the parameter you want to change.
- **b.** In the *Value* field, change its value.

Using the Spectre AMS Designer Simulator

c. Click Change.



8. Click *OK*.

Using the Spectre AMS Designer Simulator

9. On the Select Connect Rules form, click *Add*.



Modified built-in appears in the *Type* column.

10. Click OK.

The connect rules you specify on the Select Connect Rules form apply to the whole design.

Note: You might want to have several connect rules in the same design. You can set disciplines on a net, cell, instance, or library, and you can specify several connect rules accordingly.

Setting Netlister and Run Modes

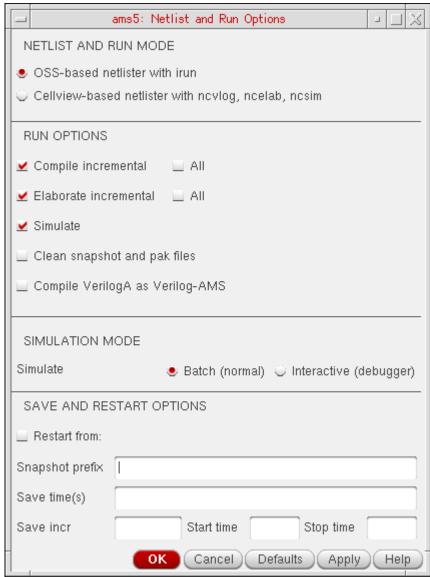
To set netlister and run modes, do the following:

1. In the Virtuoso[®] Analog Design Environment window, choose *Simulation – Netlist and Run Options*.

The Netlister and Run Options form appears.

2. For Netlister Mode, select OSS-based.

The *ncverilog* radio button appears as a *Run Mode* choice.



Click OK.

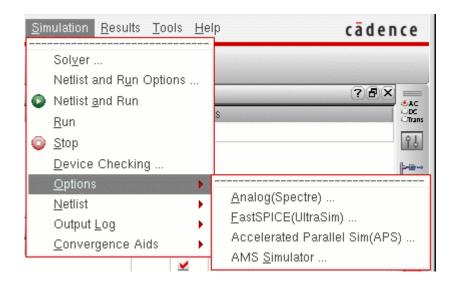
Using the Spectre AMS Designer Simulator

You are ready to simulate.

Using the Spectre AMS Designer Simulator

Viewing Options

As you proceed through this next set of steps, you will notice several choices on the *Simulation – Options* menu in the Virtuoso[®] Analog Design Environment window:



You will not change any of these options during this example, but you will view some of the forms.

See

- Viewing Analog (Spectre) Options on page 24
- <u>Viewing FastSPICE (UltraSim) Options</u> on page 25
- Viewing AMS Options on page 26

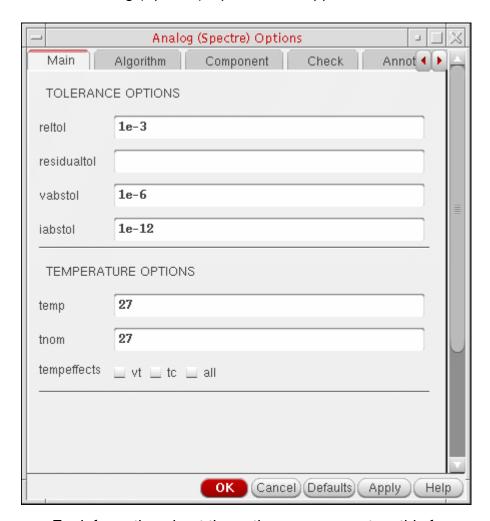
Using the Spectre AMS Designer Simulator

Viewing Analog (Spectre) Options

To view Analog (Spectre) options, do the following:

1. In the Virtuoso[®] Analog Design Environment window, choose *Simulation – Options – Analog(Spectre)*.

The Analog (Spectre) Options form appears.



For information about the options you can set on this form, see the *Spectre Circuit Simulator and Accelerated Parallel Simulator User Guide*.

