Real-Time Workshop®

For Use with Simulink®

Modeling

Simulation

Implementation

The MATH WORKS Inc.

User's Guide

Version 4

How to Contact The MathWorks:

508-647-7000	Phone
508-647-7001	Fax
The MathWorks, Inc. 3 Apple Hill Drive Natick, MA 01760-2098	Mail
`http://www.mathworks.com`	Web
`ftp.mathworks.com`	Anonymous FTP server
`comp.soft-sys.matlab`	Newsgroup
`support@mathworks.com`	Technical support
`suggest@mathworks.com`	Product enhancement suggestions
`bugs@mathworks.com`	Bug reports
`doc@mathworks.com`	Documentation error reports
`subscribe@mathworks.com`	Subscribing user registration
`service@mathworks.com`	Order status, license renewals, passcodes
`info@mathworks.com`	Sales, pricing, and general information

Printing History:			
	May 1994	First printing	Version 1
	January 1998	Second printing	Version 2.1
	January 1999	Third printing	Version 3.11 (Release 11)
	September 2000	Fourth printing	Version 4 (Release 12)

Contents

2 Technical Overview

3 Code Generation and the Build Process

5 External Mode

6 Program Architecture

7 Models with Multiple Sample Rates

15 Asynchronous Support

16 Real-Time Workshop Ada Coder

17 Targeting Real-Time Systems

Preface

Chapter Summary

Chapter 1, “Introduction to the Real-Time Workshop” introduces basic concepts and terminology of the Real-Time Workshop. It also provides information linking basic real-time development tasks to corresponding sections of this book. The “Getting Started: Basic Concepts and Tutorials” section in this chapter will get you working with hands-on exercises.

Chapter 2, “Technical Overview” is a quick introduction to the rapid prototyping process, the open architecture of the Real-Time Workshop, and the automatic program building process.

Chapter 3, “Code Generation and the Build Process” describes the automatic program building process in detail. It discusses all code generation options controlled by the Real-Time Workshop’s graphical user interface. Topics include data logging, inlining and tuning parameters, interfacing parameters and signals to your code, code generation from subsystems, and template makefiles. The chapter also summarizes available target configurations.

Chapter 4, “Generated Code Formats” compares and contrasts targets and their associated code formats. This include the real-time, real-time malloc, embedded C, and S-Function code formats.

Chapter 5, “External Mode” contains information about external mode, a simulation environment that supports on-the-fly parameter tuning, signal monitoring, and data logging.

Chapter 6, “Program Architecture” discusses the architecture of programs generated by the Real-Time Workshop, and the run-time interface.

Chapter 7, “Models with Multiple Sample Rates” describes how to handle multirate systems.

Chapter 8, “Optimizing the Model for Code Generation” discusses techniques for optimizing your generated programs.

Chapter 9, “Real-Time Workshop Embedded Coder” discusses the structure and operation of programs generated using the Real-Time Workshop Embedded Coder. The Real-Time Workshop Embedded Coder is designed for generation of code for embedded systems.

Chapter 10, “The S-Function Target” explains how to generate S-Function blocks from models and subsystems. This enables you to encapsulate models and subsystems and protect your designs by distributing only binaries.

Chapter 11, "Real-Time Workshop Rapid Simulation Target" discusses the rapid simulation target (RSIM), which executes your model in nonreal-time on your host computer. You can use this feature to generate fast, stand-alone simulations that allow batch parameter tuning and the loading of new simulation data (signals) from a standard MATLAB MAT-file without needing to recompile your model.

Chapter 12, "Targeting Tornado for Real-Time Applications" contains information that is specific to developing programs that target Tornado, and signal monitoring using StethoScope.

Chapter 13, "Targeting DOS for Real-Time Applications" contains information on developing programs that target DOS.

Chapter 14, "Custom Code Blocks" contains information about the Real-Time Workshop library, a collection of blocks and templates you can use to customize code generation for your application.

Chapter 15, "Asynchronous Support" describes the Interrupt Template library, which allow you to model synchronous/asynchronous event handling.

Chapter 16, "Real-Time Workshop Ada Coder" discusses the Real-Time Workshop Ada Target, which generates Ada code from your models. The Ada Coder is a separate product from the Real-Time Workshop.

Chapter 17, "Targeting Real-Time Systems" discusses information of interest to developers who want to develop programs for custom targets. This includes developing device driver blocks, customizing system target files and template makefiles, combining multiple models into a single executable, and APIs for external mode communication, signal monitoring, and parameter tuning.

Appendix A lists blocks whose use is restricted due to dependency on absolute time.

Appendix B is a glossary that contains definitions of terminology associated with the Real-Time Workshop and real-time development.

Related Products

The MathWorks provides several products that are especially relevant to the kinds of tasks you can perform with the Real-Time Workshop®. They are listed in the table below.

The Real-Time Workshop *requires* these products:

- MATLAB® 6.0 (Release 12)
- Simulink® 4.0 (Release 12)
- A supported compiler (See “Supported Compilers” on page xxiv and “Third-Party Compiler Installation on Windows” on page xxii)

For more information about any of these products, see either:

- The online documentation for that product, if it is installed or if you are reading the documentation from the CD
- The MathWorks Web site, at `http://www.mathworks.com`; see the “products” section

Note The toolboxes listed below all include functions that extend MATLAB’s capabilities. The blocksets listed below all include blocks that extend Simulink’s capabilities.

Product	Description
Communications Toolbox	MATLAB functions for modeling the physical layer of communications systems
Control System Toolbox	Tool for modeling, analyzing, and designing control systems using classical and modern techniques
Dials & Gauges Blockset	Graphical instrumentation for monitoring and controlling signals and parameters in Simulink models

Product	**Description**
DSP Blockset	Simulink block libraries for the design, simulation, and prototyping of digital signal processing systems
Fixed-Point Blockset	Simulink blocks that model, simulate, and automatically generate pure integer code for fixed-point applications
Fuzzy Logic Toolbox	Tool to help master fuzzy logic techniques and their application to practical control problems
Nonlinear Control Design (NCD) Blockset	Simulink block libraries that provide a time-domain-based optimization approach to system design; automatically tunes parameters based on user-defined time-domain performance constraints
Power System Blockset	Simulink block libraries for the design, simulation, and prototyping of electrical power systems
Real-Time Workshop Ada Coder	Tool that allows you to automatically generate Ada 95 code. It produces the code directly from Simulink models and automatically builds programs that can be run in real time in a variety of environments.
Real-Time Workshop Embedded Coder	Tool that allows you to automatically generate production-quality C code from Simulink models. Supports software-in-the-loop validation of generated code in Simulink.
Real-Time Windows Target	Tool that allows you to run Simulink models interactively and in real time on your PC under Windows
Simulink	Interactive, graphical environment for modeling, simulating, and prototyping dynamic systems

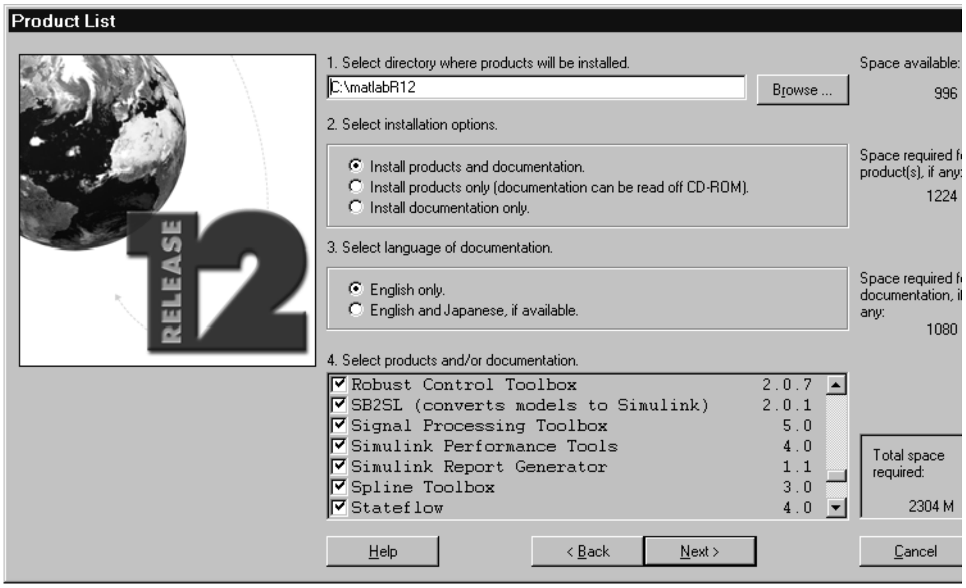
Product	**Description**
Stateflow®	Tool for graphical modeling and simulation of complex control logic
Stateflow Coder	Tool for generating highly readable, efficient C code from Stateflow diagrams
xPC Target	Tool that supports many I/O blocks for Simulink block diagrams. Supports downloading code generated by Real-Time Workshop to a second PC that runs the xPC Target real-time kernel, for rapid prototyping and hardware-in-the-loop testing of control and DSP systems.
xPC Target Embedded Option	Add-on to xPC Target to deploy the xPC Target operating system, together with embedded code, to external PCs

Installing the Real-Time Workshop

Your platform-specific *MATLAB Installation Guide* provides all of the information you need to install the Real-Time Workshop.

Prior to installing the Real-Time Workshop, you must obtain a License File or Personal License Password from The MathWorks. The License File or Personal License Password identifies the products you are permitted to install and use.

As the installation process proceeds, it displays a dialog similar to the one below, letting you indicate which products to install.



1. Select directory where products will be installed.
C:\matlabR12

Space available:
996

2. Select installation options.

(● Install products and documentation.)
○ Install products only (documentation can be read off CD-ROM).
○ Install documentation only.

Space required f
product(s), if any:
1224

3. Select language of documentation.

(● English only.)
○ English and Japanese, if available.

Space required f
documentation, i
any:
1080

4. Select products and/or documentation.

```
☑ Robust Control Toolbox                   2.0.7
☑ SB2SL (converts models to Simulink)      2.0.1
☑ Signal Processing Toolbox                  5.0
☑ Simulink Performance Tools                 4.0
☑ Simulink Report Generator                  1.1
☑ Spline Toolbox                             3.0
☑ Stateflow                                  4.0
```

Total space
required:
2304 M

The Real-Time Workshop has certain product prerequisites that must be met for proper installation and execution.

Licensed Product	Prerequisite Products	Additional Information
Simulink	MATLAB 6 (Release 12)	Allows installation of Simulink.
The Real-Time Workshop	Simulink 4.0 (Release 12)	Requires Borland C, LCC, Visual C/C++, or Watcom C compiler to create MATLAB MEX-files on your platform.
The Real-Time Workshop Ada Coder	The Real-Time Workshop 4.0	
The Real-Time Workshop Embedded Coder	The Real-Time Workshop 4.0	

If you experience installation difficulties and have Web access, connect to the MathWorks home page (`http://www.mathworks.com`). Look for the license manager and installation information under the `Tech Notes/FAQ` link under `Tech Support Info`.

Third-Party Compiler Installation on Windows

Several of the Real-Time Workshop targets create an executable that runs on your workstation. When creating the executable, the Real-Time Workshop must be able to access a compiler. The following sections describe how to configure your system so that the Real-Time Workshop has access to your compiler.

Borland

Make sure that your Borland environment variable is defined and correctly points to the directory in which your Borland compiler resides. To check this, type

```
set BORLAND
```

at the DOS prompt. The return from this includes the selected directory.

If the BORLAND environment variable is not defined, you must define it to point to where you installed your Borland compiler. On Microsoft Windows 95 or 98, add

```
set BORLAND=<path to your compiler>
```

to your autoexec.bat file.

On Microsoft Windows NT, in the control panel select **System**, go to the **Environment** page, and define BORLAND to be the path to your compiler.

LCC

The freeware LCC C compiler is shipped with MATLAB, and is installed with the product. If you want to use LCC to build programs generated by the Real-Time Workshop, you should use the version that is currently shipped with the product. Information about LCC is available at http://www.cs.virginia.edu/~lcc-win32/.

Microsoft Visual C/C++

Define the environment variable MSDevDir to be

```
MSDevDir=<path to compiler>\SharedIDE        for Visual C/C++ 5.0
MSDevDir=<path to compiler>\Common\MSDev98   for Visual C/C++ 6.0
```

Watcom

Note As of this printing, the Watcom C compiler is no longer available from the manufacturer. The Real-Time Workshop continues to ship Watcom-related target configurations at this time. However, this policy may be subject to change in the future.

Make sure that your Watcom environment variable is defined and correctly points to the directory in which your Watcom compiler resides. To check this, type

```
set WATCOM
```

at the DOS prompt. The return from this includes the selected directory.

If the `WATCOM` environment variable is not defined, you must define it to point to where you installed your Watcom compiler. On Windows 95 or 98, add

```
set WATCOM=<path to your compiler>
```

to your `autoexec.bat` file.

On Windows NT, in the control panel select **System**, go to the **Environment** page, and define `WATCOM` to be the path to your compiler.

Out-of-Environment Error Message

If you are receiving out-of-environment space error messages, you can right-click your mouse on the program that is causing the problem (for example, `dosprmpt` or `autoexec.bat`) and choose **Properties**. From there choose **Memory**. Set the **Initial Environment** to the maximum allowed and click **Apply**. This should increase the amount of environment space available.

Supported Compilers

On Windows. As of this printing, we have tested the Real-Time Workshop with these compilers on Windows.

Compiler	Versions
Borland	5.3, 5.4,5.5
LCC	Use version of LCC shipped with MATLAB.
Microsoft Visual C/C++	5.0, 6.0
Watcom	10.6, 11.0 (see “Watcom” above)

Typically you must make modifications to your setup when a new version of your compiler is released. See the MathWorks home page, `http://www.mathworks.com`, for up-to-date information on newer compilers.

On UNIX. On UNIX, the Real-Time Workshop build process uses the default compiler. `cc` is the default on all platforms except SunOS, where `gcc` is the default.

Compiler Optimization Settings

In some very rare instances, due to compiler defects, compiler optimizations applied to Real-Time Workshop generated code may cause the executable program to produce incorrect results, even though the code itself is correct.

The Real-Time Workshop uses the default optimization level for supported compilers. You can usually work around problems caused by compiler optimizations by lowering the optimization level of the compiler, or turning off optimizations. Please refer to your compiler's documentation for information on how to do this.

Typographical Conventions

This manual uses some or all of these conventions.

To Indicate...	This Guide Uses...	Example
Example code	`Monospace font`	To assign the value 5 to `A`, enter `A = 5`
Function names/syntax	`Monospace font`	The `cos` function finds the cosine of each array element. Syntax line example is `MLGetVar ML_var_name`
Mathematical expressions	*Italics* for variables Standard text font for functions, operators, and constants	This vector represents the polynomial $p = x^2 + 2x + 3$

To Indicate...	This Guide Uses...	Example

MATLAB output	`Monospace` font	MATLAB responds with
		`A =`
		`   5`

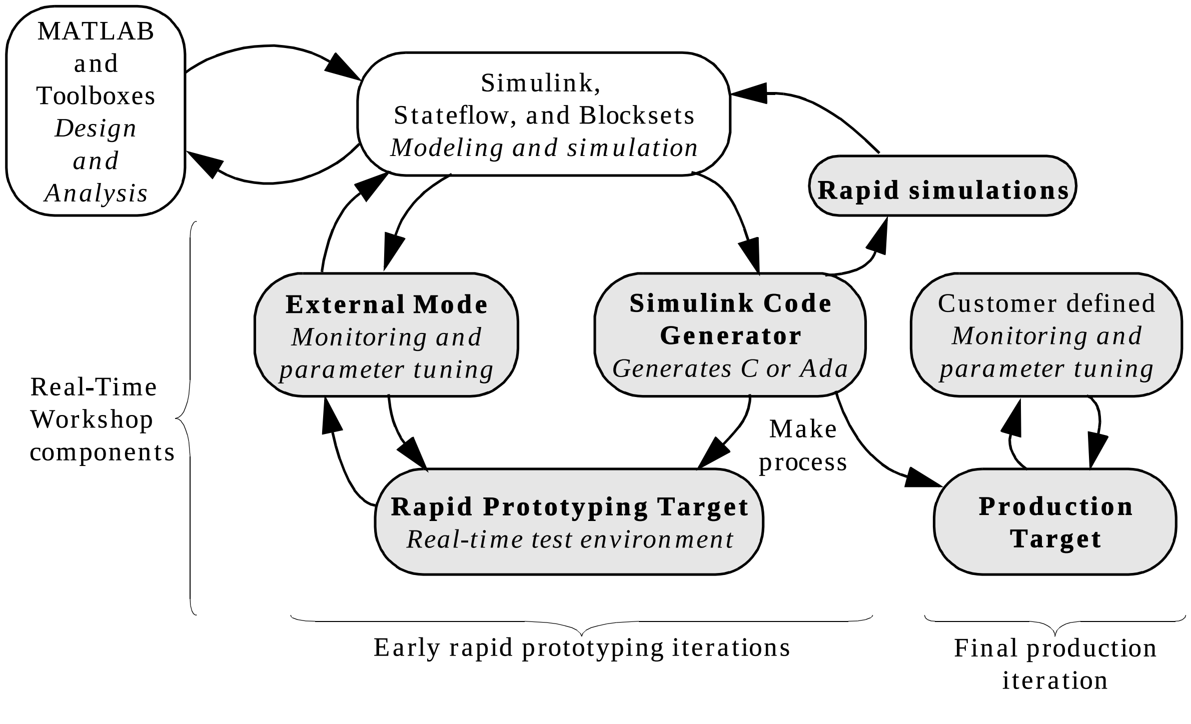
Menu names, menu items, and controls	**Boldface** with an initial capital letter	Choose the **File** menu.
New terms	*Italics*	An *array* is an ordered collection of information.
String variables (from a finite list)	*`Monospace italics`*	`sysc = d2c(sysd, '`*`method`*`')`

Introduction to the Real-Time Workshop

Product Summary

Real-Time Workshop generates optimized, portable, and customizable code from Simulink models. Using integrated makefile based targeting support, it builds programs that can help speed up your simulations, provide intellectual property protection, or run on a wide variety of real-time rapid prototyping or production targets. Simulink's external mode run-time monitor works seamlessly with real-time targets, providing an elegant signal monitoring and parameter tuning interface. Real-Time Workshop supports continuous-time, discrete-time and hybrid systems, including conditionally executed and atomic systems. Real-Time Workshop accelerates your development cycle, producing higher quality results in less time.

Real-Time Workshop is a key link in the set of system design tools provided by The MathWorks. Conceptually, Real-Time Workshop is the final piece in the design process.


MATLAB and Toolboxes
Design and Analysis
Simulink, Stateflow, and Blocksets
Modeling and simulation
Rapid simulations
External Mode
Monitoring and parameter tuning
Simulink Code Generator
Generates C or Ada
Customer defined
Monitoring and parameter tuning
Make process
Rapid Prototyping Target
Real-time test environment
Production Target
Real-Time Workshop components
Early rapid prototyping iterations
Final production iteration

Real-Time Workshop provides a real-time development environment — a direct path from system design to hardware implementation. You can shorten development cycles and reduce costs with Real-Time Workshop by testing design iterations with real-time hardware. Real-Time Workshop supports the execution of dynamic system models on hardware by automatically converting models to code and providing model-based debugging support. It is well suited for accelerating simulations, rapid prototyping, turnkey solutions, and production embedded real-time applications.

With Real-Time Workshop, you can quickly generate C code for discrete-time, continuous-time, and hybrid systems, including systems containing triggered and enabled subsystems. With the optional Real-Time Workshop Ada Coder, you can generate Ada code. The optional Stateflow Coder add-on lets you generate code for finite state machines modeled in Stateflow.

System design using the MathWorks toolset differs from one application to another. A typical product cycle starts with modeling in Simulink, followed by an analysis of the simulations in MATLAB. During the simulation process, you use the rapid simulation features of Real-Time Workshop to speed up your simulations.

After you are satisfied with the simulation results, you use Real-Time Workshop in conjunction with a rapid prototyping target, such as xPC Target. The rapid prototyping target is connected to your physical system. You test and observe your system, using your Simulink model as the interface to your physical target. After creating your model, you use Real-Time Workshop to transform your model to C or Ada code. An extensible make process and download procedure creates an executable for your model and places it on the target system. Finally, using external mode, you can monitor and tune parameters in real-time as your model executes on the target environment.

Conceptually, there are two types of targets: rapid prototyping targets and the embedded target. Code generated for the rapid prototyping targets supports increased monitoring and tuning capabilities. The generated embedded code used in the embedded target is highly optimized and suitable for deployment in production systems. You can add application-specific entry points to monitor signals and tune parameters in the embedded code.

The basic components of Real-Time Workshop are:

- *Simulink Code Generator:* automatically generates C or Ada code from your Simulink model.
- *Make Process:* The Real-Time Workshop's user-extensible make process lets you create your own production or rapid prototyping target.
- *Simulink External Mode:* External mode enables communication between Simulink and a model executing on a real-time test environment, or in another process on the same machine. External mode lets you perform real-time parameter tuning and data viewing using Simulink as a front end.
- *Targeting Support:* Using Real-Time Workshop's bundled targets, you can build systems for a number of environments, including Tornado and DOS. The generic real-time and embedded real-time targets provide a framework for developing customized rapid prototyping or production target environments. In addition to the bundled targets, Real-Time Windows Target and/or xPC Target let you turn a PC of any form factor into a rapid prototyping target, or a small to medium volume production target.
- *Rapid Simulations*: Using Simulink Accelerator (part of the Simulink Performance Tools product), S-Function Target, or Rapid Simulation Target, you can accelerate your simulations by 5 to 20 times on average. Executables built with these targets bypass Simulink's normal interpretive simulation mode, which must handle all configurations of each basic modeling primitive. The code generated by Simulink Accelerator, S-Function Target, or Rapid Simulation Target is highly optimized to execute only the algorithms used in your specific model. In addition, these targets apply many optimizations, such as eliminating ones and zeros in computations for filter blocks.

Integrated Development Environment

If the Real-Time Workshop target you are using supports Simulink external mode, you can use Simulink as the monitoring/debugging interface for the generated code. With external mode, you can:

- Change parameters via the block dialogs, gauges, and the `set_param` MATLAB command. The `set_param` command lets you interact programmatically with your target.
- View target signals in Scope blocks, Display blocks, general S-Function blocks, and via gauges.

These concepts are illustrated by Figure 1-1 and Figure 1-2.

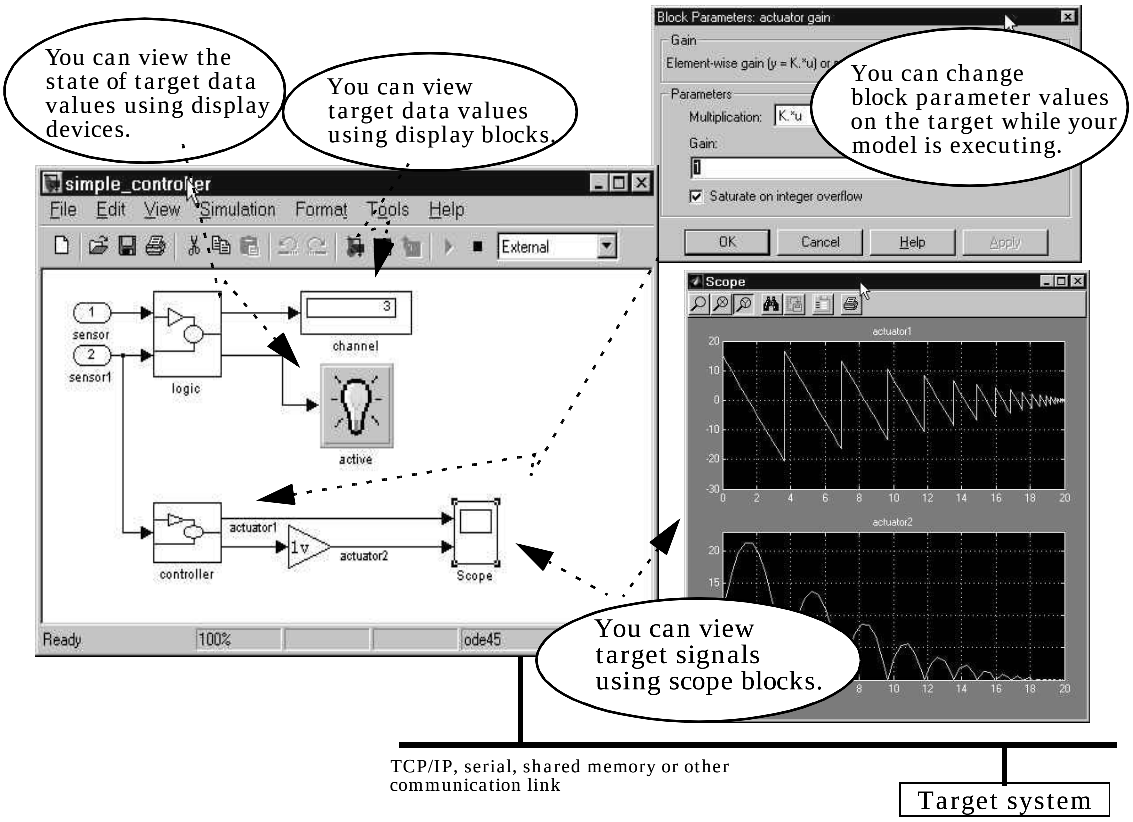

You can view the state of target data values using display devices.
You can view target data values using display blocks.
You can change block parameter values on the target while your model is executing.
You can view target signals using scope blocks.
Block Parameters: actuator gain
Gain
Element-wise gain (y = K.*u) or
Parameters
Multiplication: K.*u
Gain: 1
Saturate on integer overflow
OK
Cancel
Help
Apply
simple_controller
File Edit View Simulation Format Tools Help
External
sensor
sensor1
logic
channel
active
controller
actuator1
actuator2
Scope
Ready
100%
ode45
Scope
actuator1
actuator2
TCP/IP, serial, shared memory or other communication link
Target system

Figure 1-1: Signal Viewing and Parameter Tuning in External Mode

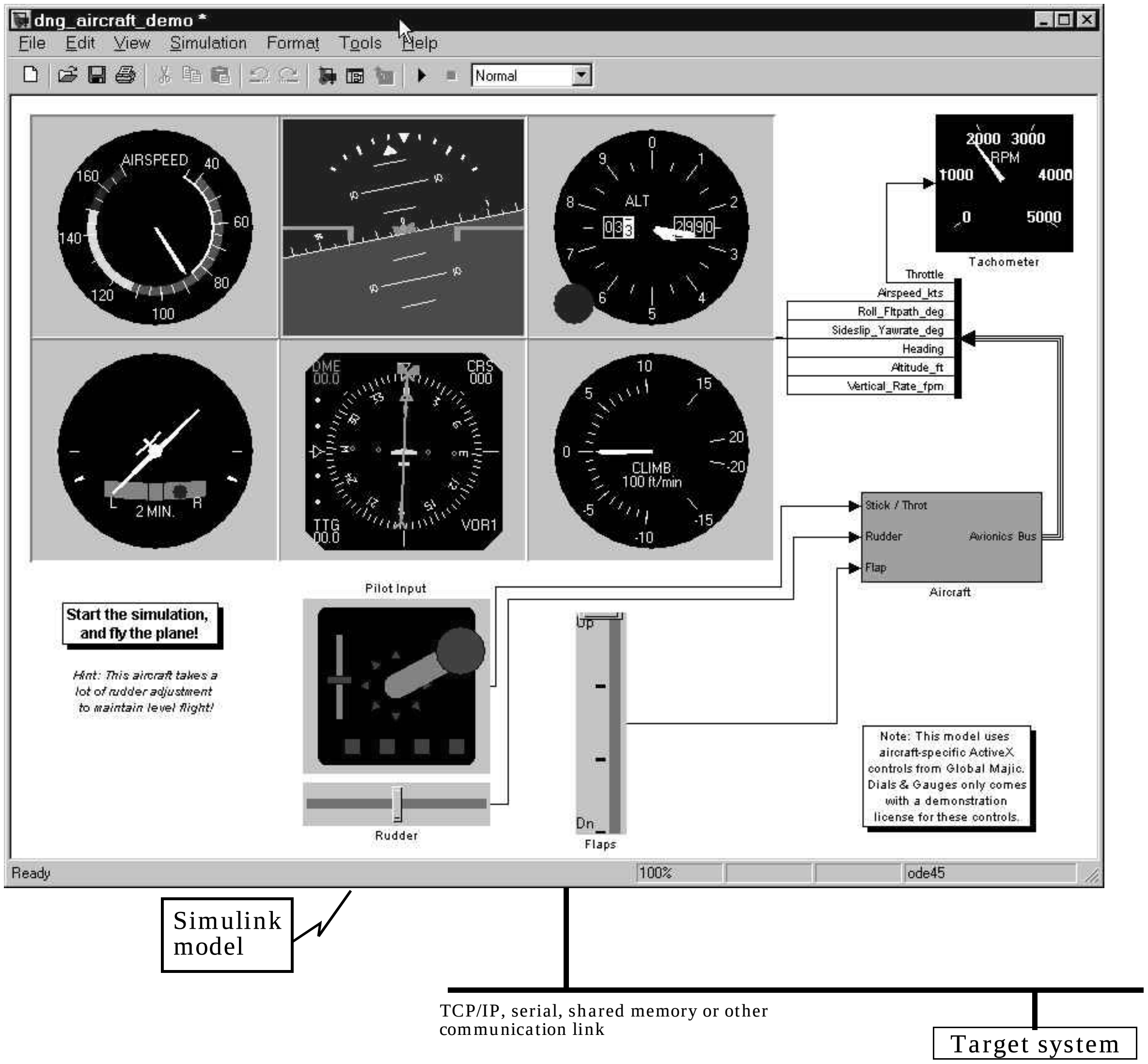

dng_aircraft_demo *
File Edit View Simulation Format Tools Help
Normal
AIRSPEED
ALT
CLIMB 100 ft/min
2 MIN.
DME 00.0
CRS 000
TTG 00.0
VOR1
RPM
Tachometer
Throttle
Airspeed_kts
Roll_Fltpath_deg
Sideslip_Yawrate_deg
Heading
Altitude_ft
Vertical_Rate_fpm
Stick / Throt
Rudder
Flap
Avionics Bus
Aircraft
Pilot Input
Rudder
Flaps
Up
Dn
Start the simulation, and fly the plane!
Hint: This aircraft takes a lot of rudder adjustment to maintain level flight!
Note: This model uses aircraft-specific ActiveX controls from Global Majic. Dials & Gauges only comes with a demonstration license for these controls.
Ready
100%
ode45
Simulink model
TCP/IP, serial, shared memory or other communication link
Target system

Figure 1-2: Dials and Gauges Provide Front End to Target System

A Next-Generation Development Tool

The MathWorks toolset, including Simulink and Real-Time Workshop, is revolutionizing the way embedded systems are designed. Simulink is a very high level language (VHLL) — a next-generation programing language. A brief look at the history of dynamic and embedded system design methodologies reveals a steady progression toward higher-level design tools and processes:

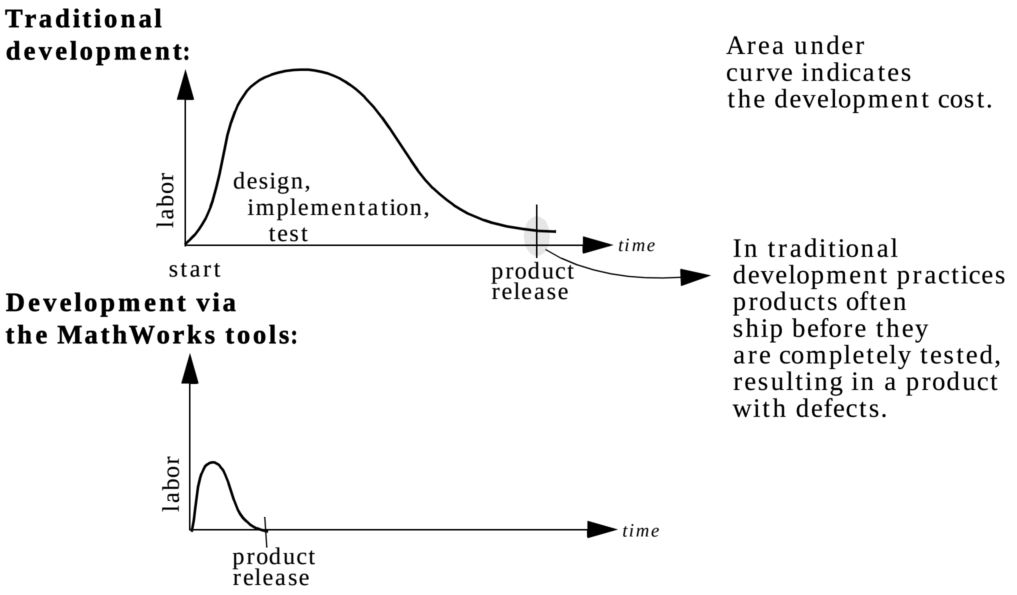
- *Design -> analog components:* Before the introduction of microcontrollers, design was done on paper and realized using analog components.
- *Design -> hand written assembly -> early microcontrollers:* In the early microprocessor era, design was done on paper and realized by writing assembly code and placing it on microcontrollers. Today, very low-end applications still use assembly language, but advancements in Real-Time Workshop and C/Ada compiler technology will soon render such techniques obsolete.
- *Design -> high-level language (HLL) -> object code -> microcontroller:* The advent of efficient HLL compilers led to the realization of paper designs in languages such as C. HLL code, transformed to assembly language by a compiler, was then placed on a microcontroller. In the early days of high-level languages, programmers often inspected the machine generated assembly code produced by compilers for correctness. Today, it is taken for granted that the assembly code is correct.
- *Design -> modeling tool -> manual HLL coding -> object code -> microcontroller:* When design tools such as Simulink appeared, designers were able to express system designs graphically and simulate them for correctness. While this process saved considerable time and improved performance, designs were still translated to C code manually before being placed on a microcontroller. This translation process was both time consuming and error prone.
- *Design -> Simulink -> Real-Time Workshop (automatic code generation) -> object code -> microcontroller*. With the addition of Real-Time Workshop, Simulink itself becomes a very high level language (VHLL). Modeling constructs in Simulink are the basic elements of the language. The Real-Time Workshop then compiles models to produce C or Ada code. This machine-generated code is produced quickly and correctly. The manual

process of transforming designs to code has now been eliminated, yielding significant improvements in system design.

The Simulink code generator included within Real-Time Workshop is a next-generation graphical block diagram compiler. Real-Time Workshop has capabilities beyond those of a typical HLL compiler. Generated code is highly readable and customizable. It is normally unnecessary to read the object code produced by the HLL compiler.. You can use the Real-Time Workshop in a wide variety of applications, improving your design process.

Key Features

The general goal of the MathWorks toolset, including Real-Time Workshop, is to enable you to *accelerate your design process while reducing cost, decreasing time to market, and improving quality*.


Traditional development:
Area under curve indicates the development cost.
labor
design, implementation, test
time
start
product release
In traditional development practices products often ship before they are completely tested, resulting in a product with defects.
Development via the MathWorks tools:
labor
time
product release

Traditional development practices tend to be very labor intensive. Poor tools often lead to a proliferation of *ad hoc* software projects that fail to deliver reusable code. With the MathWorks toolset, you can focus energy on design and achieve better results in less time with fewer people.

Real-Time Workshop, along with other components of the MathWorks tools, provides:

- A rapid and direct path from system design to implementation
- Seamless integration with MATLAB and Simulink
- A simple graphical user interface
- An open and extensible architecture

The following features of Real-Time Workshop enable you to reach the above goal:

- **Code generator for Simulink models**
  - Generates optimized, customizable code. There are several styles of generated code, which can be classified as either embedded (production phase) or rapid prototyping.
  - Supports all Simulink features, including 8, 16, and 32 bit integers and floating-point data types.
  - Fixed-Point Blockset and Real-Time Workshop allow for scaling of integer words ranging from 2 to 128 bits. Code generation is limited by the implementation of `char`, `short`, `int`, and `long` `in` embedded C compiler environments (usually 8, 16, and 32 bits).
  - Generated code is processor independent. The generated code represents your model exactly. A separate run-time interface is used to execute this code. We provide several example run-time interfaces as well as production run-time interfaces.
  - Supports any single or multitasking operating system. Also supports “bare-board” (no operating system) environments.
  - The Target Language Compiler^{TM} (TLC) allows extensive customization of the generated code.
  - Provides for custom code generation for S-functions (user-created blocks) via TLC, enabling you to embed very efficient custom code into the model’s generated code.
- **Extensive model-based debugging support**
  - External mode enables you to examine what the generated code is doing by uploading data from your target to the graphical display elements in your model. There is no need to use a conventional C or Ada debugger to look at your generated code.

  - External mode also enables you to tune the generated code via your Simulink model. When you change a parametric value of a block in your model, the new value is passed down to the generated code, running on your target, and the corresponding target memory location is updated. Again, there is no need to use an embedded compiler debugger to perform this type of operation. Your model is your debugger user interface.
- **Integration with Simulink**
  - Code validation. You can generate code for your model and create a standalone executable that exercises the generated code and produces a MAT-file containing the execution results.
  - Generated code contains system/block identification tags to help you identify the block, in your source model, that generated a given line of code. The MATLAB command `hilite_system` recognizes these tags and highlights the corresponding blocks in your model.
  - Support for Simulink Data Objects lets you define how your signals and block parameters interface to the external world.
- **Rapid simulations**
  - Real-Time Workshop supports several ways to speed up your simulations by creating optimized, model-specific executables.
- **Target support**
  - Turnkey solutions for rapid prototyping substantially reduce design cycles, allowing for fast turnaround of design iterations.
  - Bundled rapid prototyping example targets are provided.
  - Add-on targets (Real-Time Windows Target and xPC Target) for PC based hardware are available from The MathWorks. These targets enable you to turn a PC with fast, high-quality, low cost hardware into a rapid prototyping system.
  - Supports a variety of third-party hardware and tools, with extensible device driver support.
- **Extensible make process**
  - Allows for easy integration with any embedded compiler and linker.
  - Provides for easy linkage with your hand-written supervisory or supporting code.

- **Real-Time Workshop Embedded Coder provides:**
  - Customizable, portable, and readable C code that is designed to be placed in a production embedded environment.
  - More efficient code is created, because inlined S-functions are required and continuous time states are not allowed.
  - Software-in-the-loop. With Real-Time Workshop Embedded Coder, you can generate code for your embedded application and bring it back into Simulink for verification via simulation.
  - Web-viewable code generation report describes code modules, analyzes the generated code, and helps to identify code generation optimizations relevant to your program.
  - Annotation of the generated code using the Description block property.
  - Hooks for external parameter tuning and signal monitoring are provided enabling easy interfacing of the generated code in your real-time system.
- **Real-Time Workshop Ada Coder provides:**
  - Customizable, readable, efficient embeddable Ada code.
  - Generates more efficient code than the classic Real-Time Workshop targets. This is possible because all S-functions must be inlined using the Target Language Compiler.
  - Hooks for external parameter tuning and signal monitoring are provided enabling easy interfacing of the generated code in your real-time system.
  - Annotation of the generated code using the Description block property.

Benefits

You can benefit by using Real-Time Workshop in the following applications. This is not an exhaustive list, but a general survey.

- **Production Embedded Real-Time Applications**

  Real-Time Workshop lets you generate, cross-compile, link, and download production quality C or Ada code for real-time systems (such as controllers or DSP applications) onto your target processor directly from Simulink. You can customize the generated code by inserting S-functions into your model and specifying, via the Target Language Compiler, what the generated code should look like. Using the optimized, automatically generated code, you can focus your coding efforts on specific features of your product, such as device drivers and general device interfacing.

- **Rapid Prototyping**

  As a rapid prototyping tool, Real-Time Workshop enables you to implement your embedded systems designs quickly, without lengthy hand-coding and debugging. Rapid prototyping is typically used in the software/hardware integration and testing phases of the design cycle enabling you to:

  - Conceptualize solutions graphically in a block diagram modeling environment.
  - Evaluate system performance early on - prior to laying out hardware, coding production software, or committing to a fixed design.
  - Refine your design by rapid iteration between algorithm design and prototyping.
  - Tune parameters while your real-time model runs, using Simulink operating in external mode as a graphical front end.

  You can use Real-Time Workshop to generate downloadable, targeted C code that runs on top of a real-time operating system (RTOS). Alternatively, you can generate code to run on the bare hardware at interrupt level, using a simple rate monotonic scheduling executive that you create from examples provided with the Real-Time Workshop. There are many rapid prototyping targets provided; or you can create your own.

  During rapid prototyping, the generated code is fully instrumented enabling direct access via Simulink external mode for easy monitoring and debugging. The generated code contains a data structure that encapsulates the details of your model. This data structure is used in the bidirectional connection to Simulink running in external mode. Using Simulink external mode, you can monitor signal and tune parameters to further refine your model in rapid iterations enabling you to achieve desired results quickly.

- **Real-Time Simulation**

  You can create and execute code for an entire system or specified subsystems for hardware-in-the-loop simulations. Typical applications include training simulators, real-time model validation, and prototype testing.

- **Turnkey Solutions**

  Bundled Real-Time Workshop targets and third-party turnkey solutions support a variety of control and DSP applications. The target environments include embedded PC, PCI, ISA, VME, and custom hardware, running off-the-shelf real-time operating systems, DOS, or Microsoft Windows. Target system processor architectures include Motorola MC680x0 and PowerPC processors, Intel-80x86 and compatibles, Alpha, and Texas Instruments DSPs. Third-party vendors are regularly adding other architectures. For a current list of third-party turnkey solutions, see the MATLAB Connections Web page: `http://www.mathworks.com/products/connections`.

The open environment of Real-Time Workshop also lets you create your own turnkey solution.

- **Intellectual Property Protection**

  The S-Function Target, in addition to speeding up your simulation, allows you to protect your intellectual property: the designs and algorithms embodied in your models. Using the S-Function Target, you can generate and distribute binaries from your models or subsystems. End users have access to the interface, but not to the body, of your algorithms.

- **Rapid Simulations**

  The MathWorks tools can be used in the design of most dynamic systems. Generally Simulink is either used to model a high-fidelity dynamic system (e.g., an engine) or a real-time system (such as an engine controller or a signal processing system).

  When modeling high-fidelity systems, you can use Real-Time Workshop to accelerate the design process by speeding up your simulations. This is achieved by using one of the following Real-Time Workshop components:

  - Simulink Accelerator: Creates a dynamically linked library (MEX-file) from code optimized and generated for your specific model configuration. This executable is used in place of the normal interpretive mode of simulation. Typical speed improvements range from 2 to 8 times faster than normal simulation time. Simulink Accelerator supports both fixed and variable step solvers. Simulink Accelerator is part of the Simulink Performance Tools product.
  - Rapid Simulation Target: Creates a stand-alone executable from code optimized and generated for your specific model configuration. This stand-alone executable does not need to interact with a graphics subsystem. Typical speed improvements range from 5 to 20 times faster than normal simulation times. The Rapid Simulation Target is ideal for

repetitive (batch) simulations where you are adjusting model parameters or coefficients. Rapid Simulation Target supports only fixed-step solvers.

- S-Function Target: This target, like Simulink Accelerator, creates a dynamically linked library (MEX-file) from a model. You can incorporate this component into another model using the Simulink S-function block.

The MathWorks Tools and the Development Process

Figure 1-3 is a high-level view of a traditional development process *without* the MathWorks toolset.

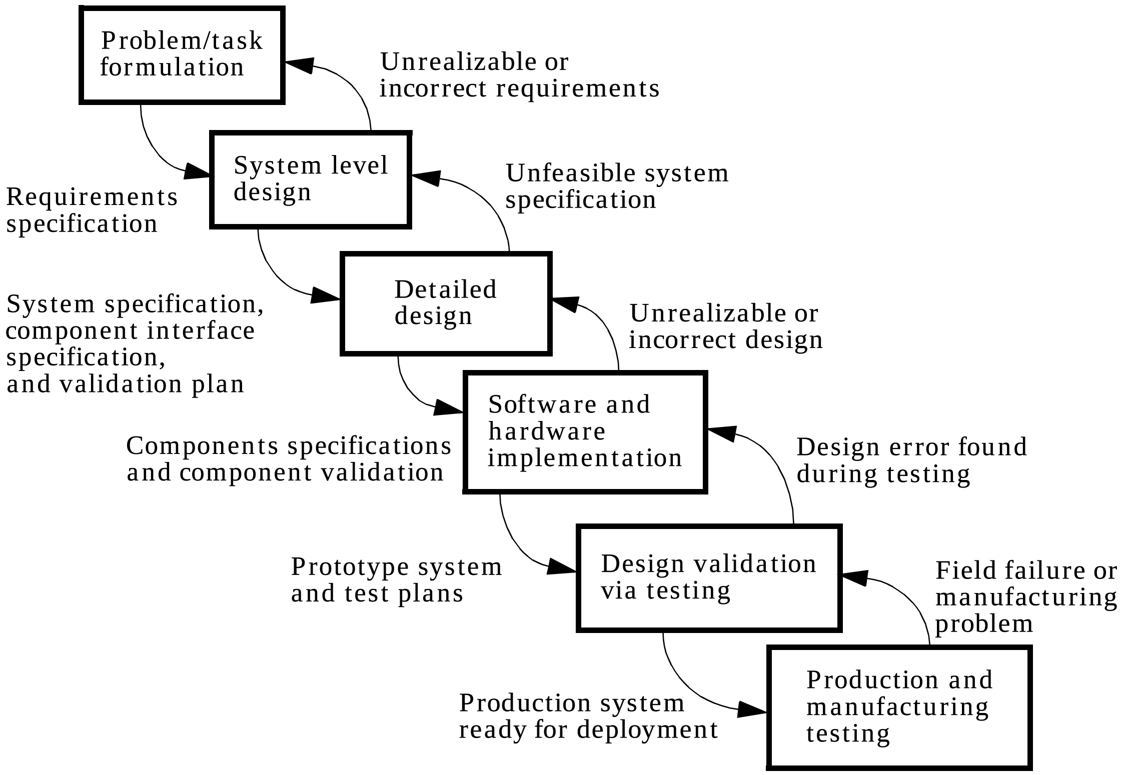

Problem/task formulation
Unrealizable or incorrect requirements
System level design
Requirements specification
Unfeasible system specification
Detailed design
System specification, component interface specification, and validation plan
Unrealizable or incorrect design
Software and hardware implementation
Components specifications and component validation
Design error found during testing
Design validation via testing
Prototype system and test plans
Field failure or manufacturing problem
Production and manufacturing testing
Production system ready for deployment

Figure 1-3: Traditional Development Process Without MathWorks Toolset

In Figure 1-3, each block represents a work phase. Documents are used to coordinate the different work phases. In this environment, it is easy to go back one work phase, but hard to go back multiple work phases. In this environment, design engineers (such as control system engineers or signal processing engineers) are not usually involved in the prototyping phase until many months after they have specified the design. This can result in poor time to market and inferior quality.