2. When you are finished viewing Spectre options, click Cancel to close the form.

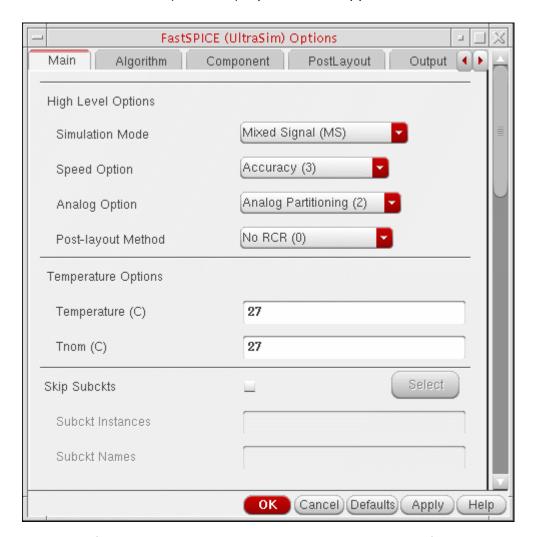
Using the Spectre AMS Designer Simulator

Viewing FastSPICE (UltraSim) Options

To view FastSPICE (UltraSim) options, do the following:

1. In the Virtuoso[®] Analog Design Environment window, choose *Simulation – Options – FastSPICE(UltraSim)*.

The FastSPICE (UltraSim) Options form appears.



For information about the options you can set on this form, see the *Virtuoso UltraSim Simulator User Guide*.

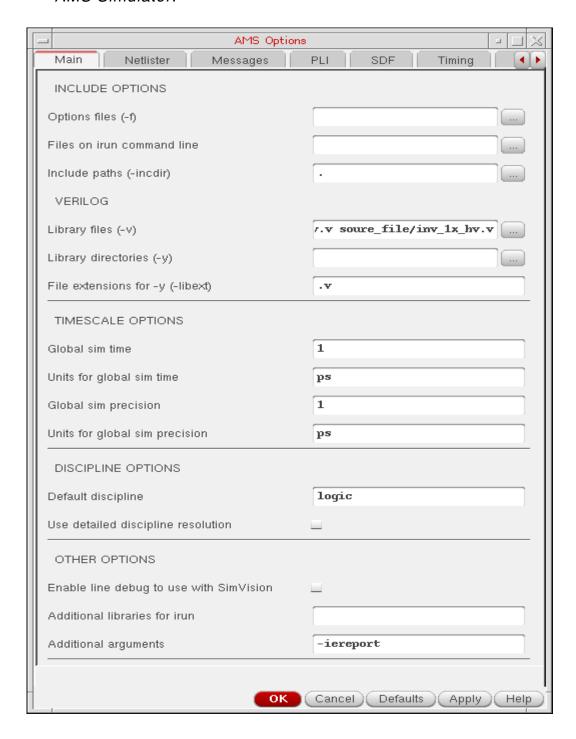
2. When you are finished viewing FastSPICE options, click *Cancel* to close the form.

Using the Spectre AMS Designer Simulator

Viewing AMS Options

To view AMS simulation options, do the following:

1. In the Virtuoso[®] Analog Design Environment window, choose *Simulation – Options – AMS Simulator*.



Using the Spectre AMS Designer Simulator

2. Scroll down to the bottom of this form to see that -iereport appears in the *Additional* arguments field.

When you specify the -iereport option, the elaborator generates an interface element (IE) report. The IE report appears at the top of the simulation log file. This report contains information about each IE the software inserted into the design, such as its name, net, discipline, and so on.

3. When you are finished viewing options, click *Cancel* to close the form.

Netlisting and Running

To netlist and run, do the following:

➤ In the Virtuoso[®] Analog Design Environment window, choose Simulation – Netlist and Run.