In this environment, different tools are used in each phase. Designs are communicated via paper. This enforces a serial, rather than an iterative, development process. Developers must re-enter the result of the previous phase before they can begin work on a new phase. This leads to miscommunication and errors, resulting in lost work hours. Errors found in later phases are very expensive and time consuming to correct. Correction often involves going back several phases. This is difficult because of the poor communication between the phases.

The MathWorks does not suggest or impose a development process. The MathWorks toolset can be used to complement any development process. In the above process, use of our tools in each phase can help eliminate paper work.

Our toolset also lends itself well to the spiral design process shown in Figure 1-4.

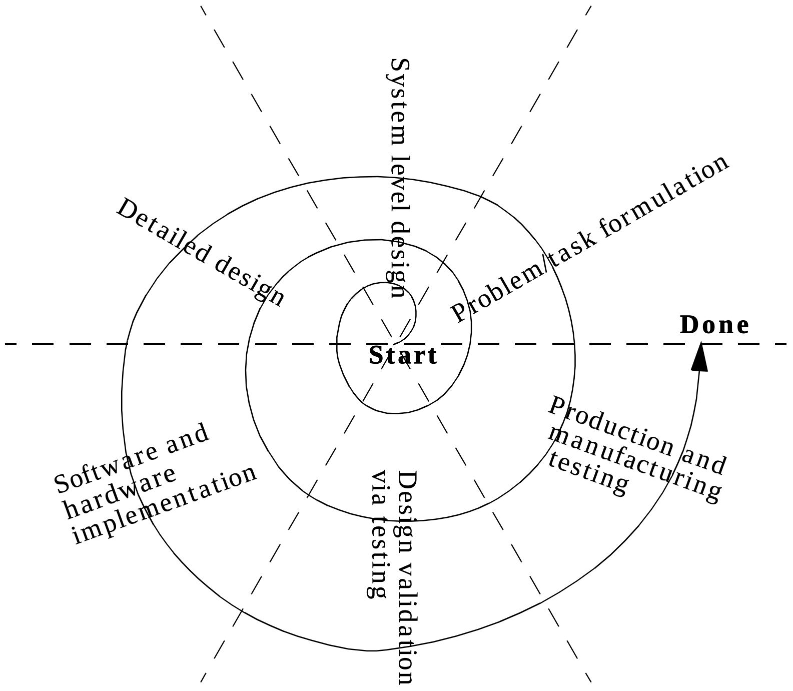

System level design
Problem/task formulation
Done
Detailed design
Start
Software and hardware implementation
Design validation via testing
Production and manufacturing testing

Figure 1-4: Spiral Design Process

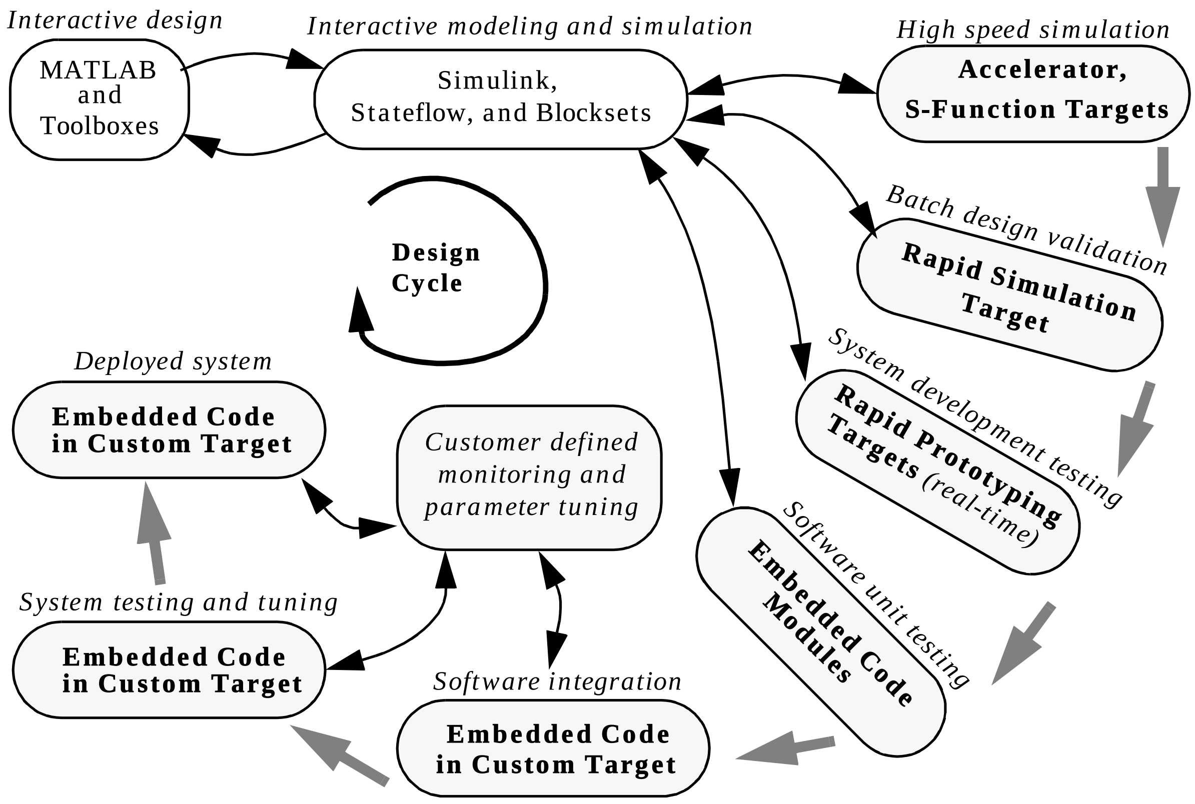
Using the MathWorks toolset, *your model represents your understanding of your system*. This understanding is passed from phase to phase in the model, reducing the need to go back to a previous phase. In the event that rework is necessary in a previous phase, it is easier to transition back one or more phases, because the same model and tools are used in all phases.

A spiral design process iterates quickly between phases, enabling engineers to work on innovative features. The only way to do this cost effectively is to use tools that make it easy to move from one phase to another. For example, in a matter of minutes a control system engineer or a signal processing engineer can validate an algorithm on a real-world rapid prototyping system. The spiral process lends itself naturally to parallelism in the overall development process. You can provide early working models to validation and production groups, involving them in your system development process from the start. This helps compress the overall development cycle while increasing quality.

Another advantage of the MathWorks toolset is that it enables people to work on tasks that they are good at and enjoy doing. For example, control system engineers specialize in design control laws, while embedded system engineers enjoy pulling together a system consisting of hardware and low-level software. It is possible to have very talented people perform different roles, but it is not efficient. Embedded system engineers, for example, are rewarded by specifying and building the hardware and creating low-level software such as device drivers, or real-time operating systems. They do not find data entry operations, such as the manual conversion of a set of equations to efficient code, to be rewarding. This is where the MathWorks toolset shines. The equations are represented as models and Real-Time Workshop converts them to highly efficient code ready for deployment.

Role of the MathWorks Tools in Your Development Process

The following figure outlines where the MathWorks toolset, including Real-Time Workshop, helps you in your development process.


Interactive design
MATLAB and Toolboxes
Interactive modeling and simulation
Simulink, Stateflow, and Blocksets
High speed simulation
Accelerator, S-Function Targets
Batch design validation
Rapid Simulation Target
System development testing
Rapid Prototyping Targets (real-time)
Software unit testing
Embedded Code Modules
Design Cycle
Deployed system
Embedded Code in Custom Target
Customer defined monitoring and parameter tuning
System testing and tuning
Embedded Code in Custom Target
Software integration
Embedded Code in Custom Target

Early in the design phase, you will start with MATLAB and Simulink to help you formulate your problems and create your initial design. Real-Time Workshop helps with this process by enabling high-speed simulations via Simulink Accelerator (also part of Simulink Performance Tools), and the S-function Target for componentization and model speed-up.

After you have a functional model, you may need to tune your model's coefficients. This can be done quickly using Real-Time Workshop's Rapid

Simulation Target for Monte-Carlo type simulations (varying coefficients over many simulations).

After you've tuned your model, you can move into system development testing by exercising your model on a rapid prototyping system such as Real-Time Windows Target or xPC Target. With a rapid prototyping target, you connect your model to your physical system. This lets you locate design flaws or modeling errors quickly.

After your prototype system is created, you can use Real-Time Workshop Embedded Coder to create embeddable code for deployment on your custom target. The signal monitoring and parameter tuning capabilities enable you to easily integrate the embedded code into a production environment equipped with debugging and upgrade capabilities.

Code Formats

The Real-Time Workshop code generator transforms your model to HLL code. Real-Time Workshop supports a variety of code formats designed for different execution environments, or targets.

In the traditional embedded system development process, an engineer develops an algorithm (or equations) to be implemented in an embedded system. These algorithms are manually converted to a computer language such as C or Ada. This translation process, usually done by an embedded system engineer, is much like data entry.

Using Simulink to specify the algorithm (or equations), and Real-Time Workshop to generate corresponding code, engineers can bypass this redundant translation step. This enables embedded system engineers to focus on the key issues involved in creating an embedded system: the hardware configuration, device drivers, supervisory logic, and supporting logic for the model equations. Simulink itself is the programming language that expresses the algorithmic portion of the system.

The Simulink code generator provided with Real-Time Workshop is an open "graphical compiler" supporting a variety of code formats. The relationship between code formats and targets is shown below.

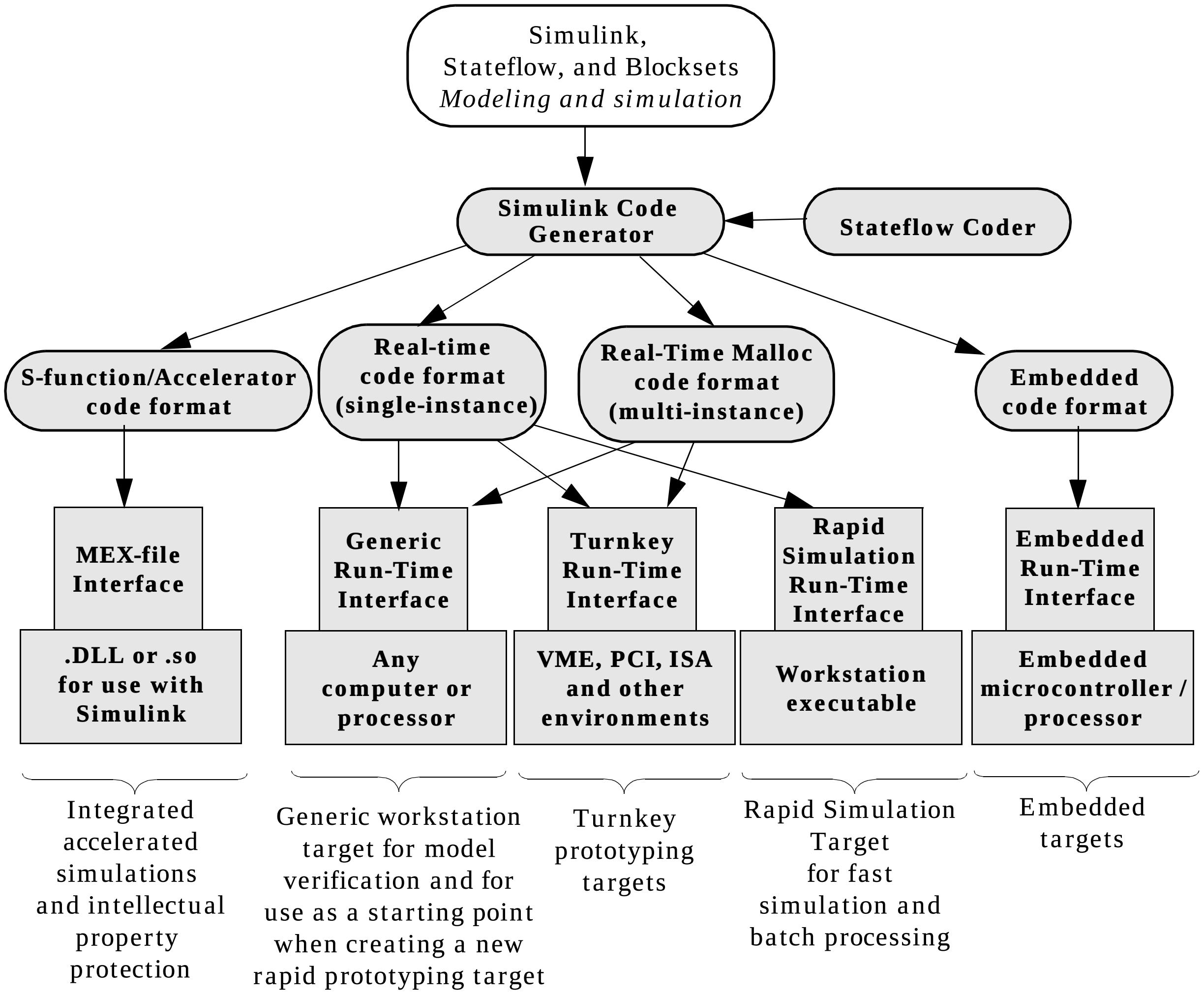

Simulink, Stateflow, and Blocksets
Modeling and simulation
Simulink Code Generator
Stateflow Coder
S-function/Accelerator code format
Real-time code format (single-instance)
Real-Time Malloc code format (multi-instance)
Embedded code format
MEX-file Interface
Generic Run-Time Interface
Turnkey Run-Time Interface
Rapid Simulation Run-Time Interface
Embedded Run-Time Interface
.DLL or .so for use with Simulink
Any computer or processor
VME, PCI, ISA and other environments
Workstation executable
Embedded microcontroller / processor
Integrated accelerated simulations and intellectual property protection
Generic workstation target for model verification and for use as a starting point when creating a new rapid prototyping target
Turnkey prototyping targets
Rapid Simulation Target for fast simulation and batch processing
Embedded targets

Figure 1-5: Relationship Between Code Formats and Targets

S-Function/Accelerator Code Format

This code format, used by the S-Function Target and Simulink Accelerator, generates code that conforms to Simulink C MEX S-function API.

Real-Time Code Format

The real-time code format is ideally suited for rapid prototyping. This code format (C only) supports increased monitoring and tuning capabilities, enabling easy connection with external mode. Real-time code format supports continuous-time models, discrete-time single- or multirate models, and hybrid continuous-time and discrete-time models. Real-time code format supports both inlined and noninlined S-functions. Memory allocation is declared statically at compile time.

Real-Time Malloc Code Format

The real-time malloc code format is similar to the real-time code format. The primary difference is that the real-time malloc code format declares memory dynamically. This supports multiple instances of the same model, with each instance including a unique data set. Multiple models can be combined into one executable without name clashing. Multiple instances of a given model can also be created in one executable.

Embedded Code Format

The embedded code format is designed for embedded targets. The generated code is optimized for speed, memory usage, and simplicity. Generally, this format is used in deeply embedded or deployed applications. There are no dynamic memory allocation calls; all persistent memory is statically allocated. Real-Time Workshop can generate either C or Ada code in the embedded code format. Note Ada code requires Real-Time Workshop Ada Coder, an add-on product.

The embedded code format provides a simplified calling interface and reduced memory usage. This format manages model and timing data in a compact real-time object structure. This contrasts with the other code formats, which use a significantly larger, model-independent Simulink data structure (`SimStruct`) to manage the generated code.

The embedded code format improves readability of the generated code, reduces code size, and speeds up execution. The embedded code format supports all discrete-time single- or multirate models.

Because of its optimized and specialized data structures, the embedded code format supports only inlined S-functions.

Target Environments

The Real-Time Workshop supports many target environments. These include ready-to-run configurations and third-party targets. You can also develop your own custom target.

This section begins with a list of available target configurations. Following the list, we summarize the characteristics of each target.

Available Target Configurations

Target Configurations Bundled with Real-Time Workshop. The MathWorks supplies the following target configurations with Real-Time Workshop:

- DOS (4GW) Target (example only)
- Generic Real-Time (GRT) Target
- LE/O (Lynx Embedded OSEK) Real-Time Target (example only)
- Rapid Simulation Target
- Tornado (VxWorks) Real-Time Target

Target Configurations Bundled with Real-Time Workshop Ada Coder. The MathWorks supplies the following target configurations with Real-Time Workshop Ada Coder (a separate product from the Real-Time Workshop):

- Ada Real-Time Multitasking Target
- Ada Simulation Target

Target Configurations Bundled with Real-Time Workshop Embedded Coder. The MathWorks supplies the following target configuration with Real-Time Workshop Embedded Coder (a separate product from the Real-Time Workshop):

- Real-Time Workshop Embedded Coder Target

Turnkey Rapid Prototyping Target Products. These self-contained solutions (separate products from the Real-Time Workshop) include:

- Real-Time Windows Target

- xPC Target

DSP Target Products. See *Texas Instruments DSP Developer's Kit User's Guide* for information on these targets:

- Texas Instruments Code Composer Studio Target
- Texas Instruments EVM67x Target

Third-Party Targets. Numerous software vendors have developed customized targets for the Real-Time Workshop. For an up-to-date listing of third-party targets, visit the MATLAB Connections Web page at `http://www.mathworks.com/products/connections`

View **Third-Party Solutions by Product Type**, and then select **RTW Target**.

Custom Targets. Typically, to target custom hardware, you must write a harness (main) program for your target system to execute the generated code, and I/O device drivers to communicate with your hardware. You must also create a system target file and a template makefile.

The Real-Time Workshop supplies generic harness programs as starting points for custom targeting. Chapter 17, "Targeting Real-Time Systems" provides the information you will need to develop a custom target.

Rapid Simulation Target

Rapid Simulation Target (RSIM) consists of a set of target files for non-real-time execution on your host computer. RSIM enables you to use the Real-Time Workshop to generate fast, stand-alone simulations. RSIM allows batch parameter tuning and downloading of new simulation data (signals) from a standard MATLAB MAT-file without the need to recompile the model.

The speed of the generated code also makes RSIM ideal for Monte Carlo simulations. The RSIM target enables the generated code to read and write data from or to standard MATLAB MAT-files. RSIM reads new signals and parameters from MAT-files at the start of simulation.

RSIM enables you to run stand-alone, fixed-step simulations on your host computer or on additional computers. If you need to run 100 large simulations, you can generate the RSIM model code, compile it, and run the executables on 10 identical computers. The RSIM target allows you to change the model parameters and the signal data, achieving significant speed improvements by using a compiled simulation.

S-Function and Accelerator Targets

S-Function Target provides the ability to transform a model into a Simulink S-function component. Such a component can then be used in a larger model. This allows you to speed up simulations and/or reuse code. You can include multiple instances of the same S-function in the same model, with each instance maintaining independent data structures. You can also share S-function components without exposing the details of the a proprietary source model.

The Accelerator Target is similar to the S-Function Target in that an S-function is created for a model. The Accelerator Target differs from the S-Function Target in that the generated S-function operates in the background. It provides for faster simulations while preserving all existing simulation capabilities (parameter change, signal visualization, full S-function support, etc.).

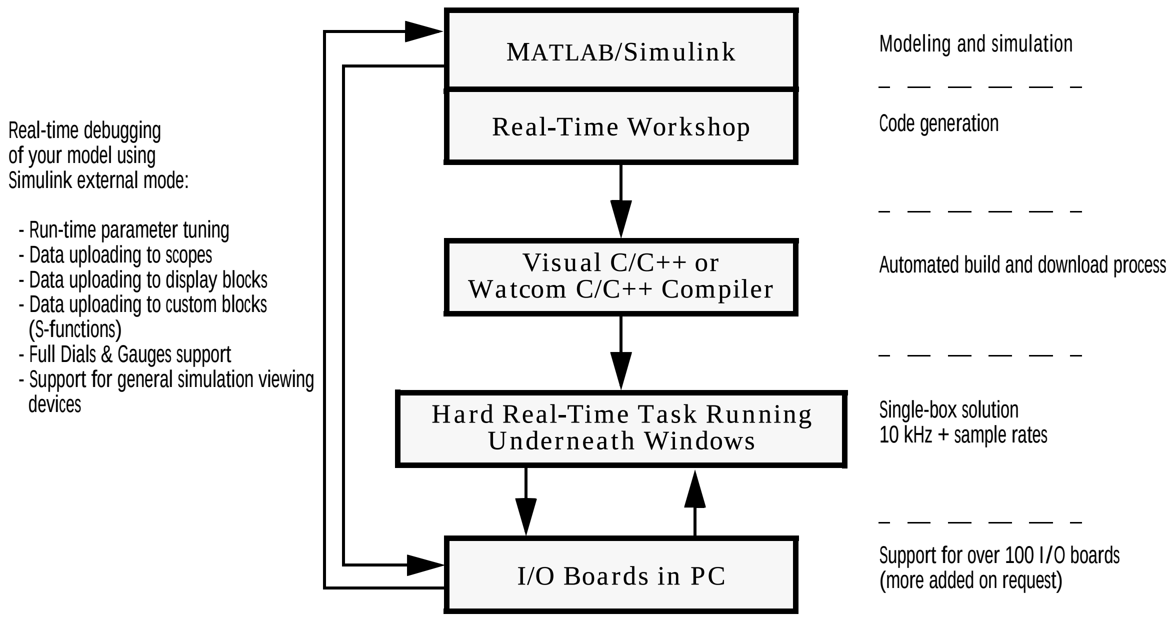
Turnkey Rapid Prototyping Targets

Real-Time Windows Target and xPC Target are add-on products to Real-Time Workshop. Both of these targets turn an Intel 80x86/Pentium or compatible PC into a real-time system. Both support a large selection of off-the-shelf I/O cards (both ISA and PCI).

With turnkey target systems, all you need to do is install the MathWorks software and a compiler, and insert the I/O cards. You can then use a PC as a real-time system connected to external devices via the I/O cards.

Real-Time Windows Target. Real-Time Windows Target brings rapid prototyping and hardware-in-the-loop simulation to your desktop. It is the most portable solution available today for rapid prototyping and hardware-in-the-loop simulation when used on a laptop outfitted with a PCMCIA I/O card. Real-Time Windows Target is ideal since a second PC or other real-time hardware is often unnecessary, impractical or cumbersome.

This picture shows the basic components of the Real-Time Windows Target.


MATLAB/Simulink
Real-Time Workshop
Visual C/C++ or Watcom C/C++ Compiler
Hard Real-Time Task Running Underneath Windows
I/O Boards in PC
Modeling and simulation
Code generation
Automated build and download process
Single-box solution
10 kHz + sample rates
Support for over 100 I/O boards (more added on request)
Real-time debugging of your model using Simulink external mode:
- Run-time parameter tuning
- Data uploading to scopes
- Data uploading to display blocks
- Data uploading to custom blocks (S-functions)
- Full Dials & Gauges support
- Support for general simulation viewing devices

As a prototyping environment, Real-Time Windows Target is exceptionally easy to use, due to tight integration with Simulink and external mode. It is much like using Simulink itself, with the added benefit of gaining real-time performance and connectivity to the real world through a wide selection of supported I/O boards. You can control your real-time execution with buttons located on the Simulink toolbar. Parameter tuning is done “on-the-fly,” by simply editing Simulink blocks and changing parameter values. For viewing signals, Real-Time Windows Target uses standard Simulink Scope blocks, without any need to alter your Simulink block diagram. Signal data can also be logged to a file or set of files for later analysis in MATLAB.

Real-Time Windows Target is often called the “one-box rapid prototyping system,” since both Simulink and the generated code run on the same PC. A run-time interface enables you to run generated code on the same processor that runs Windows NT or Windows 95/98/2000. The generated code executes in hard real-time, allowing Windows to execute when there are free CPU cycles. Real-Time Windows Target supports over 100 I/O boards, including ISA, PCI, CompactPCI, and PCMCIA. Sample rates in excess of 10 to 20 kHz can be achieved on Pentium PCs.

In universities, Real-Time Windows Target provides a cost effective solution since only a single computer is required. In commercial applications, Real-Time Windows Target is often used at an engineer's desk prior to taking a project to an expensive dedicated real-time testing environment. Its portability is unrivaled, allowing you to use your laptop as a real-time test bed for applications in the field.

Figure 1-6 illustrates the use of Real-Time Windows Target in a model using magnetic levitation to suspend a metal ball in midair. The system is controlled by the model shown in Figure 1-7.

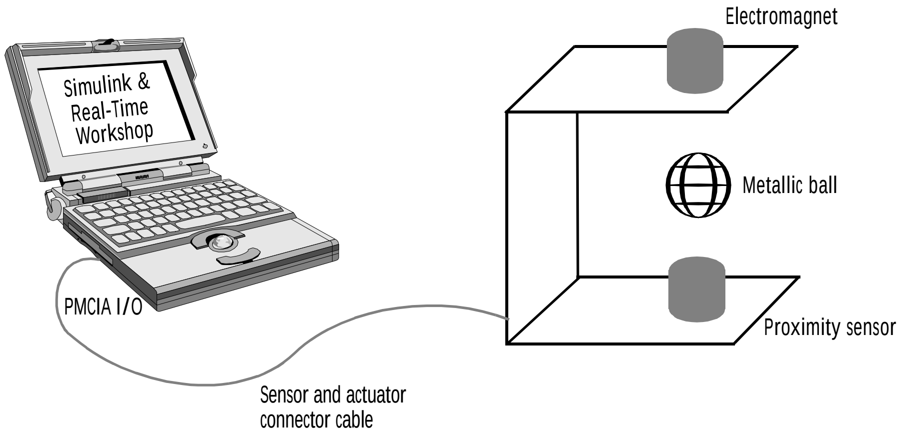


Figure 1-6: Magnetic Levitation System

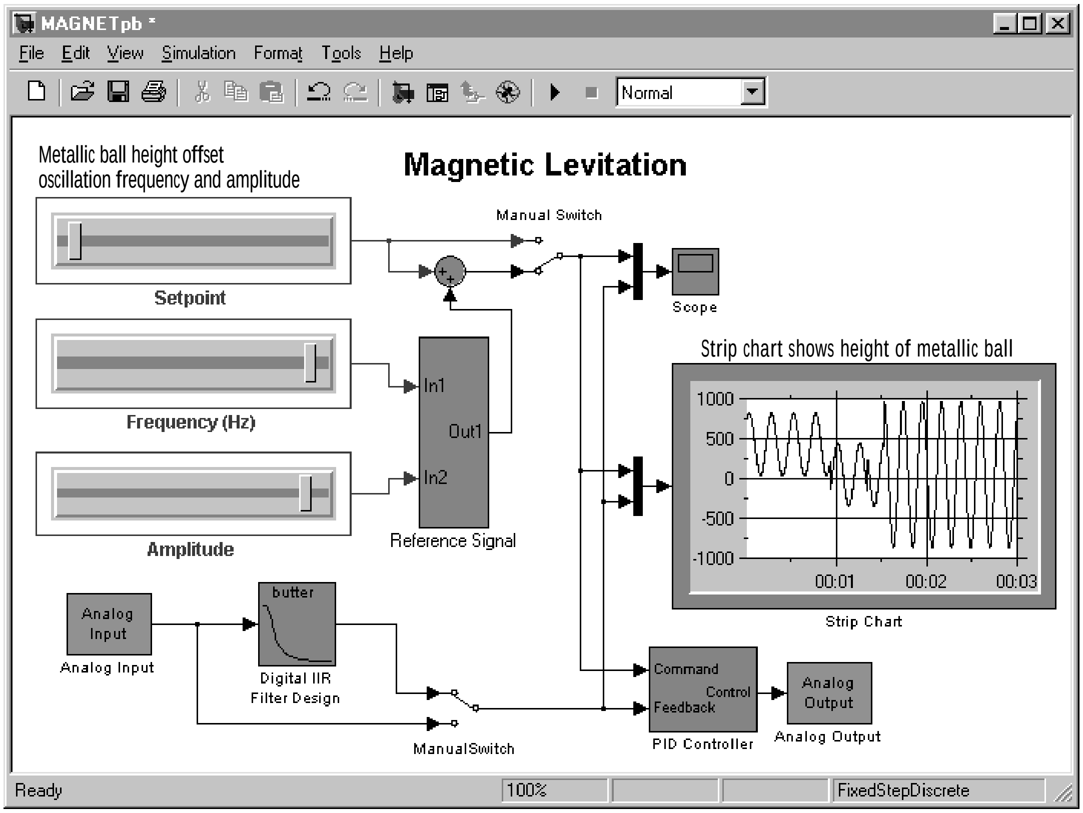


Figure 1-7: Model for Controlling Magnetic Levitation System

xPC Target. xPC Target is often referred to as a two-box solution. xPC Target requires two PCs: a host PC to run Simulink, and a target PC to run the generated code. The target PC runs an extremely compact real-time kernel that uses 32-bit protected mode. Communication between the host and the target is supported either via an Ethernet networking connection or via a serial cable. Figure 1-8 illustrates the XPC target in a rapid prototyping environment.

Since the target PC is dedicated to running the generated code, xPC Target achieves both increased performance and increased system stability. The target PC is required to have a PC-compatible architecture, but can have any of several form factors, including PC motherboards, CompactPCI,PC104, and single-board computers (SBCs).

xPC Target is also useful in limited production environments. Given the cost of PC hardware, it may make sense to deploy xPC Target in low volume or high-end production systems. This is achieved by using the xPC Target Embedded Option, which is an add-on to xPC Target.

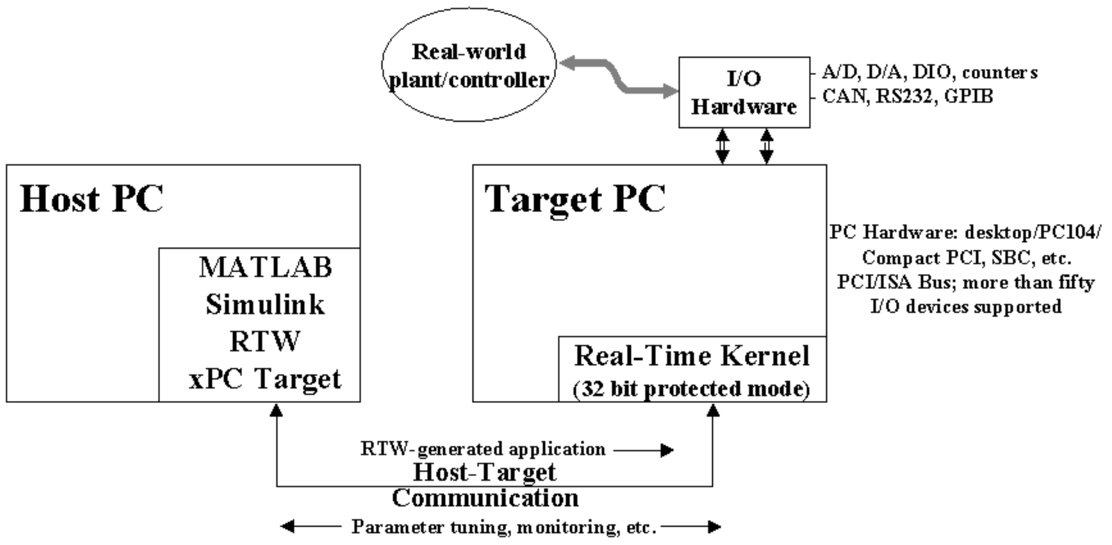

Real-world plant/controller
I/O Hardware
A/D, D/A, DIO, counters
CAN, RS232, GPIB
Host PC
MATLAB
Simulink
RTW
xPC Target
Target PC
PC Hardware: desktop/PC104/ Compact PCI, SBC, etc. PCI/ISA Bus; more than fifty I/O devices supported
Real-Time Kernel (32 bit protected mode)
RTW-generated application
Host-Target Communication
Parameter tuning, monitoring, etc.

Figure 1-8: xPC Target Rapid Prototyping Environment

Rapid Prototyping Targets

There are two classes of rapid prototyping targets: those using the real-time code format and those using the real-time malloc code format. These differ in the way they allocate memory (statically versus dynamically). Most rapid prototyping targets use the real-time code format.

We define two forms of rapid prototyping environments:

- *Heterogeneous rapid prototyping* environments use rapid prototyping hardware (such as an Intel-80x86/Pentium or similar processor) that differs from the final production hardware. For example, an Intel-80x86/Pentium or similar processor might be used during rapid prototyping of a system that is eventually deployed onto a fixed-point Motorola microcontroller.

• *Homogeneous rapid prototyping* environments are characterized by the use of similar hardware for the rapid prototyping system and the final production system. The main difference is that the rapid prototyping system has extra memory and/or interfacing hardware to support increased debugging capabilities, such as communication with external mode.

Homogeneous rapid prototyping environments eliminate uncertainty because the rapid prototyping environment is closer to the final production system. However, a turnkey system for your specific hardware may not exist. In this case, you must weigh the advantages and disadvantages of using one of the existing turnkey systems for heterogeneous rapid prototyping, versus creating a homogeneous rapid prototyping environment.

Several rapid prototyping targets are bundled with Real-Time Workshop.

Generic Real-Time (GRT) Target. This target uses the real-time code format and supports external mode communication. It is designed to be used as a starting point when creating a custom rapid prototyping target, or for validating the generated code on your workstation.

Generic Real-Time Malloc (GRTM) Target. This target is similar to the GRT target but it uses the real-time malloc code format. This format uses the C `malloc` and `free` routines to manage all data. With this code format, you can have multiple instances of your model and/or multiple models in one executable.

Tornado Target. The Tornado target uses the real-time or real-time malloc code format. A set of run-time interface files are provided to execute your models on the Wind River System's real-time operating system, VxWorks. The Tornado target supports singletasking, multitasking, and hybrid continuous and discrete-time models.

The Tornado run-time interface and device driver files can also be used as a starting point when targeting other real-time operating system environments. The run-time interface provides full support for external mode, enabling you to take full advantage of the debugging capabilities for parameter tuning and data monitoring via graphical devices.

DOS Target. The DOS target (provided as an example only) uses the real-time code format to turn a PC running the DOS operating system into a real-time system. This target includes a set of run-time interface files for executing the generated code. This run-time interface installs interrupt service routines to execute the generated code and handle other interrupts. While the DOS target is running, the user does not have access to the DOS operating system. Sample device drivers are provided.

The MathWorks recommends that you use Real-Time Windows Target or xPC Target as alternatives to the DOS Target. The DOS target is provided only as an example and its support will be discontinued in the future.

OSEK Targets. The OSEK target (provided as an example only) lets you use the automotive standard open real-time operating system. The run-time interface and OSEK configuration files that are included with this target make it easy to port applications to a wide range of OSEK environments.

Embedded Targets

The embedded real-time target is the main component of the Real-Time Workshop Embedded Coder. It consists of a set of run-time interface files that drive code, generated in the embedded code format, on your workstation. This target is ideal for memory-constrained embedded applications. Real-Time Workshop supports generation of embedded code in both C and Ada.

In its default configuration, the embedded real-time target is designed for use as a starting point for targeting custom embedded applications, and as a means by which you can validate the generated code. To create a custom embedded target, you start with the embedded real-time target run-time interface files and edit them as needed for your application.

In the terminology of Real-Time Workshop, an embedded target is a deeply embedded system. Note that it is possible to use a rapid prototyping target in an embedded (production) environment. This may make more sense in your application.

Code Generation Optimizations

The Simulink code generator included with Real-Time Workshop is packed with optimizations to help create fast and minimal size code. The optimizations are classified either as cross-block optimizations, or block specific optimizations. Cross-block optimizations apply to groups of blocks or the general structure of a model. Block specific optimizations are handled locally by the object generating code for a given block. Listing each block specific optimization here is not practical; suffice it to say that the Target Language Compiler technology generates very tight and fast code for each block in your model.

The following sections discuss some of the cross-block optimizations.

Multirate Support

One of the more powerful features of Simulink is its implicit support for multirate systems. The ability to run different parts of a model at different rates guarantees optimal use of the target processor. In addition, Simulink enforces correctness by requiring that you create your model in a manner that guarantees deterministic execution.

Inlining S-Function Blocks for Optimal Code

The ability to add blocks to Simulink via S-functions is enhanced by the Target Language Compiler. You can create blocks that embed the minimal amount of instructions into the generated code. For example, if you create a device driver using an S-function, you can have the generated code produce one line for the device read, as in the following code fragment.

```
mdlOutputs(void)
{
    .
    .
    rtB.deviceout = READHW(); /* Macro to read hw device using
    .                            assembly code */
    .
}
```

Note that the generic S-function API is suitable for any basic block-type operation.

Loop Rolling Threshold

The code generated for blocks can contain `for` loops, or the loop iterations can be “flattened out” into inline statements. For example, the general gain block equation is

```
for (i = 0; i < N; i++) {
    y[i] = k[i] * u[i];
}
```

If `N` is less than a specified roll threshold, Real-Time Workshop expands out the `for` loop, otherwise Real-Time Workshop retains the `for` loop.

Tightly Coupled Optimal Stateflow Interface

The generated code for models that combine Simulink blocks and Stateflow charts is tightly integrated and very efficient.

Stateflow Optimizations

The Stateflow Coder contains a large number of optimizations that produce highly readable and very efficient generated code.

Inlining of Systems

In Simulink, a system starting at a nonvirtual subsystem boundary (e.g. an enabled, triggered, enabled and triggered, function-call, or atomic subsystem) can be inlined by selecting the **RTW inline subsystem** option from the subsystem block properties dialog. The default action is to inline the subsystem, unless it is a function-call subsystem with multiple callers.

Block I/O Reuse

Consider a model with a D/A converter feeding a gain block (for scaling), then feeding a transfer function block, then feeding a A/D block. If all signals refer to the same memory location, then less memory will be used. This is referred to as block I/O reuse. It is a powerful optimization technique for re-using memory locations. It reduces the number of global variables improving the executing speed (faster execution) and reducing the size of the generated code.

Declaration of Block I/O Variables in Local Scope

If input/output signal variables are not used across function scope, then they can be placed in local scope. This optimization technique reduces code size and improves the execution speed (faster execution).

Inlining of Parameters

If you select the **Inline parameters** option, the numeric values of block parameters that represent coefficients are embedded in the generated code. If **Inline parameters** is off, block parameters that represent coefficients can be changed while the model is executing.

Note that it is still possible to "tune" key parameters by using the **Workspace parameter attributes** dialog.

Inlining of Invariant Signals

An invariant signal is a block output signal that does not change during Simulink simulation. For example, the output of a sum block that is fed by two constants cannot change. When **Inline invariant signals** is specified, a single numeric value is placed in the generated code to represent the output value of the sum block. The **Inline invariant signals** option is available when the **Inline parameters** option is on.

Parameter Pooling

The **Parameter pooling** option is available when **Inline parameters** is selected. If Real-Time Workshop detects identical usage of parameters (e.g. two lookup tables with same tables), it will pool these parameters together, thereby reducing code size.

Block Reduction Optimizations

Real-Time Workshop can detect block patterns (e.g. an accumulator represented by a constant, sum and a delay block) and reduce these patterns to a single operation, resulting in very efficient generated code.

Creation of Contiguous Signals to Speed Block Computations

Some block algorithms (for example a matrix multiply) can be implemented more efficiently if the signals entering the blocks are contiguous. Noncontiguous signals occur because of the handling of virtual blocks. For example, the output of a Mux block is noncontiguous. When this class of block requires a contiguous signal, Simulink will insert (if needed) a copy block operator to make the signal contiguous. This results in better code efficiency.

Support for Noncontiguous Signals by Blocks

Noncontiguous signals occur because of the block virtualization capabilities of Simulink. For example, the output of a Mux block is generally a noncontiguous signal (i.e., the output signal consists of signals from multiple sources). General blocks in Simulink support this behavior by generating very efficient code to handle each different signal source in a noncontiguous signal.

Data Type Support

Simulink models support a wide range of data types. You can use double precision values to represent real-world values and then when needed use integers or Booleans for discrete valued signals. You can also use fixed-point (integer scaling) capabilities to target models for fixed-point embedded processors. The wide selection of data types in Simulink models enables you to realize your models efficiently.

Frame Support

In signal processing, a frame of data represents time sampled sequences of an input. Many devices have support in hardware for collecting frames of data. With Simulink and the DSP Blockset, you can use frames and perform frame based operations on the data. Frames are a very efficient way of handling high frequency signal processing applications.

Matrix Support

Most blocks in Simulink support the use of matrices. This enables you to create models that represent high levels of abstractions and produce very efficient generated code.

Virtualization of Blocks

Nearly half of the blocks in a typical model are connection type blocks (e.g. Virtual Subsystem, Inport, Outport, Goto, From, Data Store Memory, Selector, Bus Selector, Mux, Demux, Ground, and Terminator). These blocks are provided to enable you to create complex models with your desired levels of abstraction. Simulink treats these blocks as virtual, meaning that they impose no overhead during simulation or in the generated code.

Open and Extensible Modeling Environment

The Simulink / Real-Time Workshop environment is extensible in several ways.

Custom Code Support

S-functions are dynamically linked objects (.DLL or .so) that bind with Simulink to extend the modeling environment. By developing S-functions, you can add custom block algorithms to Simulink. Such S-functions provide supporting logic for the model. S-functions are flexible, allowing you to implement complex algorithmic equations or basic low-level device drivers. The Real-Time Workshop support for S-functions includes the ability to inline S-function code directly into the generated code. Inlining, supported by the Target Language Compiler, can significantly reduce memory usage and calling overhead.

Support for Supervisory Code

The generated code implements an algorithm that corresponds exactly to the algorithm defined in your model. With the embedded code format, you can call the generated model code as a procedure. This enables you to incorporate the generated code into larger systems that decide when to execute the generated code. Conceptually, you can think of the generated code as set of equations, wrapped in a function called by your supervisory code. This facilitates integration of model code into large existing systems, or into environments that consist of more than signal-flow processing (Simulink) and state machines (Stateflow).

Monitoring and Parameter Tuning APIs

External mode provides a communication channel for interfacing the generated code running on your target with Simulink. External mode lets you use Simulink as a debugging front end for an executing model. Typically, the external mode configuration works in conjunction with either the real-time code format or the real-time malloc code format.

The Real-Time Workshop provides other mechanisms for making model signals and block parameters visible to your own monitoring and tuning interfaces. These mechanisms, suitable for use on all code formats, include:

- The **Model Parameter Configuration** dialog enables you to declare how to allocate memory for variables that are used in your model. For example, if a Gain block contains the variable k, you can declare k as an external variable, a pointer to an external variable, a global variable, or let the Real-Time Workshop decide where and how to declare the variable.

  The **Model Parameter Configuration** feature enables you to specify block parameters as tunable or global. This gives your supervisory code complete access to any block parameter variables that you may need to alter while your model is executing. You can also use this feature to interface parameters to specific constant read-only memory locations.

- You can mark signals in your model as *test points*. Declaring a test point indicates that you may want to see the signal's value while the model is executing. After marking a signal as a test point, you specify how the memory for the signal is to be allocated. This gives your supervisory code complete read-only access to signals in your model, so that you can monitor the internal workings of your model.
- C and Target Language Compiler APIs provide another form of access to the signals and parameters in your model. The Target Language Compiler API is a means to access the internal signals and parameters during code generation. With this information, you can generate monitoring/tuning code that is optimized specifically for your model or target.

Interrupt Support

Interrupt blocks enable you to create models that handle synchronous and asynchronous events, including interrupt service routines (ISRs), hardware-generated interrupts, and asynchronous read and write operations. The blocks provided work with the Tornado target. You can use these blocks as templates when creating new interrupt blocks for your target environment. Interrupt blocks include:

- Asynchronous Interrupt block
- Task Synchronization block
- Asynchronous Buffer block (read)
- Asynchronous Buffer block (write)
- Asynchronous Rate Transition block

Custom Code Library

The Custom Code library contains a set of blocks that allow you to easily insert target-specific code into your model without the need of an inlined S-function. Code can be placed strategically throughout the model.

Getting Started: Basic Concepts and Tutorials

This section is designed to help you to obtain hands-on experience with the Real-Time Workshop in short order. There are two parts, which we recommend you read in order:

1. "Basic Real-Time Workshop Concepts" on page 1-37 introduces concepts and terminology you should know before working with the Real Time Workshop.
2. "Quick Start Tutorials" on page 1-40 provides several hands-on exercises that demonstrate the Real-Time Workshop user interface, code generation and build process, and other essential features.

Basic Real-Time Workshop Concepts

Target and Host

A *target* is an environment — hardware or operating system — on which your generated code will run. The process of specifying this environment is called *targeting*.

The process of generating target-specific code is controlled by a *system target file*, a *template makefile*, and a *make command*. To select a desired target, you can specify these items individually, or you can choose from a wide variety of ready-to-run configurations.

The *host* is the system you use to run MATLAB, Simulink, and the Real-Time Workshop. Using the build tools on the host, you create code and an executable that runs on your target system.

Available Target Configurations

The Real-Time Workshop supports many target environments. These include ready-to-run configurations and third-party targets. You can also develop your own custom target.

For a complete list of bundled targets, with their associated system target files and template makefiles, see "The System Target File Browser" on page 3-34.

Code Formats

A *code format* specifies a framework for code generation suited for specific applications.

When you choose a target configuration, you implicitly choose a code format. If you use the Real-Time Workshop Embedded Coder, for example, the code generated will be in *embedded C* format. The embedded C code format is a compact format designed for production code generation. Its small code size, memory usage, and simple call structure make it optimal for embedded applications.

Many other targets, such as the generic real-time (GRT) target, use the *real-time* code format. This format, less compact but more flexible, is optimal for rapid prototyping applications.

For a complete discussion of available code formats, see Chapter 4, “Generated Code Formats.”

The Generic Real-Time Target

The Real-Time Workshop provides a generic real-time development target. The generic real-time (GRT) target provides an environment for simulating fixed-step models in single or multitasking mode. A program generated with the GRT target runs your model, in simulated time, as a stand-alone program on your workstation.

The GRT target allows you to perform code validation by logging system outputs, states, and simulation time to a data file. The data file can then be loaded into the MATLAB workspace for analysis or comparison with the output of the original model.

The GRT target also provides a starting point for targeting custom hardware. You can modify the GRT harness program, `grt_main.c`, to execute code generated from your model at interrupt level under control of a real-time clock.

Target Language Compiler Files

The Real-Time Workshop uses *Target Language Compiler files* (or *TLC files)* to translate your Simulink model into code. The Target Language Compiler uses two types of TLC files during the code generation and build process. The *system target file*, which describes how to generate code for a chosen target, is the entry point for the TLC program that creates the executable. *Block target files* define how the code looks for each of the Simulink blocks in your model.

System and block target files have the extension `.tlc`. By convention, a system target file has a name corresponding to your target. For example, `grt.tlc` is the system target file for the generic real-time (GRT) target.

Template Makefiles

The Real-Time Workshop uses template makefiles to build an executable from the generated code.

The Real-Time Workshop build process creates a makefile from the template makefile. Each line from the template makefile is copied into the makefile; tokens encountered during this process are expanded into the makefile.

The name of the makefile created by the build process is *model*`.mk`. The *model*`.mk` file is passed to a make utility. The make utility compiles and links an executable from a set of files.