Status appears in the upper left corner of the window. Simulation output information appears in the ncverilog.log file. The simulation time appears at the end of the file:

```
Time Usage:
  Total user time: 0:04:16 (256.980 sec), system time: 0:00:01 (1.100 sec),
real time: 0:04:26 (266.600 sec)
```

Note: This simulation ran for 5 minutes 47 seconds on our Solaris machine with a 1.6 G CPU.

Because *-iereport* appears in the *Additional arguments* field on the Main tab of the AMS tabbed window (see "Viewing AMS Options" on page 26), an IE report appears at the top of the log file. That report might look something like this:

```
Automatically inserted instance: pll_160MHz_sim.I3.I11.I15.net18__E2L__logic (merged):

connectmodule name: E2L,
inserted across signal: net18
and ports of discipline: logic
Sensitivity infomation:
No Sensitivity info
Discipline of Port (Ain): electrical, Analog port
Discipline of Port (Dout): logic, Digital port
Drivers of port Dout: No drivers
Loads of port Dout: No loads
```

As you scroll down past UltraSim version and build time information, you will notice messages related to the compiled C flow for Verilog-A. Those messages might look something like this:

```
File read: .../Migrate_2_AMSD/models/spectre/resd_va.va Created directory amsControl.ahdlSimDB/ (775) Created directory
```

Using the Spectre AMS Designer Simulator

You can use the compiled C flow to boost performance particularly when you are using Verilog-A to model becours or CMOS devices such as MOSFETs, resistors, and capacitors. See "Using the Compiled C Code Flow" in the *Cadence Verilog-A Language Reference* for more information.

The IUS 5.83 release supports a feature called FastCross that speeds up the simulation by reducing the number of global time steps. Information about the total global time steps appears near the end of the log file:

```
Total Accepts: 1.452 M
```

You can close each window by choosing *File - Close Window*.

Using the Spectre AMS Designer Simulator

Viewing Waveforms

When the simulation finishes, a graph window appears in Virtuoso Visualization and Analysis XL. For derails on using this waveform tool, see the <u>Virtuoso Visualization and Analysis XL User Guide</u>.

Using the Spectre AMS Designer Simulator

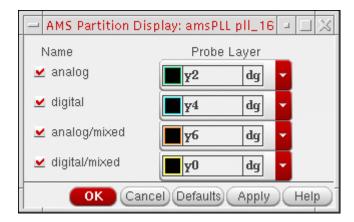
Displaying Partitions

Note: You can view partitions only after you have completed simulation and elaboration.

You can verify the partitions for AMS Designer:

- **1.** Make the *AMS* menu available in the menu bar. For this, choose *Launch Plug-ins Mixed Signal Options AMS*.
- **2.** In the schematic window, descend into *13*:
 - a. Select I3.
 - **b.** Type e.
 - c. Click OK.
- **3.** In the schematic window, choose *AMS Display Partition Initialize*.
- **4.** In the schematic window, choose *AMS Display Partition Interactive*.

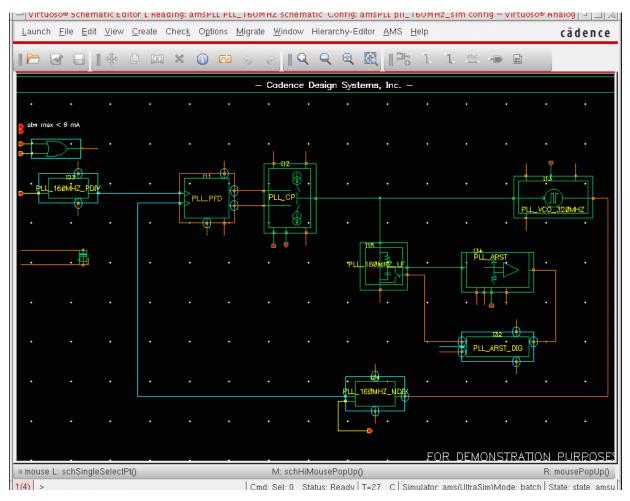
The AMS Partition Display form appears.