By convention, a template makefile has an extension of `.tmf` and a name corresponding to your target and compiler. For example, `grt_vc.tmf` is the template makefile for building a generic real-time program under Visual C/C++.

The Build Process

A high-level M-file command controls the Real-Time Workshop build process. The default command, used with most targets, is `make_rtw`.When you initiate a build, the Real-Time Workshop invokes `make_rtw`.The `make_rtw` command, in turn, invokes the Target Language Compiler and other utilities such as `make`. The build process consists of the following stages:

1. First, `make_rtw` compiles the block diagram and generates a model description file, *model*`.rtw`.

2. Next, `make_rtw` invokes the Target Language Compiler to generate target-specific code, processing *model*`.rtw` as specified by the selected system target file.

3. Next, `make_rtw` creates a makefile, *model*`.mk`, from the selected template makefile.

4. Finally, `make` is invoked. `make` compiles and links a program from the generated code, as instructed in the generated makefile.

"Automatic Program Building" in Chapter 2 gives an overview of the build process. Chapter 3, "Code Generation and the Build Process" gives full details on code generation and build options and parameters.

Model Parameters and Code Generation

The simulation parameters of your model directly affect code generation and program building. For example, if your model is configured to stop execution after 60 seconds, the program generated from your model will also run for 60 seconds. The Real-Time Workshop also imposes certain requirements and restrictions on the model from which code is generated.

Before you generate code and build an executable, you must verify that you have set the model parameters correctly in the **Simulation Parameters** dialog box. See "Simulation Parameters and Code Generation" in Chapter 3 for more information.

Quick Start Tutorials

This section provides hands-on experience with the code generation, program building, data logging, and code validation capabilities of the Real-Time Workshop.

"Tutorial 1: Building a Generic Real-Time Program" on page 1-42 shows how to generate C code from a Simulink demonstration model and build an executable program.

"Tutorial 2: Data Logging" on page 1-49 explains how to modify the demonstration program to save data in a MATLAB MAT-file, for plotting.

"Tutorial 3: Code Validation" on page 1-52, demonstrates how to validate the generated program by comparing its output to that of the original model.

"Tutorial 4: A First Look at Generated Code" on page 1-56 examines code generated from a very simple model, illustrating the effect of one of the Real-Time Workshop code generation options.

These tutorials assume basic familiarity with MATLAB and Simulink. You should also read "Getting Started: Basic Concepts and Tutorials" on page 1-37 before proceeding.

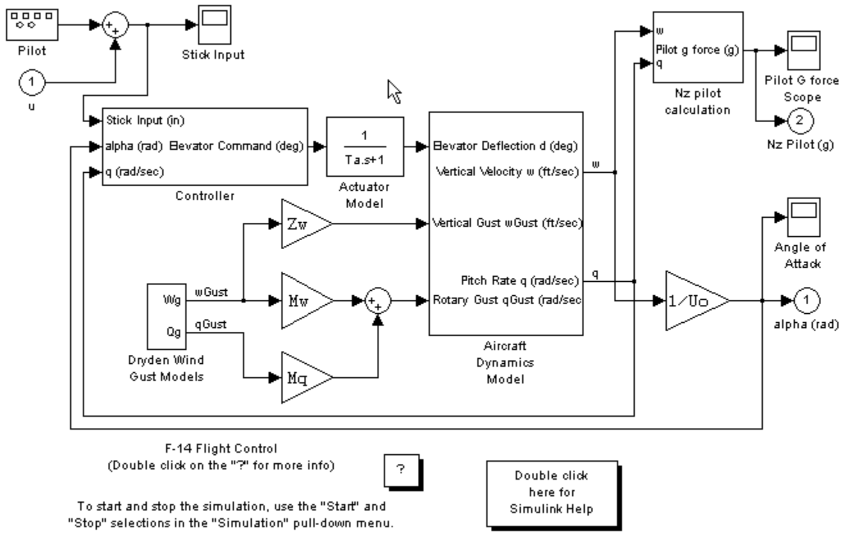
The procedures for building, running, and testing your programs are almost identical in UNIX and PC environments. The discussion notes differences where applicable.

Make sure that a MATLAB compatible C compiler is installed on your system before proceeding with these tutorials. See the "Preface" for more information on supported compilers and compiler installation.

The f14 Demonstration Model

Tutorials 1-3 use a demonstration Simulink model, `f14.mdl`, from the *matlabroot*`/toolbox/simulink/simdemos/aerospace` directory. (By default, this directory is on your MATLAB path.) `f14` is a model of a flight controller for the longitudinal motion of a Grumman Aerospace F-14 aircraft.

This is the `f14` model.



The model simulates the pilot's stick input with a square wave having a frequency of 0.5 (radians per second) and an amplitude of ± 1. The system outputs are the aircraft angle of attack and the G forces experienced by the pilot. The input and outputs are visually monitored by Scope blocks.

Tutorial 1: Building a Generic Real-Time Program

This tutorial walks through the process of generating C code and building an executable program from the demonstration model. The resultant stand-alone program runs on your workstation, independent of external timing and events.

Working and Build Directories

It is convenient to work with a local copy of the `f14` model, stored in its own directory, `f14example`. This discussion assumes that the `f14example` directory resides on drive `d:`. Set up your working directory as follows:

1 Create the directory from the MATLAB command line by typing

   `!mkdir d:\f14example` (on PC)

   or

   `!mkdir ~/f14example` (on UNIX)

2 Make `f14example` your working directory.

   ```
   cd d:/f14example
   ```

3 Open the `f14` model.

   ```
   f14
   ```

   The model appears in the Simulink window.

4 From the **File** menu, choose **Save As**. Save a copy of the `f14` model as `d:/f14example/f14rtw.mdl`.

Be aware that during code generation, the Real-Time Workshop creates a *build directory* within your working directory. The build directory name is *model_target*`_rtw`, derived from the name of the source model and the chosen target. The build directory stores generated source code and other files created during the build process. We examine the build directory and its contents at the end of this tutorial.

Setting Program Parameters

To generate code correctly from the `f14rtw` model, you must change some of the simulation parameters. In particular, note that the Real-Time Workshop requires the use of a fixed-step solver. To set parameters, use the **Simulation Parameters** dialog box as follows:

1 From the **Simulation** menu, choose **Simulation Parameters**. The **Simulation Parameters** dialog box opens.

2 Click the Solver tab and enter the following parameter values on the Solver page.

   Start Time: `0.0`

   Stop Time: `60`

   Solver options: set **Type** to `Fixed-step`. Select the `ode5 (Dormand-Prince)` solver algorithm.

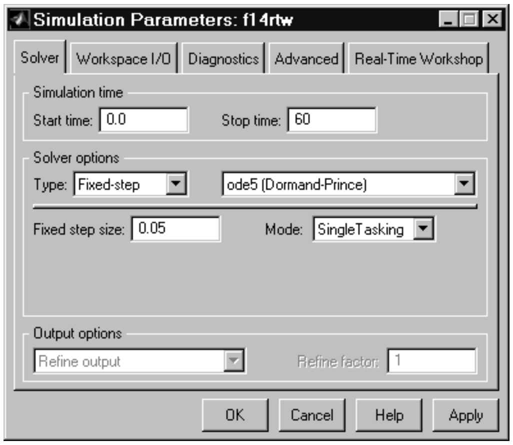
   Fixed step size: `0.05`

   Mode: `Single Tasking`

3 Click **Apply**. Then click **OK** to close the dialog box.

4 Save the model. Simulation parameters persist with the model, for use in future sessions.

Figure 1-9 shows the Solver page with the correct parameter settings.


Simulation Parameters: f14rtw

Solver | Workspace I/O | Diagnostics | Advanced | Real-Time Workshop

Simulation time
Start time: 0.0   Stop time: 60

Solver options
Type: Fixed-step   ode5 (Dormand-Prince)
Fixed step size: 0.05   Mode: SingleTasking

Output options
Refine output   Refine factor: 1

OK | Cancel | Help | Apply

Figure 1-9: Solver Page of Simulation Parameters Dialog Box

Selecting the Target Configuration

To specify the desired target configuration, you choose a system target file, a template makefile, and a `make` command.

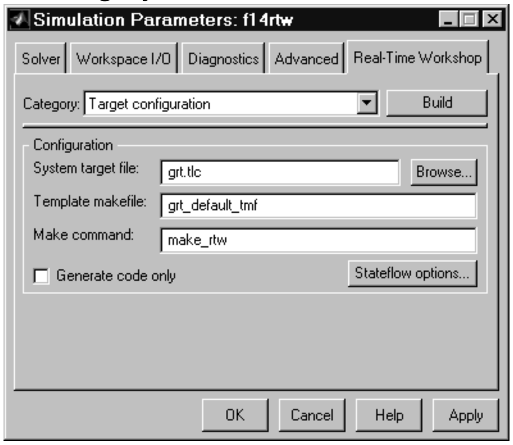
In these tutorials, you do not need to specify these parameters individually. Instead, you use the ready-to-run generic real-time (GRT) target configuration. The GRT target is designed to build a stand-alone executable program that runs on your workstation.

To select the GRT target:

1 From the **Simulation** menu, choose **Simulation Parameters**. The **Simulation Parameters** dialog box opens.

2 Click on the Real-Time Workshop tab of the **Simulation Parameters** dialog box. The Real-Time Workshop page activates.

3 The Real-Time Workshop page has several sub-pages, which are selected via the **Category** pull-down menu. Select **Target configuration** from the **Category** menu.



Category: Target configuration

System target file: grt.tlc

Template makefile: grt_default_tmf

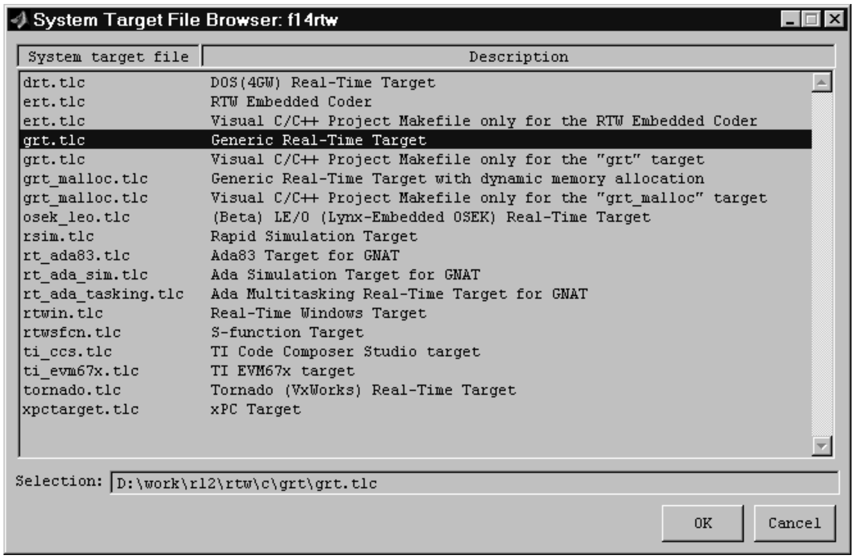
Make command: make_rtw

☐ Generate code only

Figure 1-10: Real-Time Workshop Page (Target Configuration Category)

4 Click on the **Browse** button next to the **System target file** field. This opens the System Target File Browser. The browser displays a list of all currently available target configurations. When you select a target configuration, the

Real-Time Workshop automatically chooses the appropriate system target file, template makefile, and make command.



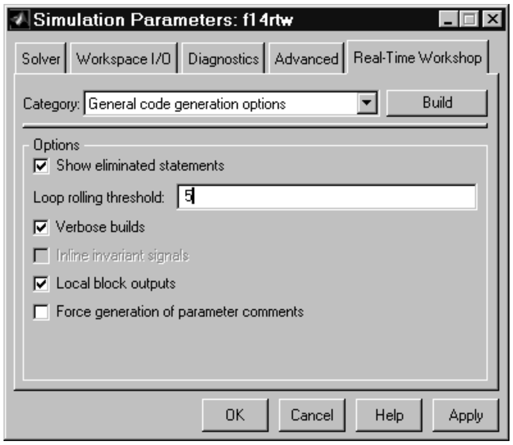
```
System target file     Description
drt.tlc                DOS(4GW) Real-Time Target
ert.tlc                RTW Embedded Coder
ert.tlc                Visual C/C++ Project Makefile only for the RTW Embedded Coder
grt.tlc                Generic Real-Time Target
grt.tlc                Visual C/C++ Project Makefile only for the "grt" target
grt_malloc.tlc         Generic Real-Time Target with dynamic memory allocation
grt_malloc.tlc         Visual C/C++ Project Makefile only for the "grt_malloc" target
osek_leo.tlc           (Beta) LE/O (Lynx-Embedded OSEK) Real-Time Target
rsim.tlc               Rapid Simulation Target
rt_ada83.tlc           Ada83 Target for GNAT
rt_ada_sim.tlc         Ada Simulation Target for GNAT
rt_ada_tasking.tlc     Ada Multitasking Real-Time Target for GNAT
rtwin.tlc              Real-Time Windows Target
rtwsfcn.tlc            S-function Target
ti_ccs.tlc             TI Code Composer Studio target
ti_evm67x.tlc          TI EVM67x target
tornado.tlc            Tornado (VxWorks) Real-Time Target
xpctarget.tlc          xPC Target

Selection: D:\work\r12\rtw\c\grt\grt.tlc
```

Figure 1-11: The System Target File Browser

5 From the list of available configurations, select **Generic Real-Time Target** (as in Figure 1-11) and then click **OK**.

6 The Real-Time Workshop page now displays the correct system target file (`grt.tlc`), template makefile (`grt_default_tmf`), and make command (`make_rtw`), as in Figure 1-10.

7 Select **General code generation options** from the **Category** menu. The options displayed here are common to all target configurations. Check to make sure that all options are set to their defaults, as below.



Category: General code generation options

Options

[✓] Show eliminated statements

Loop rolling threshold: 5

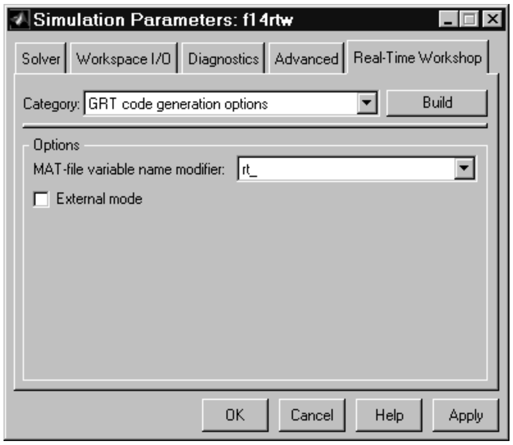
[✓] Verbose builds

[ ] Inline invariant signals

[✓] Local block outputs

[ ] Force generation of parameter comments

8 Select **GRT code generation options** from the **Category** menu. The options displayed here are specific to the GRT target. Check to make sure that all options are set to their defaults, as below.



Category: GRT code generation options

Options

MAT-file variable name modifier: rt_

[ ] External mode

9 Select **TLC debugging** from the **Category** menu. Make sure that all options in this category are deselected.

10 Select **Target configuration** from the **Category** menu. Make sure that the **Generate code only** option is deselected.

11 Save the model.

Building and Running the Program

The Real-Time Workshop build process generates C code from the model, and then compiles and links the generated program. To build and run the program:

1 Click the **Build** button in the **Simulation Parameters** dialog box to start the build process.

2 A number of messages concerning code generation and compilation appear in the MATLAB command window. The initial messages are

```
### Starting Real-Time Workshop build procedure for model: f14rtw
### Generating code into build directory: .\f14rtw_grt_rtw
```

The content of the succeeding messages depends on your compiler and operating system.The final message is

```
### Successful completion of Real-Time Workshop build procedure
for model: f14rtw
```

3 The working directory now contains an executable, `f14rtw.exe` (on PC), or `f14rtw` (on UNIX). In addition, a build directory, `f14rtw_grt_rtw`, has been created.

To observe the contents of the working directory after the build, type the `dir` command from the MATLAB command window.

```
dir
.               f14rtw.exe      f14rtw_grt_rtw
..              f14rtw.mdl
```

4 To run the executable from the MATLAB command window, type:

```
!f14rtw
```

The “!” character passes the command that follows it to the operating system, which runs the stand-alone `f14rtw` program.

The program produces one line of output.

```
**starting the model**
```

5 Finally, to see the contents of the build directory, type

```
dir f14rtw_grt_rtw
```

Contents of the Build Directory

The build process creates a build directory and names it *model*_*target*_`rtw`, concatenating the name of the source model and the chosen target. In this example, the build directory is named `f14rtw_grt_rtw`.

`f14rtw_grt_rtw` contains these generated source code files:

- `f14rtw.c` — the stand-alone C code that implements the model.
- `f14rtw.h` — an include header file containing information about the state variables
- `f14rtw_export.h` — an include header file containing information about exported signals and parameters

The build directory also contains other files used in the build process, such as the object (`.obj`) files and the generated makefile (`f14rtw.mk`).

Tutorial 2: Data Logging

The Real-Time Workshop MAT-file data logging facility enables a generated program to save system states, outputs, and simulation time at each model execution time step. The data is written to a MAT-file, named (by default) *model*`.mat`, where *model* is the name of your model. In this tutorial, data generated by the model `f14rtw` is logged to the file `f14rtw.mat`

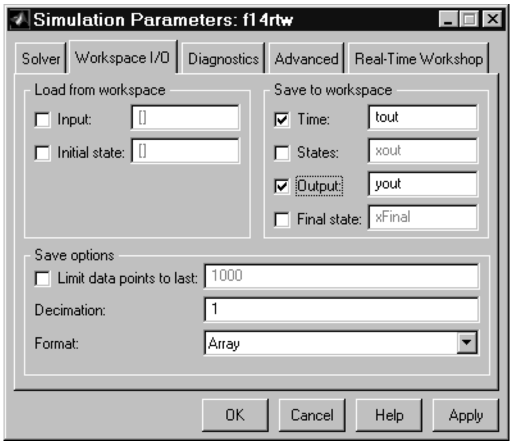
To configure data logging, you use the Workspace I/O page of the **Simulation Parameters** dialog. The process is nearly the same as configuring a Simulink model to save output to the MATLAB workspace. For each workspace return variable you define and enable, the Real-Time Workshop defines a parallel MAT-file variable. For example, if you save simulation time to the variable `tout`, your generated program logs the same data to a variable named (by default) `rt_tout`.

In this tutorial, you will modify the `f14rtw` model such that the generated program saves the simulation time and system outputs to the file `f14rtw.mat`.

Then, you will load the data into the MATLAB workspace and plot simulation time against one of the outputs.

To use the data logging feature:

1 Select the Workspace I/O page of the **Simulation Parameters** dialog box. The Workspace I/O page specifies how data is loaded from and saved to the workspace.



Load from workspace
- [ ] Input: []
- [ ] Initial state: []

Save to workspace
- [✓] Time: tout
- [ ] States: xout
- [✓] Output: yout
- [ ] Final state: xFinal

Save options
- [ ] Limit data points to last: 1000
- Decimation: 1
- Format: Array

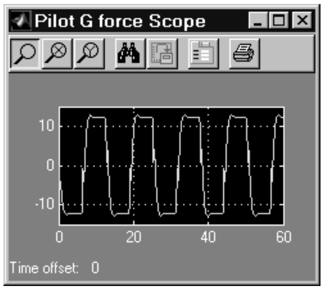
2 Check the **Time** option. Enabling the **Time** option causes the Real-Time Workshop to generate code that logs the simulation time to the MAT-file matrix `rt_tout`.

3 Check the **Output** option. Enabling the **Output** option causes the Real-Time Workshop to generate code that logs the root Output blocks (Angle of Attack and Pilot G Force) to the MAT-file matrix `rt_yout`.

   The sort order of the `rt_yout` array is based on the port number of the Outport blocks, starting with 1. Angle of Attack and Pilot G Force will be logged to `rt_yout(:,1)` and `rt_yout(:,2)`, respectively.

4 If any other options are enabled, uncheck them. Set **Decimation** to `1` and **Format** to `Array`. Then click **Apply**.

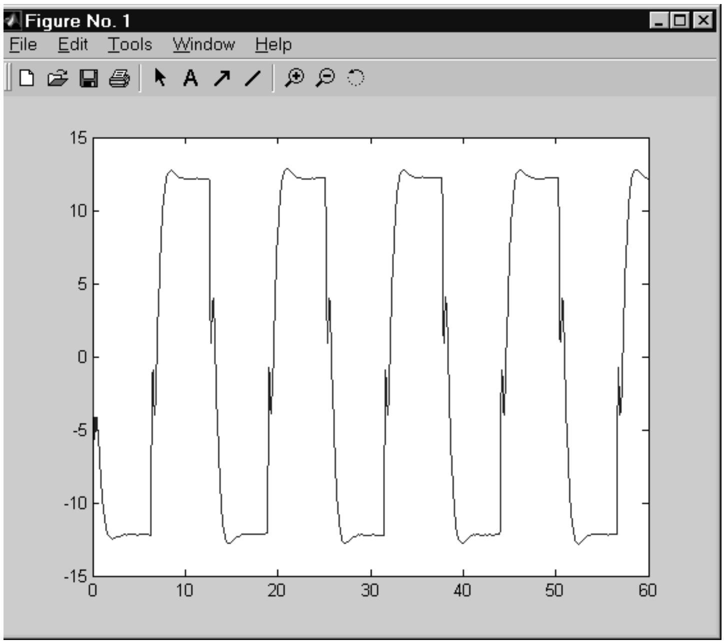
5 Open the Pilot G Force Scope block. To run the model, click on the **Start** button in the toolbar of the Simulink window. The Scope displays below:



6 Verify that the simulation time and Pilot G Force outputs have been correctly saved to the workspace by plotting simulation time versus Pilot G Force.

```
plot(tout,yout(:,2))
```

The resultant plot is shown below.



7 The `f14rtw` program must be rebuilt, because you have changed the model by enabling data logging. Select **Build Model** from the **Real-Time Workshop** submenu of the **Tools** menu in the Simulink window. This is an alternative way to start the Real-Time Workshop build process. It is identical to using **Build** button in the **Simulation Parameters** dialog box.

8 When the build concludes, run the executable with the command

```
!f14rtw
```

9 The program now produces two message lines, indicating that the MAT-file has been written.

```
**starting the model**
** created f14rtw.mat **
```

10 Clear the workspace, load the MAT-file data, and look at the workspace variables.

```
clear
load f14rtw.mat
whos
```

11 Observe that the variables `rt_tout` (time) and `rt_yout` (G Force and Angle of Attack) have been loaded from the file. Plot G Force as a function of time.

```
plot(rt_tout,rt_yout(:,2))
```

12 The plot should appear identical to the plot you produced in step 5 above.

Tutorial 3: Code Validation

In this tutorial, the code generated from the `f14rtw` model is validated against the model. The code is validated by capturing and comparing data from runs of the Simulink model and the generated program.

Note To obtain a valid comparison between outputs of the model and the generated program, make sure that you have selected the same integration scheme (`fixed-step`, `ode5 (Dormand-Prince)`) and the same step size (`0.05`) for both the Simulink run and the Real-Time Workshop build process. Also, make sure that the model is configured to save simulation time, as in Tutorial 2.

Logging Signals via Scope Blocks

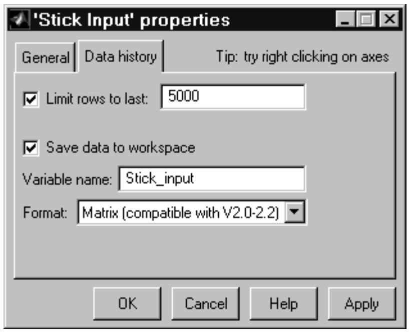
This example uses Scope blocks (rather than Outport blocks) to log both input and output data. To configure the Scope blocks to log data:

1 In the previous exercise, you cleared the workspace. Before proceeding with this tutorial, reload the model so that the proper workspace variables are declared and initialized.

```
f14rtw
```

2 Open the Stick Input Scope block and click on the **Properties** button on the toolbar of the Scope window. The **Scope Properties** dialog opens.

3 Select the **Data History** page of the **Scope Properties** dialog.



☑ Limit rows to last: 5000

☑ Save data to workspace

Variable name: Stick_input

Format: Matrix (compatible with V2.0-2.2)

4 Check the **Save data to workspace** option and enter the name of the variable (`Stick_input`) that is to receive the Scope data.

In the example above, the Stick Input signal will be logged to the matrix `Stick_input` during simulation. The generated code will log the same signal data to the MAT-file variable `rt_Stick_input` during a run of the executable program.

5 Click the **Apply** button.

6 Configure the Pilot G Force and Angle of Attack Scope blocks similarly, using the variable names `Pilot_G_force` and `Angle_of_attack`.

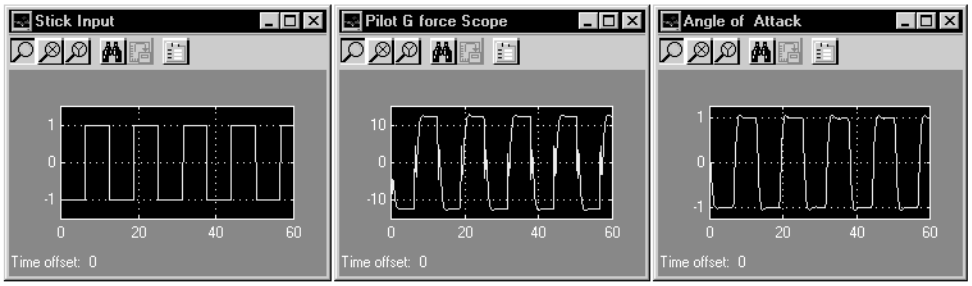
7 Save the model.

Logging Simulation Data

The next step is to run the simulation and log the signal data from the Scope blocks:

1 Open the Stick Input, Pilot G Force, and Angle of Attack Scope blocks.

2 Run the model. The Scope blocks display.



3 Use the whos command to observe that the matrix variables `Stick_input`, `Pilot_G_force`, and `Angle_of_attack` have been saved to the workspace.

4 Plot one or more of the logged variables against simulation time. For example,

```
plot(tout, Stick_input(:,2))
```

Logging Data from the Generated Program

Since you have modified the model, you must rebuild and run the `f14rtw` executable in order to obtain a valid data file:

1 Select **Build Model** from the **Real-Time Workshop** submenu of the **Tools** menu in the Simulink window.

2 When the build completes, run the stand-alone program from MATLAB.

```
!f14rtw
```

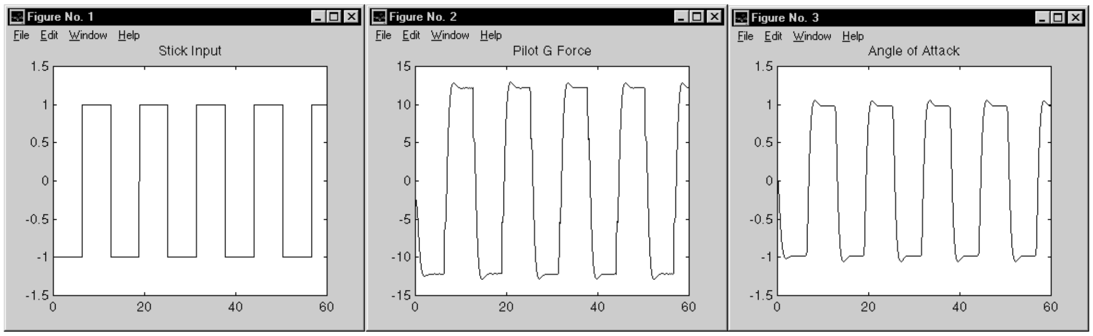
3 Load the data file `f14rtw.mat` and observe the workspace variables.

```
load f14rtw
whos
```

The data loaded from the MAT-file will include `rt_Pilot_G_force`, `rt_Angle_of_attack`, `rt_Stick_input`, and `rt_tout`.

4 You can now use MATLAB to plot the three workspace variables as a function of time.

```
plot(rt_tout,rt_Stick_input(:,2))
figure
plot(rt_tout,rt_Pilot_G_force(:,2))
figure
plot(rt_tout,rt_Angle_of_attack(:,2))
```



Comparing Results of the Simulation and the Generated Program

Your Simulink simulations and the generated code should produce nearly identical output.

You have now obtained data from a Simulink run of the model, and from a run of the program generated from the model. It is a simple matter to compare the f14rtw model output to the results achieved by the Real-Time Workshop.

Comparing Angle_of_attack (simulation output) to rt_Angle_of_attack (generated program output) produces

```
max(abs(rt_Angle_of_attack-Angle_of_attack))
ans =
  1.0e-015 *
         0    0.4441
```

Comparing Pilot_G_force (simulation output) to rt_Pilot_G_force (generated program output) produces

```
max(abs(rt_Pilot_G_force-Pilot_G_force))
ans =
  1.0e-013 *
         0    0.7283
```

So overall agreement is within 10^{-13}. This slight error can be caused by many factors, including:

- Different compiler optimizations
- Statement ordering
- Run-time libraries

For example, a function such as sin(2.0) may return a slightly different value, depending on which C library you are using.

For the same reasons, your comparison results may not be identical to those above.

Tutorial 4: A First Look at Generated Code

In this tutorial, you examine code generated from a from a simple model, to observe the effects of one of the many code optimization features available in the Real-Time Workshop.

Figure 1-12 shows the source model.

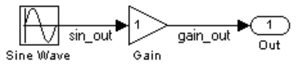


Figure 1-12: example.mdl

Setting up the Model

First, create the model and set up basic Simulink and Real-Time Workshop parameters as follows:

1 Create a directory `example_codegen`, and make it your working directory.

```
!mkdir example_codegen
cd example_codegen
```

2 Create a new model and save it as `example.mdl`.

3 Add Sine Wave, Gain, and Out1 blocks to your model and connect them as shown in Figure 1-12. Label the signals as shown.

4 From the **Simulation** menu, choose **Simulation Parameters**. The **Simulation Parameters** dialog box opens.

5 Click the Solver tab and enter the following parameter values on the Solver page:

   Solver options: set **Type** to `Fixed-step`. Select the `ode5 (Dormand-Prince)` solver algorithm.

   Leave the other Solver page parameters set to their default values.

6 Click **Apply**.

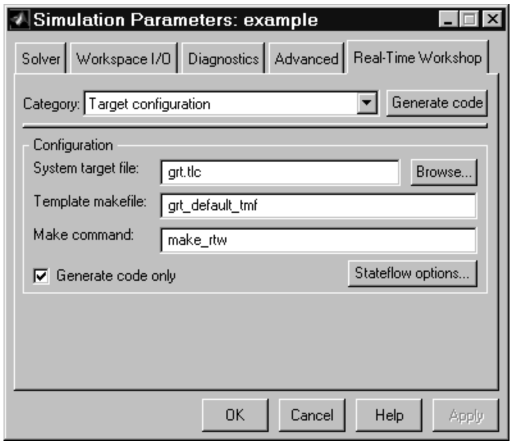
7 Click the Workspace I/O tab and make sure all check boxes are deselected.

8 Click **Apply**.

9 Click the Real-Time Workshop tab. Select **Target configuration** from the **Category** pull-down menu. Next, select the **Generate code only** option.

This option causes the Real-Time Workshop to generate code without invoking make to compile and link the code. This option is convenient for this exercise, as we are only interested in looking at the generated code. Note that the **Build** button caption changes to **Generate code**.

Also, make sure that the generic real-time (GRT) target is selected. The page should appear as below.



Category: Target configuration

System target file: grt.tlc

Template makefile: grt_default_tmf

Make command: make_rtw

☑ Generate code only

10 Click **Apply**.

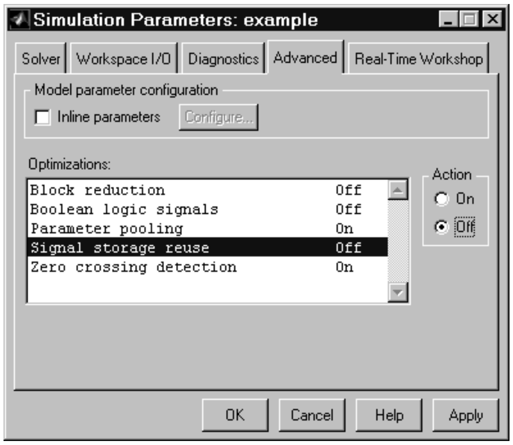
11 Save the model.

Generating Code Without Buffer Optimization

When the block I/O optimization feature is enabled, the Real-Time Workshop uses local storage for block outputs wherever possible. We now disable this option to see what the generated code looks like:

1 From the **Simulation** menu, choose **Simulation Parameters**. The **Simulation Parameters** dialog box opens.

2 Click the Advanced tab. Select the **Signal storage reuse** option and select the **Off** radio button, as shown below.



Simulation Parameters: example

Model parameter configuration

Inline parameters [ ]

Optimizations:

```
Block reduction          Off
Boolean logic signals    Off
Parameter pooling        On
Signal storage reuse     Off
Zero crossing detection  On
```

Action: On [✓ Off]

3 Click **Apply**.

4 Click the Real-Time Workshop tab and select the **Target configuration** category. Then click the **Generate code** button.

5 Because the **Generate code only** option was selected, the Real-Time Workshop does not invoke your make utility. The code generation process ends with the message

```
### Successful completion of Real-Time Workshop build procedure
for model: example
```

6 The generated code is in the build directory, `example_grt_rtw`. The file `example_grt_rtw\example.c` contains the output computation for the model. Open this file into the MATLAB editor.

```
edit example_grt_rtw\example.c
```

7 In `example.c`, find the function `MdlOutputs`.

The generated C code consists of procedures that implement the algorithms defined by your Simulink block diagram. Your target’s execution engine executes the procedures as time moves forward. The modules that implement the execution engine and other capabilities are referred to collectively as the *run-time interface modules*.

In our example, the generated `MdlOutputs` function implements the actual algorithm for multiplying a sine wave by a gain. The `MdlOutputs` function computes the model's block outputs. The run-time interface must call `MdlOutputs` at every time step.

With buffer optimizations turned off, `MdlOutputs` assigns buffers to each block input and output. These buffers (`rtB.sin_out`, `rtB.gain_out`) are members of a global data structure, rtB. The code is shown below.

```
void MdlOutputs(int_T tid)
{
  /* Sin Block: <Root>/Sine Wave */
  rtB.sin_out = rtP.Sine_Wave_Amp *
    sin(rtP.Sine_Wave_Freq * ssGetT(rtS) + rtP.Sine_Wave_Phase);
  /* Gain Block: <Root>/Gain */
  rtB.gain_out = rtB.sin_out * rtP.Gain_Gain;
  /* Outport Block: <Root>/Out1 */
  rtY.Out1 = rtB.gain_out;
}
```

We now turn buffer optimization on and observe how it improves the code.

Generating Code with Buffer Optimization

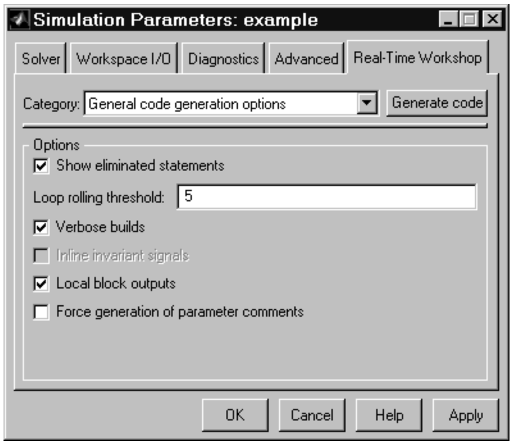
Enable buffer optimization and re-generate the code as follows:

1. From the **Simulation** menu, choose **Simulation Parameters**. The **Simulation Parameters** dialog box opens.

2. Click the Advanced tab. Select the **Signal storage reuse** option and select the **On** radio button.

3. Click **Apply**.

4. Click the Real-Time Workshop tab. Select **General code generation options** from the **Category** pull-down menu.



Category: General code generation options

Options

[✓] Show eliminated statements

Loop rolling threshold: 5

[✓] Verbose builds

[ ] Inline invariant signals

[✓] Local block outputs

[ ] Force generation of parameter comments

5 Make sure that the **Local block outputs** option is both enabled and selected, as shown above.

6 Click **Apply** and select the **Target Configuration** category again.

7 Click the **Generate code** button.

8 As before, the Real-Time Workshop generates code in the `example_grt_rtw` directory. Note that the previously-generated code is overwritten.

9 Edit `example_grt_rtw/example.c`, and examine the function `MdlOutputs`.

With buffer optimization enabled, the code in `MdlOutputs` reuses `rtb_temp0`, a temporary buffer with local scope, rather than assigning global buffers to each input and output.

```
void MdlOutputs(int_T tid)
{
  /* local block i/o variables */
  real_T rtb_temp0;
  /* Sin Block: <Root>/Sine Wave */
  rtb_temp0 = rtP.Sine_Wave_Amp *
    sin(rtP.Sine_Wave_Freq * ssGetT(rtS) + rtP.Sine_Wave_Phase);
  /* Gain Block: <Root>/Gain */
  rtb_temp0 *= rtP.Gain_Gain;
```

```
    /* Outport Block: <Root>/Out1 */
    rtY.Out1 = rtb_temp0;
  }
```

This code is more efficient in terms of memory usage. The efficiency improvement gained by enabling **Local block outputs** would be more significant in a large model with many signals.

Note that, by default, **Local block outputs** is enabled. Chapter 3, “Code Generation and the Build Process” contains details on this and other code generation options.

For further information on the structure and execution of *model*.c files, refer to Chapter 6, “Program Architecture.”

Where to Find Information in This Manual

The list below will guide you to information relevant to your development tasks and interests.

Single- and Multitasking Code Generation

The Real-Time Workshop fully supports single- and multitasking code generation. See Chapter 6, “Program Architecture” and Chapter 7, “Models with Multiple Sample Rates” for a complete description.

Customizing Generated Code

The Real-Time Workshop Custom Code library supports customization of the generated code. See Chapter 14, “Custom Code Blocks” for a description of this library.

An alternative approach to customizing generated code is to modify Target Language Compiler (TLC) files. The Target Language Compiler is an interpreted language that translates Simulink models into C code. Using the Target Language Compiler, you can direct the code generation process.

There are two TLC files, `hookslib.tlc` and `cachelib.tlc`, that contain functions you can use to customize Real-Time Workshop generated code. See the *Target Language Compiler Reference Guide f*or more information about these TLC files. See also the source code, located in *`matlabroot`*`/rtw/c/tlc`.

Optimizing Generated Code

The default code generation settings are generic for flexible rapid prototyping systems. The penalty for this flexibility is code that is less than optimal. There are several optimization techniques that you can use to minimize the source code size and memory usage once you have a model that meets your requirements.

See Chapter 3, “Code Generation and the Build Process” and Chapter 8, “Optimizing the Model for Code Generation” for details on code optimization techniques available for all target configurations.

Chapter 9, “Real-Time Workshop Embedded Coder” contains information about optimization specifically for embedded code.

Validating Generated Code

Using the Real-Time Workshop data logging features, you can create an executable that runs on your workstation and creates a data file. You can then compare the results of your program with the results of running an equivalent Simulink simulation.

For more information on how to validate Real-Time Workshop generated code, see “Workspace I/O Options and Data Logging” on page 3-18. See also “Tutorial 2: Data Logging” and “Tutorial 3: Code Validation” in this chapter.

Incorporating Generated Code into Larger Systems

If your Real-Time Workshop generated code is intended to function within an existing code base (for example, if you want to use the generated code as a plug-in function), you should use the Real-Time Workshop Embedded Coder. Chapter 9, “Real-Time Workshop Embedded Coder” documents the entry points and header files you will need to interface your code to Real-Time Workshop Embedded Coder generated code.

Incorporating Your Code into Generated Code

To interface your hand-written code with Real-Time Workshop generated code, you can use an S-function wrapper. See the *Writing S-Functions* manual for more information.

Creating and Communicating with Device Drivers

S-functions provide a flexible method for communicating with device drivers. See Chapter 17, “Targeting Real-Time Systems” for a description of how to build device drivers. Also, for a complete discussion of S-functions, see the *Writing S-Functions* manual.

Code Tracing

The Real-Time Workshop includes special tags throughout the generated code that make it easy to trace generated code back to your Simulink model. See “Tracing Generated Code Back to Your Simulink Model” on page 3-28 for more information about this feature.

Automatic Build Procedure

Using the Real-Time Workshop, you can generate code with the push of a button. The automatic build procedure, initiated by a single mouse click, generates code, a makefile, and optionally compiles (or cross-compiles) and downloads a program. See “Automatic Program Building” on page 2-12 for an overview, and Chapter 3, “Code Generation and the Build Process” for complete details.

Parameter Tuning

Parameter tuning enables you to change block parameters while a generated program runs, thus avoiding recompiling the generated code. The Real-Time Workshop supports parameter tuning in four different environments:

- External mode: You can tune parameters from Simulink while running the generated code on a target processor. See Chapter 5, “External Mode” for information on this mode.
- External C application program interface (API): You can write your own C API interface for parameter tuning using support files provided by The MathWorks. See Chapter 17, “Targeting Real-Time Systems” for more information.
- Rapid simulation: You can use the Rapid Simulation Target (`rsim`) in batch mode to provide fast simulations for performing parametric studies. Although this is not an on-the-fly application of parameter tuning, it is nevertheless a useful way to evaluate a model. This mode is also useful for Monte Carlo simulation. See Chapter 11, “Real-Time Workshop Rapid Simulation Target” for further information.
- Simulink: Prior to generating real-time code, you can tune parameters on-the-fly in your Simulink model.

See also “Interfacing Signals and Parameters” on page 1-66.

Monitoring Signals and Logging Data

There are several ways to monitor signals and data in the Real-Time Workshop:

- External mode: You can monitor and log signals from an externally executing program via Scope blocks and several other types of external mode compatible blocks. See “External Signal & Triggering Dialog Box” on page 5-24 for a discussion of this method.

- External C application program interface (API): You can write your own C API for signal monitoring using support files provided by The MathWorks. See Chapter 17, “Targeting Real-Time Systems” for more information.
- MAT-file logging: You can use a MAT-file to log data from the generated executable. See “Workspace I/O Options and Data Logging” on page 3-18 for more information.
- Simulink: You can use any of Simulink’s data logging capabilities.

Interfacing Signals and Parameters

You can interface signals and parameters in your model to hand-written code by specifying the storage declarations of signals and parameters. For more information, see:

- “Parameters: Storage, Interfacing, and Tuning” on page 3-51
- “Signals: Storage, Optimization, and Interfacing” on page 3-65
- “Interfacing Parameters and Signals” on page 17-65

Sample Implementations

The Real-Time Workshop provides sample implementations that illustrate the development of real-time programs under DOS and Tornado, as well as generic real-time programs under Windows and UNIX.

These sample implementations are located in the following directories:

- *matlabroot*`/rtw/c/grt`: Generic real-time examples
- *matlabroot*`/rtw/c/dos`: DOS examples
- *matlabroot*`/rtw/c/tornado`: Tornado examples

Technical Overview

The Rapid Prototyping Process

The Real Time Workshop supports *rapid prototyping*, a process that allows you to:

- Conceptualize solutions graphically in a block diagram modeling environment
- Evaluate system performance early on — before laying out hardware, coding production software, or committing to a fixed design
- Refine your design by rapid iteration between algorithm design and prototyping
- Tune parameters while your real-time model runs, using Simulink in *external mode* as a graphical front-end

Key Aspects of Rapid Prototyping

Figure 2-1 contrasts the rapid prototyping development process with the traditional development process.

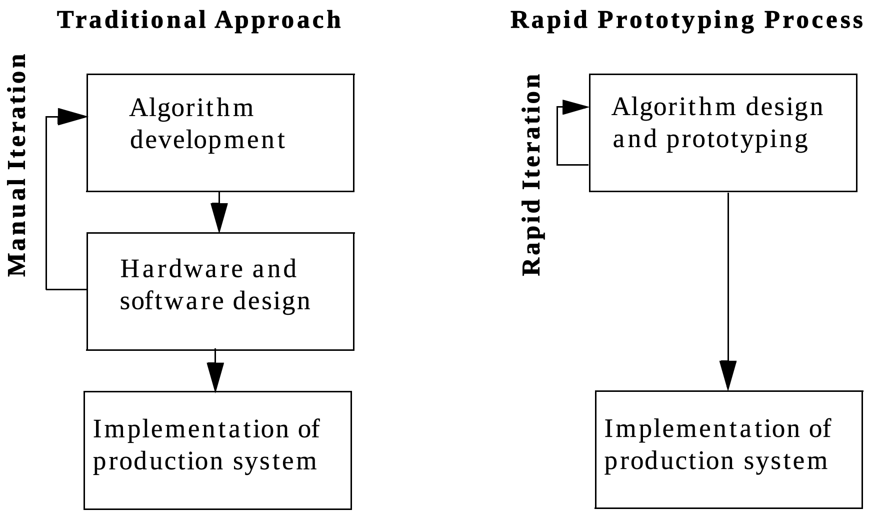

Traditional Approach
Rapid Prototyping Process
Manual Iteration
Algorithm development
Hardware and software design
Implementation of production system
Rapid Iteration
Algorithm design and prototyping
Implementation of production system

Figure 2-1: Traditional vs. Rapid Prototyping Development Processes

The traditional approach to real-time design and implementation typically involves multiple teams of engineers, including an algorithm design team, software design team, hardware design team, and an implementation team. When the algorithm design team has completed its specifications, the software design team implements the algorithm in a simulation environment and then specifies the hardware requirements. The hardware design team then creates the production hardware. Finally, the implementation team integrates the hardware into the larger overall system.

This traditional development process can be lengthy, because the algorithm design engineers do not work with the actual hardware. The rapid prototyping process combines the algorithm, software, and hardware design phases, eliminating potential bottlenecks. The process allows engineers to see the results and rapidly iterate on the design before expensive hardware is developed.

The key to rapid prototyping is *automatic program building*. Automatic program building puts algorithm development (including coding, compiling, linking, and downloading to target hardware) under control of a single process. Automatic program building allows you to make design changes directly to the block diagram.

You begin the rapid prototyping process with the development of a model in Simulink. In control engineering, you model plant dynamics and other dynamic components that constitute a controller and/or an observer. In digital signal processing, your model typically explores input signal characteristics, such as the signal-to-noise ratio.

You then simulate your model in Simulink. You use MATLAB, Simulink, and toolboxes to aid in the development of algorithms and analysis of the results. If the results are not satisfactory, you can iterate the modeling/analysis process until results are acceptable.

Once you have achieved the desired results, you use the Real-Time Workshop to generate downloadable C code that implements the appropriate portions of the model. Using Simulink in external mode, you can tune parameters and further refine your model, again rapidly iterating to achieve required results. At this stage, the rapid prototyping process is complete. You can begin the final implementation for production with confidence that the underlying algorithms work properly in your real-time production system.

The figure below shows the rapid prototyping process in more detail.