5. Click OK.

Using the Spectre AMS Designer Simulator

On the schematic, mixed-signal items appear in orange and yellow.



You can see the set up for connect rules and disciplines both from the AMS menu in the schematic window and by choosing $Setup-Connect\ Rules$ in the Virtuoso[®] Analog Design Environment window.

Using the Spectre AMS Designer Simulator

Understanding Connect Rules and Disciplines in AMS Designer

The AMS Designer simulator uses disciplines, connect modules, and connect rules in place of A2D and D2A interface elements. A discipline denotes an object as analog or digital (with, for example, an electrical or logic discipline). When you connect objects of different disciplines, connect rules determine which connect modules to insert between the objects. The inserted connect modules convert signals to values that are appropriate for each discipline. You can modify connect rule parameters such as supply voltage and rise time in your connect modules to tailor conversion of your design.

Note: For more information about disciplines, connect rules, and connect modules, see "Mixed-Signal Aspects of Verilog-AMS" in the *Cadence Verilog-AMS Language Reference*.

Cadence provides sample connect rules in the following directory:

\$AMSHOME/tools/affirma ams/etc/connect lib

The sample connect rules (CRs) here are built in and ready for use in the Virtuoso[®] Analog Design Environment (ADE). Built-in CRs work for a certain set of voltage supplies only (such as 1.8V, 3V, and 5V). You can modify the parameters to customize a built-in CR for your design needs. Advanced designers can write customized CRs and include them in the simulation.

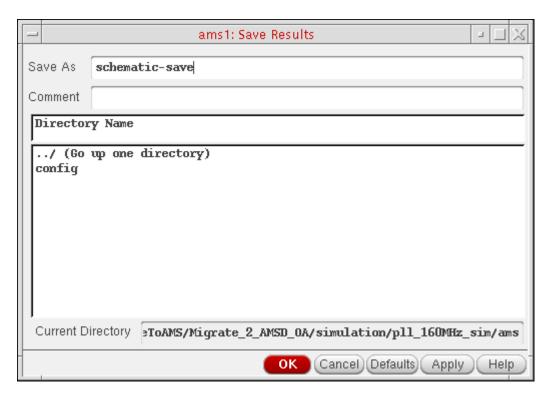
For this example, the voltage supply is 2.5 V. We can customize the 3V built-in CR to fit our simulation.

Note: Cadence provides full-fast, full, mid, and basic built-in CRs. You can speed up the simulation for complicated designs using the full-fast CRs. For this example, we do not need a bidirectional CR; we can choose a simpler CR.

Simulating the Design Using the Spectre Solver

The AMS Designer simulator has two analog solvers: UltraSim and Spectre. To simulate the example design using the Spectre solver, you can do the following:

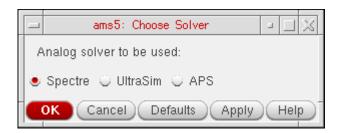
- 1. Save the results from the previous simulation as follows:
 - a. In the Virtuoso[®] Analog Design Environment window, choose Results Save.
 The Save Results form appears.



- **b.** In the *Save As* field, type a name for your results.
- c. Click OK.
- **2.** Choose Simulation Solver.

Using the Spectre AMS Designer Simulator

The Choose Solver form appears.



- 3. Select Spectre as the Analog solver to be used.
- 4. Click OK.
- **5.** In the Virtuoso[®] Analog Design Environment window, choose *Simulation Netlist and Run*.

Status appears in the upper left corner of the window. Simulation output information appears in the neverilog.log file. The simulation time appears at the end of the file.

```
Total time required for tran analysis `tran' was 3.11407 ks (51m 54.1s).
```

You can compare these results to those from the simulation using AMS Designer with the UltraSim solver.

Note: This simulation ran for 60 minutes on our Solaris machine with a 1.6 G CPU.

When the simulation finishes, a graph window appears. For derails on using this waveform tool, see the *Virtuoso Visualization and Analysis XL User Guide*.