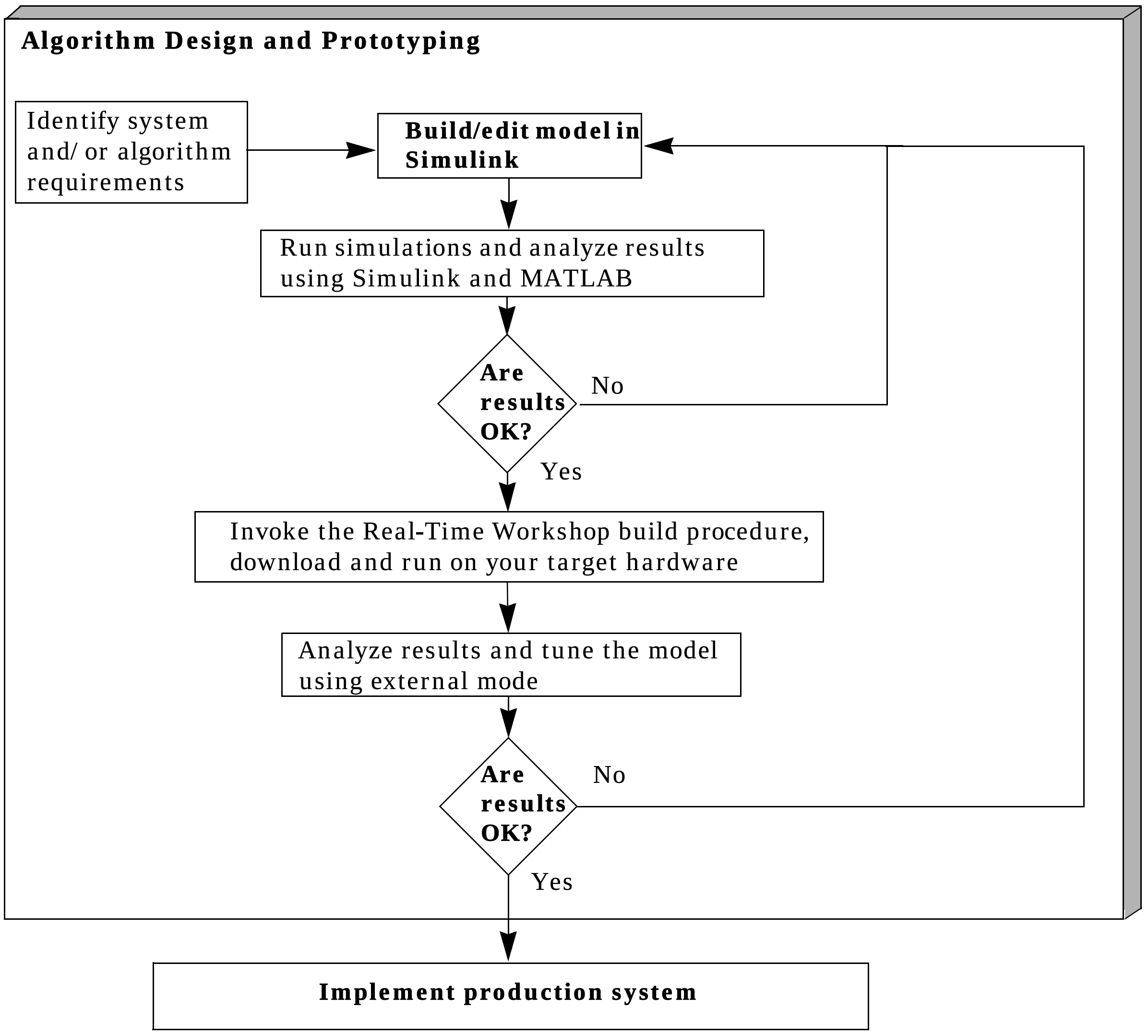

Algorithm Design and Prototyping
Identify system and/ or algorithm requirements
Build/edit model in Simulink
Run simulations and analyze results using Simulink and MATLAB
Are results OK?
No
Yes
Invoke the Real-Time Workshop build procedure, download and run on your target hardware
Analyze results and tune the model using external mode
Are results OK?
No
Yes
Implement production system

Figure 2-2: The Rapid Prototyping Development Process

This highly productive development cycle is possible because the Real-Time Workshop is closely tied to MATLAB and Simulink. Each package contributes to the design of your application:

- MATLAB: Provides design, analysis, and data visualization tools.
- Simulink: Provides system modeling, simulation, and validation.
- Real-Time Workshop: Generates C or Ada code from Simulink model; provides framework for running generated code in real-time, tuning parameters, and viewing real-time data.

Rapid Prototyping for Digital Signal Processing

The first step in the rapid prototyping process for digital signal processing is to consider the kind and quality of the data to be worked on, and to relate it to the system requirements. Typically this includes examining the signal-to-noise ratio, distortion, and other characteristics of the incoming signal, and relating them to algorithm and design choices.

System Simulation and Algorithm Design

In the rapid prototyping process, the block diagram plays two roles in algorithm development. The block diagram helps to identify processing bottlenecks, and to optimize the algorithm or system architecture. The block diagram also functions as a high-level system description.That is, the diagram provides a hierarchical framework for evaluating the behavior and accuracy of alternative algorithms under a range of operating conditions.

Analyzing Results, Parameter Tuning, and Signal Monitoring Using External Mode

After creating an algorithm (or a set of candidate algorithms), the next stage is to consider architectural and implementation issues. These include complexity, speed, and accuracy. In a conventional development environment, this would mean running the algorithm and recoding it in C or in a hardware design and simulation package.

Simulink's external mode allows you to change parameters interactively, while your signal processing algorithms execute in real time on the target hardware. After building the executable and downloading it to your hardware, you tune (modify) block parameters in Simulink. Simulink automatically downloads the new values to the hardware. You can monitor the effects of your parameter changes by simply connecting Scope blocks to signals that you want to observe.

Rapid Prototyping for Control Systems

Rapid prototyping for control systems is similar to digital signal processing, with one major difference. In control systems design, it is necessary to develop a model of your plant prior to algorithm development in order to simulate closed loop performance. Once your plant model is sufficiently accurate, the rapid prototyping process for control system design continues in much the same manner as digital signal processing design.

Rapid prototyping begins with developing block diagram plant models of sufficient fidelity for preliminary system design and simulation. Once simulations show encouraging system performance, the controller block diagram is separated from the plant model and I/O device drivers are attached. Automatic code generation immediately converts the entire system to real-time executable code. The executable can be automatically loaded onto target hardware, allowing the implementation of real-time control systems in a very short time.

Modeling Systems in Simulink

The first step in the design process is development of a plant model. The Simulink collection of linear and nonlinear components helps you to build models involving plant, sensor, and actuator dynamics. Because Simulink is customizable, you can further simplify modeling by creating custom blocks and block libraries from continuous- and discrete-time components.

Using the System Identification Toolbox, you can analyze test data to develop an empirical plant model; or you can use the Symbolic Math Toolbox to translate the equations of the plant dynamics into state-variable form.

Analysis of Simulation Results

You can use MATLAB and Simulink to analyze the results produced from a model developed in the first step of the rapid prototyping process. At this stage, you can design and add a controller to your plant.

Algorithm Design and Analysis

From the block diagrams developed during the modeling stage, you can extract state-space models through linearization techniques. These matrices can be used in control system design. You can use the following toolboxes to facilitate control system design, and work with the matrices that you derived:

- Control System Toolbox
- LMI Control Toolbox
- Model Predictive Control Toolbox
- Robust Control Toolbox

Once you have your controller designed, you can create a closed-loop system by connecting it to the Simulink plant model. Closed-loop simulations allow you to determine how well the initial design meets performance requirements.

Once you have a satisfactory model, it is a simple matter to generate C code directly from the Simulink block diagram, compile it for the target processor, and link it with supplied or user-written application modules.

Analyzing Results, Parameter Tuning, and Signal Monitoring Using External Mode

You can load output data from your program into MATLAB for analysis, or display the data with third party monitoring tools. You can easily make design changes to the Simulink model and then regenerate the C code.

Simulink's external mode allows you to change parameters interactively, while your algorithms execute in real time on the target hardware. After building the executable and downloading it to your hardware, you tune (modify) block parameters in Simulink. Simulink automatically downloads the new values to the hardware. You can monitor the effects of your parameter changes by simply connecting Scope blocks to signals that you want to observe.

Open Architecture of the Real-Time Workshop

The Real-Time Workshop is an open system designed for use with a wide variety of operating environments and hardware types. There are many ways to modify and extend the key elements of Real-Time Workshop. Figure 2-3 shows these elements.

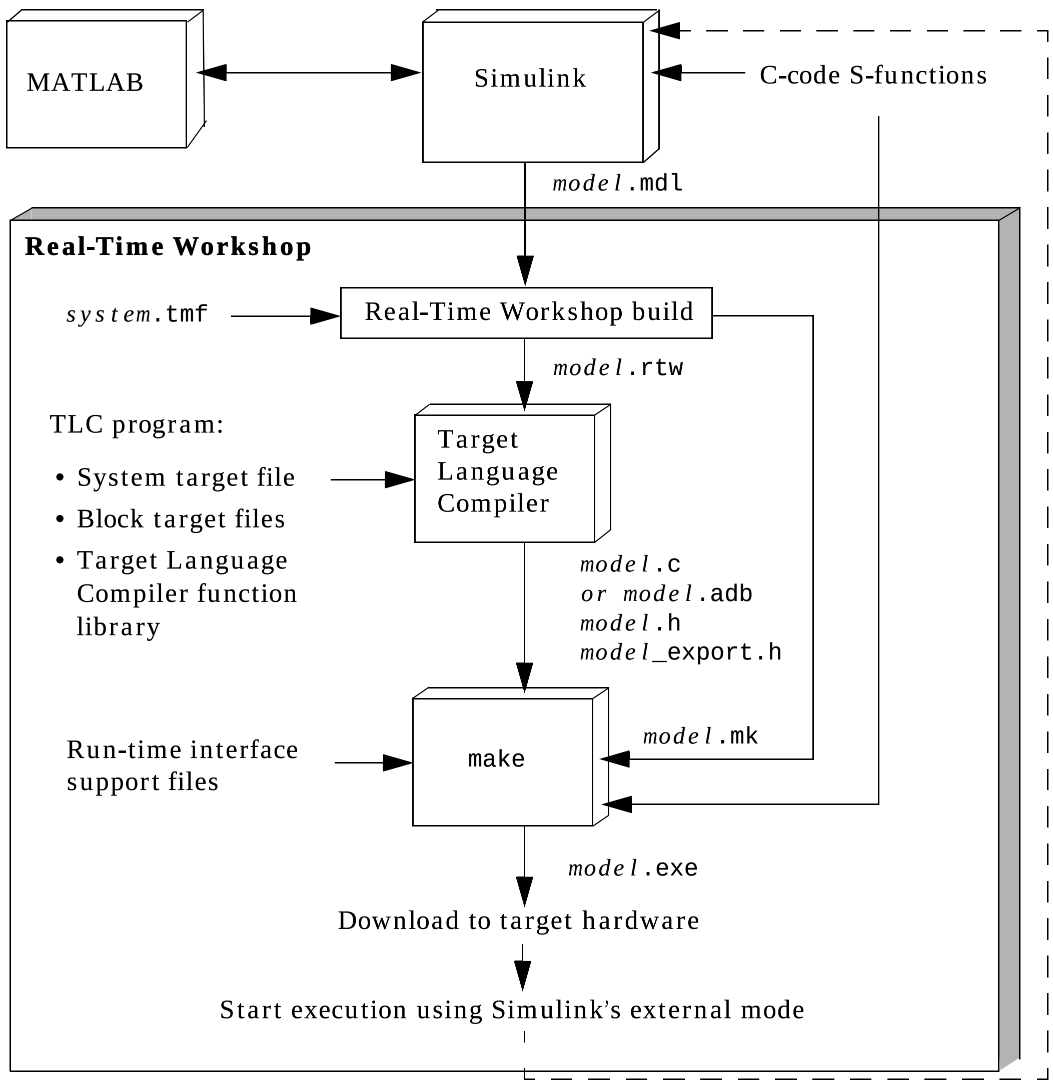


Figure 2-3: The Real-Time Workshop Architecture

You can configure the Real-Time Workshop program generation process to your own needs by modifying the following components:

- Simulink and the model file (*model*`.mdl`)

  Simulink provides a very high-level language (VHLL) development environment. The language elements are blocks and subsystems that visually embody your algorithms. You can think of the Real-Time Workshop as a compiler that processes a VHLL source program (*model*`.mdl`), and emits code suitable for a traditional high-level language (HLL) compiler.

  S-functions written in C or Ada let you extend Simulink's VHLL by adding new general purpose blocks, or incorporating legacy code into a block.

- The intermediate model description (*model*`.rtw`)

  The initial stage of the code generation process is to analyze the source model. The resultant description file contains a hierarchical structure of records describing systems and blocks and their connections.

  The S-function API includes a special function, `mdlRTW`, that lets you customize the code generation process by inserting parameter data from your own blocks into the *model*`.rtw` file.

- The Target Language Compiler (TLC) program

  The Target Language Compiler interprets a program that reads the intermediate model description and generates code that implements the model as a program.

  You can customize the elements of the TLC program in two ways. First, you can implement your own system target file, which controls overall code generation parameters. Second, you can implement block target files, which control how code is generated from individual blocks such as your own S-function blocks.

- Source code generated from the model.

  There are several ways to customize generated code, or interface it to custom code:

  - Exported entry points let you interface your hand-written code to the generated code. This makes it possible to develop your own timing and execution engine, or to combine code generated from several models into a single executable.

- You can automatically make signals, parameters, and other data structures within generated code visible to your own code, facilitating parameter tuning and signal monitoring.
- *Custom code blocks* allow you to insert your own code directly into the generated code, either at the model or subsystem level.

• Run-time interface support files

The *run-time interface* consists of code interfacing to the generated model code. You can create a custom set of run-time interface files, including:

- A harness (main) program
- Code to implement a custom external mode communication protocol
- Code that interfaces to parameters and signals defined in the generated code
- Timer and other interrupt service routines
- Hardware I/O drivers

• The template makefile and *model*`.mk`

A makefile, *model*`.mk`, controls the compilation and linking of generated code. The Real-Time Workshop generates *model*`.mk` from a template makefile during the code generation/build process. You can create a custom template makefile to control compiler options and other variables of the `make` process.

All of these components contribute to the process of transforming a Simulink model into an executable program. The next section is an overview of this process.

Automatic Program Building

The Real-Time Workshop automatic program building process creates programs for real-time applications in a variety of host environments. Automatic program building uses the `make` utility to control the compilation and linking of generated source code.

A high-level M-file command controls the Real-Time Workshop build process. The default command, used with most targets, is `make_rtw`.

The build process consists of the following steps:

1. Analysis of the model and compilation of a model description file
2. Generation of code from the model by the Target Language Compiler
3. Generation of a makefile, customized for a given build
4. Creation of an executable program by the make utility under the control of the customized makefile

The shaded box in Figure 2-4 outlines these steps.

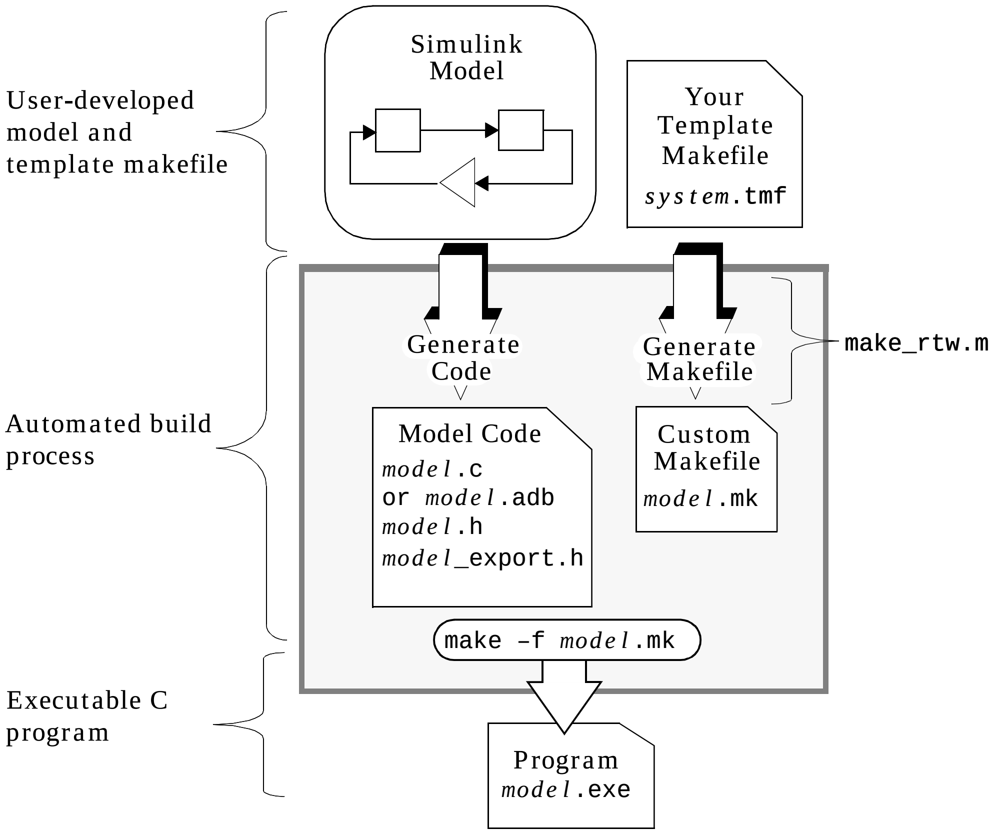


Figure 2-4: Automatic Program Building

Steps in the Build Process

Analysis of the Model

The build process begins with the analysis of your Simulink block diagram. The analysis process consists of these tasks:

- Evaluating simulation and block parameters
- Propagating signal widths and sample times

- Determining the execution order of blocks within the model
- Computing work vector sizes such as those used by S-functions (for more information about work vectors, refer to the *Writing S-Functions* manual.)

During this phase, the Real-Time Workshop reads your model file (*model*`.mdl`) and compiles an intermediate representation of the model. This intermediate description is stored, in a language-independent format, in an ASCII file named *model*`.rtw`. The *model*.rtw file is the input to the next stage of the build process.

model`.rtw` files are similar in format to Simulink model (`.mdl`) files. "Overview of a model.rtw File" on page 2–17 explains the basic features of a `.rtw` file. For a detailed description of the contents of *model*`.rtw` files, see the *Target Language Compiler Reference Guide.*

Generation of Code by the Target Language Compiler

In the second stage of the build procedure, the Target Language Compiler transforms the intermediate model description stored in *model*`.rtw` into target-specific code.

The Target Language Compiler is an interpreted programming language designed for the sole purpose of converting a model description into code. The Target Language Compiler executes a *TLC program* comprising several *target files* (`.tlc` files). The TLC program specifies how to generate code from the model, using the *model*`.rtw` file as input.

The TLC program consists of:

- The *system target file*

  The system target file is the entry point or main file.
- *Block target files*

  For each block in a Simulink model, there is a block target file that specifies how to translate that block into target-specific code.
- The *Target Language Compiler function library*

  The Target Language Compiler function library contains functions that support the code generation process.

The Target Language Compiler begins by reading in the *model*`.rtw` file. It then compiles and executes the commands in the target files — first the system target file, then the individual block target files. The output of the Target Language Compiler is a source code version of the Simulink block diagram.

Generation of the Customized Makefile

The third step in the build procedure is to generate a customized makefile, *model*`.mk`. The generated makefile instructs the `make` utility to compile and link source code generated from the model, as well as any required harness program, libraries, or user-provided modules.

The Real-Time Workshop creates *model*`.mk` from a *system template makefile*, *system*`.tmf`. The system template makefile is designed for your target environment. The template makefile allows you to specify compilers, compiler options, and additional information used during the creation of the executable.

The *model*`.mk` file is created by copying the contents of *system*`.tmf` and expanding tokens that describe your model's configuration.

The Real-Time Workshop provides many system template makefiles, configured for specific target environments and development systems. "The System Target File Browser" on page 3-34 lists all template makefiles that are bundled with the Real-Time Workshop.

You can fully customize your build process by modifying an existing template makefile or providing your own template makefile.

Creation of the Executable

Creation of an executable program is the final stage of the build process. This stage is optional, as illustrated in Figure 2-5.

If you are targeting a system such as an embedded microcontroller or a DSP board, you can choose to generate only source code. You can then cross compile your code and download it to your target hardware. "Making an Executable" in Chapter 3 discusses the options that control whether or not the build creates an executable.

The creation of the executable, if enabled, takes place after the *model*`.mk` file has been created. At this point, the build process invokes the `make` utility, which in turn runs the compiler. To avoid unnecessary recompilation of C files, the `make` utility performs date checking on the dependencies between the object and C files; only out-of-date source files are compiled.

Optionally, `make` can also download the executable to your target hardware.

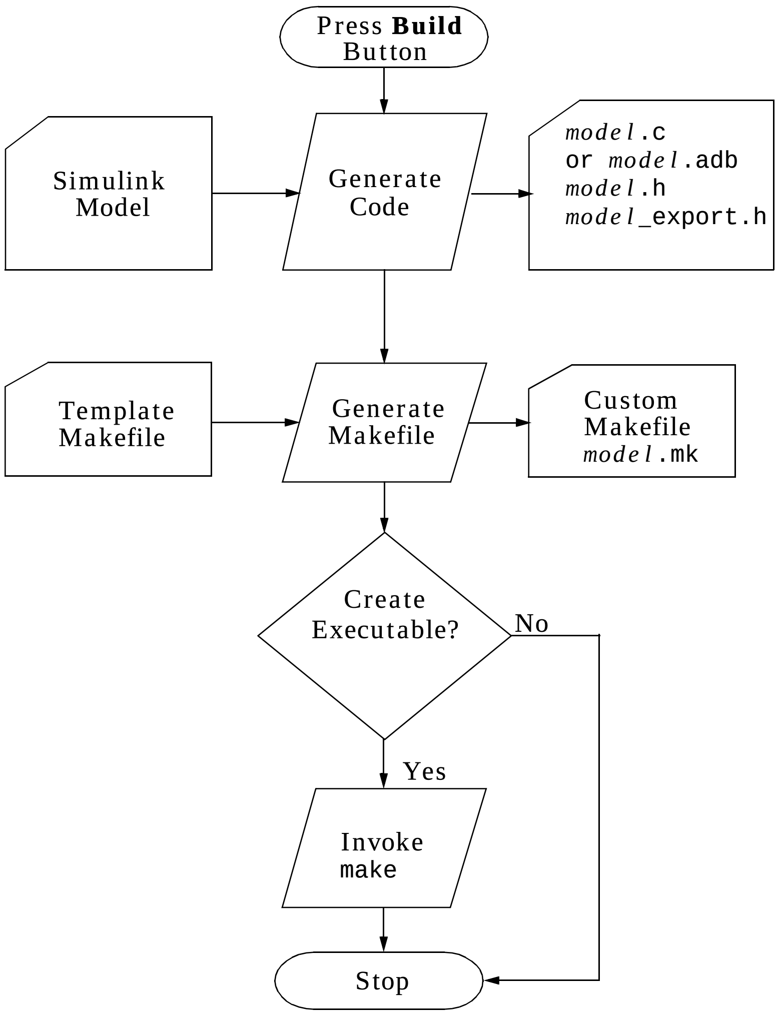


Figure 2-5: Automatic Program Building: Control Flow

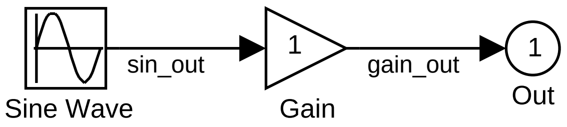
Summary of Files Created by the Build Procedure

The following is a list of the main of the *model*.* files created during the code generation and build process. Each performs a specific function in the Real-Time Workshop. Depending on code generation options, other files may also be created by the build process.

- *model*`.mdl`, created by Simulink, is analogous to a high-level programming language source file.
- *model*`.rtw`, generated by the Real-Time Workshop build process, is analogous to the object file created from a high-level language source program.
- *model*`.c`, generated by the Target Language Compiler, is the C source code corresponding to the *model*`.mdl` file.
- *model*`.h`, generated by the Target Language Compiler, is a header file that maps the links between blocks in the model.
- *model*`_export.h`, generated by the Target Language Compiler, is a header file that contains exported signal, parameter, and function symbols.
- *model*`.mk`, generated by the Real-Time Workshop build process, is the customized makefile used to build an executable.
- *model*`.exe` (on PC) or *model* (on UNIX), is an executable program, created under control of the `make` utility by your development system.

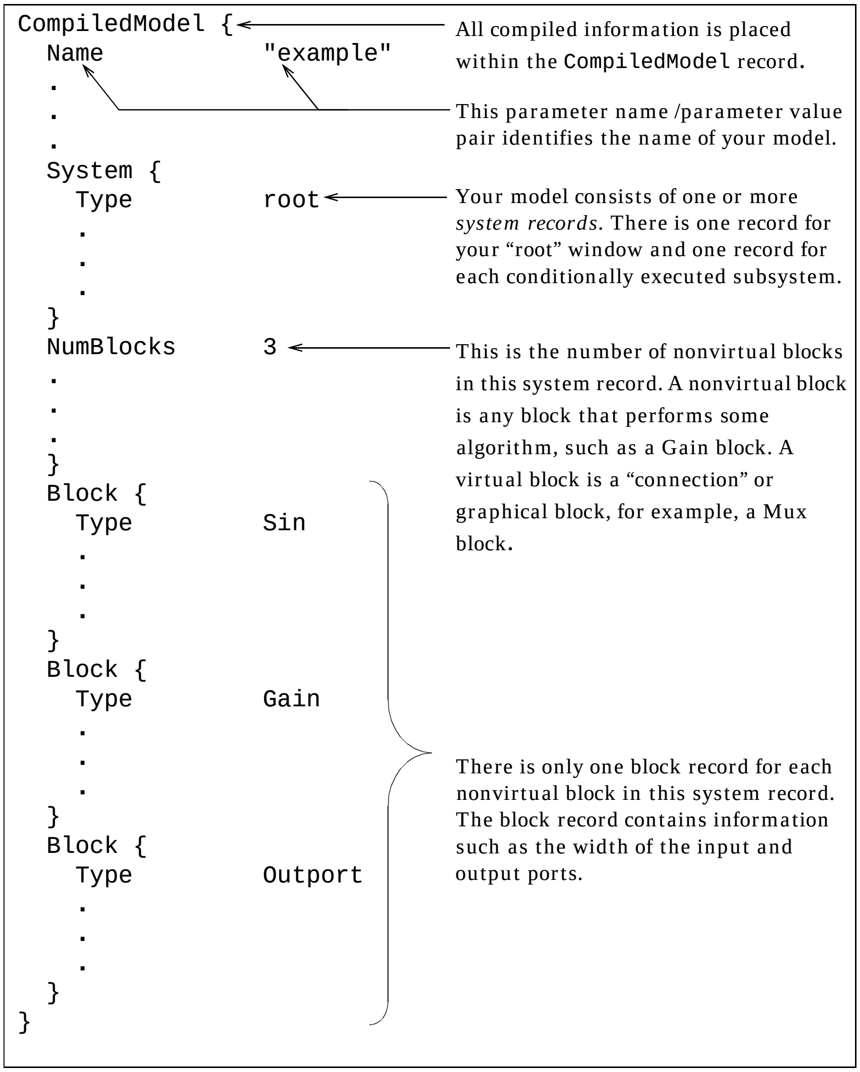
Overview of a model.rtw File

This section examines the basic features of a *model*`.rtw` file. The `.rtw` file shown is generated from the source model shown below.



This model is saved in a file called `example.mdl`. The Real-Time Workshop generates `example.rtw`., an ASCII file. The `example.rtw` file consists of parameter name/parameter value pairs, stored in a hierarchical structure consisting of records.

Below is an excerpt from `example.rtw`.



```
CompiledModel {
  Name              "example"
  .
  .
  .
  System {
    Type            root
    .
    .
    .
  }
  NumBlocks         3
  .
  .
  .
  }
  Block {
    Type            Sin
    .
    .
    .
  }
  Block {
    Type            Gain
    .
    .
    .
  }
  Block {
    Type            Outport
    .
    .
    .
  }
}
```

All compiled information is placed within the `CompiledModel` record.

This parameter name /parameter value pair identifies the name of your model.

Your model consists of one or more *system records*. There is one record for your “root” window and one record for each conditionally executed subsystem.

This is the number of nonvirtual blocks in this system record. A nonvirtual block is any block that performs some algorithm, such as a Gain block. A virtual block is a “connection” or graphical block, for example, a Mux block.

There is only one block record for each nonvirtual block in this system record. The block record contains information such as the width of the input and output ports.

For more information on `.rtw` files, see the *Target Language Compiler Reference Guide*, which contains detailed descriptions of the contents of *model*`.rtw` files.

Code Generation and the Build Process

Introduction

Chapter 2, “Technical Overview” introduced code generation and the build process. This chapter covers these topics in detail.

The Real-Time Workshop page of the **Simulation Parameters** dialog enables you to control most aspects of the code generation and build process. The first section of this chapter,“Overview of the Real-Time Workshop User Interface” on page 3-4, introduces the features controlled by the Real-Time Workshop page.

The sections that follow concern the code generation phase of the build process:

- “Simulation Parameters and Code Generation” on page 3-17 discusses how options on the Simulink Solver, Workspace I/O, Diagnostics, and Advanced pages interact with code generation. These options include choice of single- or multitasking execution, and several methods of logging data to MAT-files.

  The section also illustrates how to use tags in the generated code to trace back to the blocks that generated the code.

  The section ends with a discussion of “Other Interactions Between Simulink and the Real-Time Workshop” such as sample time propagation and order of execution of blocks.
- “Selecting a Target Configuration” on page 3-34 illustrates the use of the System Target File Browser, and summarizes the target configurations that you can access through the browser.
- “Nonvirtual Subsystem Code Generation” on page 3-41 describes generation of separate code modules from nonvirtual subsystems.
- “Generating Code and Executables from Subsystems” on page 3-49 describes how to generate and build a stand-alone executable from a subsystem.
- “Parameters: Storage, Interfacing, and Tuning” on page 3-51 documents how to generate storage declarations in order to export and import model parameters to and from user-written code.
- “Signals: Storage, Optimization, and Interfacing” on page 3-65 documents how signal storage optimizations work, and how to generate storage declarations in order to export and import model signals to and from user-written code.

- "Simulink Data Objects and Code Generation" on page 3-79 documents how to represent and store signals and parameters in Simulink data objects, and how code is generated from these object.

The remaining sections cover the make (post-code generation) part of the build process:

- "Making an Executable" on page 3-97 documents options that control whether or not an executable is created during the build process.
- "Directories Used in the Build Process" on page 3-98 documents the output directories used and/or created during the build process.
- "Choosing and Configuring Your Compiler" on page 3-99 discusses the installation of a compiler and choice of a template makefile appropriate for use with your compiler.
- "Template Makefiles and Make Options" on page 3-102 includes a summary of available template makefiles and `make` command options.

Overview of the Real-Time Workshop User Interface

Many parameters and options affect the way that Real-Time Workshop generates code from your model and builds an executable. To set these parameters and options, you interact with the pages of the **Simulation Parameters** dialog box.

The Simulink Solver, Workspace I/O, Diagnostics, and Advanced pages affect both the behavior of the model in simulation, and the code generated from the model. “Simulation Parameters and Code Generation” on page 3-17 discusses how Simulink settings affect the code generation process.

The Real-Time Workshop page lets you set parameters that directly affect code generation and optimization. You also initiate and control the build process from the Real-Time Workshop page.

Using the Real-Time Workshop Page

There are two ways to open the Real-Time Workshop page:

- From the **Simulation** menu, choose **Simulation Parameters**. When the **Simulation Parameters** dialog box opens, click on the Real-Time Workshop tab.
- Alternatively, select **Options** from the **Real-Time Workshop** submenu of the **Tools** menu in the Simulink window.

The Real-Time Workshop page is divided into two sections. The upper section contains the **Category** pull-down menu and the **Build** button.

Category Menu

The **Category** menu lets you select and work with various groups of options and controls. The selected group of options is displayed in the lower section of the page. Figure 3-1 shows the **Category** menu in the Real-Time Workshop page.

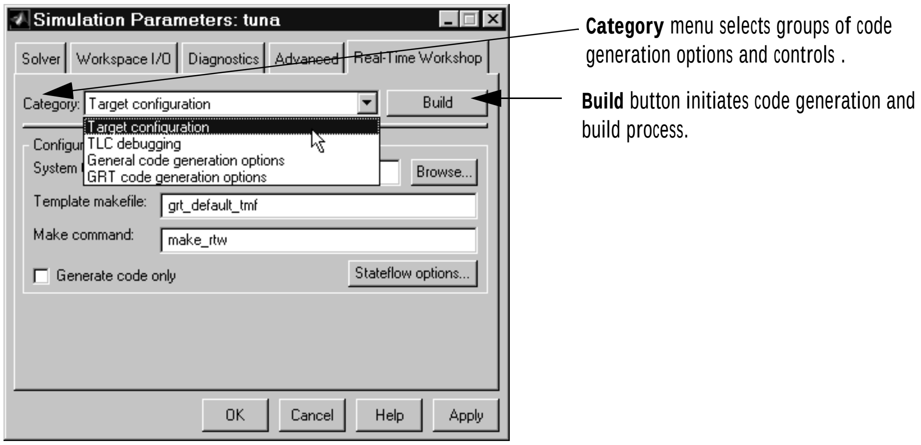


Figure 3-1: Category Menu and Build Button in Real-Time Workshop Page

The categories of options available from the **Category** menu are:

- **Target configuration**: high-level options related to control of the code generation and build process and selection of control files.
- **TLC debugging**: Target Language Compiler debugging and execution profiling options.
- **General code generation options**: Code generation settings that are common to all target configurations.
- **Target-specific code generation options**: One or more groups of options that are specific to the selected target configuration. In Figure 3-1, for example, the GRT target is selected.

Build Button

Click on the **Build** button to initiate the code generation and build process.

The following methods of initiating a build are exactly equivalent to clicking the **Build** button:

- Select **Build Model** from the **Real-Time Workshop** submenu of the **Tools** menu in the Simulink window (or use the key sequence **Ctrl+B)**.

- Invoke the `rtwbuild` command from the MATLAB command line. The syntax of the `rtwbuild` command is

  ```
  rtwbuild modelname
  ```

  or

  ```
  rtwbuild('modelname')
  ```

where *modelname* is the name of the source model. If the source model is not loaded into Simulink, `rtwbuild` loads the model.

Using ToolTips

The Real-Time Workshop page supports “ToolTip” online help. Place your mouse over any edit field name or check box to display a message box that explains the option.

The following sections summarize each category of options or parameters controlled by the Real-Time Workshop page, with references to subsequent sections that give details on each option or parameter.

Target Configuration Options

Figure 3-2 shows the **Target configuration** options of the Real-Time Workshop page.

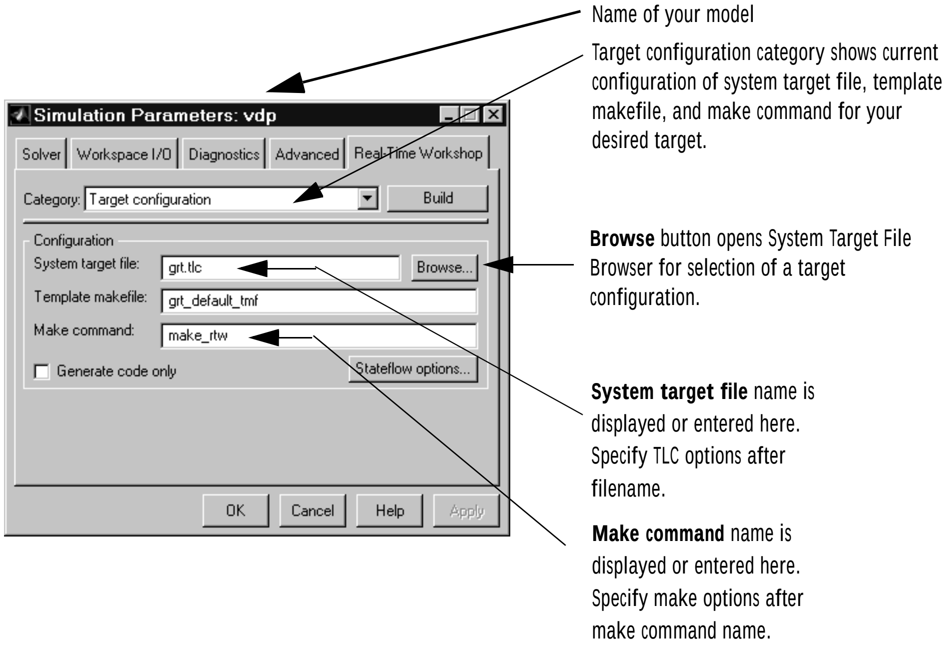

Name of your model
Target configuration category shows current configuration of system target file, template makefile, and make command for your desired target.
Browse button opens System Target File Browser for selection of a target configuration.
System target file name is displayed or entered here. Specify TLC options after filename.
Make command name is displayed or entered here. Specify make options after make command name.
Simulation Parameters: vdp
Solver
Workspace I/O
Diagnostics
Advanced
Real-Time Workshop
Category: Target configuration
Build
Configuration
System target file: grt.tlc
Browse...
Template makefile: grt_default_tmf
Make command: make_rtw
Generate code only
Stateflow options...
OK
Cancel
Help
Apply

Figure 3-2: The Real-Time Workshop Page: Target Configuration Options

Browse Button

The **Browse** button opens the System Target File Browser (See Figure 3-6 on page 3-35). The browser lets you select a preset target configuration consisting of a system target file, template makefile, and make command.

"Selecting a Target Configuration" on page 3-34 details the use of the browser and includes a complete list of available target configurations.

System Target File Field

The **System target file** field has these functions:

- If you have selected a target configuration using the System Target File Browser, this field displays the name of the chosen system target file (*target*`.tlc`).
- If you are using a target configuration that does not appear in the System Target File Browser, you must enter the name of the desired system target file in this field.
- After the system target filename, you can enter code generation options and variables for the Target Language Compiler. See "Target Language Compiler Variables and Options" on page 3-93 for details.

Template Makefile Field

The **Template makefile** field has these functions:

- If you have selected a target configuration using the System Target File Browser, this field displays the name of the chosen template makefile.
- You can enter the name of the desired template makefile in this field. You must do this if you are using a target configuration that does not appear in the System Target File Browser. This is necessary if you have written your own template makefile for a custom target environment.

Make Command Field

A high-level M-file command, invoked when a build is initiated, controls the Real-Time Workshop build process. Each target has an associated `make` command. The **Make command** field displays this command.

Almost all targets use the default command, `make_rtw`. "Targets Available from the System Target File Browser" on page 3-36 lists the `make` command associated with each target.

Third-party targets may supply another `make` command. See the vendor's documentation.

In addition to the name of the `make` command, you can supply arguments in the **Make command** field. These arguments include compiler-specific options, include paths, and other parameters. When the build process invokes the make utility, these arguments are passed along in the `make` command line.

"Template Makefiles and Make Options" on page 3-102 documents the **Make command** arguments you can use with each supported compiler.

Generate Code Only Option

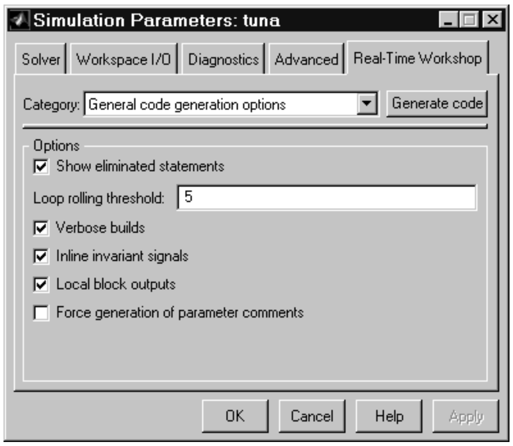
When this option is selected, the build process generates code but does not invoke the make command. The code is not compiled and an executable is not built.

When this option is selected, the caption of the **Build** button changes to **Generate code**.

Stateflow Options Button

If the model contains any Stateflow blocks, this button will launch the Stateflow Options dialog. Refer to the *Stateflow User's Guide* for information.

General Code Generation Options


Simulation Parameters: tuna

Solver | Workspace I/O | Diagnostics | Advanced | Real-Time Workshop

Category: General code generation options   Generate code

Options
- [✓] Show eliminated statements
- Loop rolling threshold: 5
- [✓] Verbose builds
- [✓] Inline invariant signals
- [✓] Local block outputs
- [ ] Force generation of parameter comments

OK   Cancel   Help   Apply

These options are common to all target configurations.

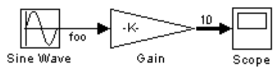
Show Eliminated Statements Option

If this option is selected, statements that were eliminated as the result of optimizations (such as parameter inlining) appear as comments in the generated code.

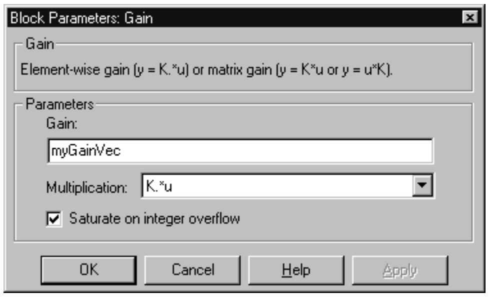
Loop Rolling Threshold Field

The loop rolling threshold determines when a wide signal or parameter should be wrapped into a `for` loop and when it should be generated as a separate statement for each element of the signal. The default threshold value is 5.

For example, consider the model below



The gain parameter of the Gain block is the vector `myGainVec`.



Gain

Element-wise gain (y = K.*u) or matrix gain (y = K*u or y = u*K).

Parameters

Gain:

myGainVec

Multiplication: K.*u

☑ Saturate on integer overflow

Assume that the loop rolling threshold value is set to the default, 5.

If `myGainVec` is declared as

```
myGainVec = [1:10];
```

an array of 10 elements, `rtP.Gain_Gain[]` is declared within the `Parameters` data structure, `rtP`. The size of the gain array exceeds the loop rolling threshold. Therefore the code generated for the Gain block iterates over the array in a `for` loop, as shown in the following code fragment.

```
/* Gain Block: <Root>/Gain */
{
  int_T i1;

  real_T *y0 = &rtB.Gain[0];
  const real_T *p_Gain_Gain = &rtP.Gain_Gain[0];
```

```
    for (i1=0; i1 < 10; i1++) {
       y0[i1] = rtb_foo * (p_Gain_Gain[i1]);
       }
  }
```

If myGainVec is declared as

```
myGainVec = [1:3];
```

an array of 3 elements, rtP.Gain_Gain[] is declared within the Parameters data structure, rtP. The size of the gain array is below the loop rolling threshold. The generated code consists of inline references to each element of the array, as in the code fragment below.

```
/* Gain Block: <Root>/Gain */
rtB.Gain[0] = rtb_foo * (rtP.Gain_Gain[0]);
rtB.Gain[1] = rtb_foo * (rtP.Gain_Gain[1]);
rtB.Gain[2] = rtb_foo * (rtP.Gain_Gain[2]);
```

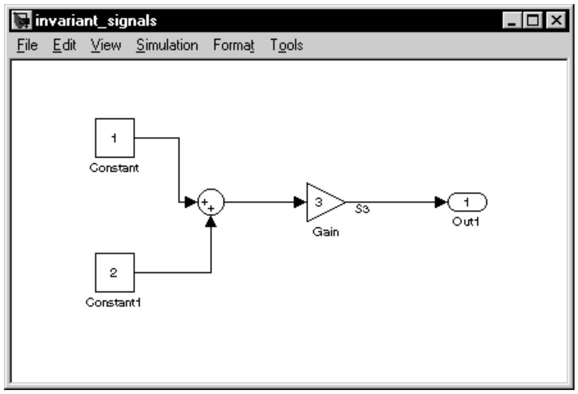
See the *Target Language Compiler Reference Guide* for more information on loop rolling.

Verbose Builds Option

If this option is selected, the MATLAB command window displays progress information during code generation; compiler output is also made visible.

Inline Invariant Signals Option

An invariant signal is a block output signal that does not change during Simulink simulation. For example, the signal S3 in this block diagram is an invariant signal.



Note The **Inline invariant signals** option is unavailable unless the **Inline parameters** option (on the Advanced page) is selected.

Given the model above, if both **Inline parameters** and **Inline invariant signals** is selected, the Real-time Workshop inlines the invariant signal S3 in the generated code.

Note that an *invariant signal is* not the same as an *invariant constant*. (See the *Using Simulink* manual for information on invariant constants.) In the above example, the two constants (1 and 2) and the gain value of 3 are invariant constants. To inline these invariant constants, select **Inline parameters**.

Local Block Outputs Option

When this option is selected, block signals will be declared locally in functions instead of being declared globally (when possible).

Note This check box is disabled when the **Signal storage reuse** item on the Advanced page is turned off.

For further information on the use of the **Local block outputs** option, see:

- "Signals: Storage, Optimization, and Interfacing" on page 3-65
- "Tutorial 4: A First Look at Generated Code" on page 1–56

Force Generation of Parameter Comments Option

The **Force generation of parameter comments** option controls the generation of comments in the model parameter structure declaration in `*model*_prm.h`. Parameter comments indicate parameter variable names and the names of source blocks.

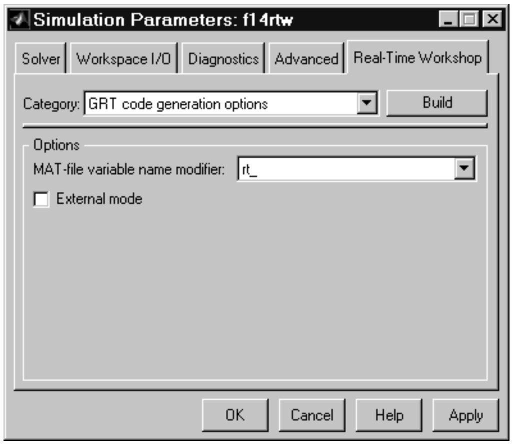
When this option is off (the default), parameter comments are generated if less than 1000 parameters are declared. This reduces the size of the generated file for models with a large number of parameters.

When this option is on, parameter comments are generated regardless of the number of parameters.

Target Specific Code Generation Options

Different target configurations support different code generation options that are not supported by all available targets. For example, the `grt`, `grt_malloc`, Tornado, xPC, and Real-Time Windows targets support external mode, but other targets do not.

This section summarizes the options specific to the generic real-time (GRT) target. For information on options specific to other targets, see the documentation relevant to those targets. "Available Targets" on page 3-36 lists targets and related chapters and manuals.



Category: GRT code generation options

MAT-file variable name modifier: rt_

☐ External mode

Figure 3-3: GRT Code Generation Options

MAT-file Variable Name Modifier Menu

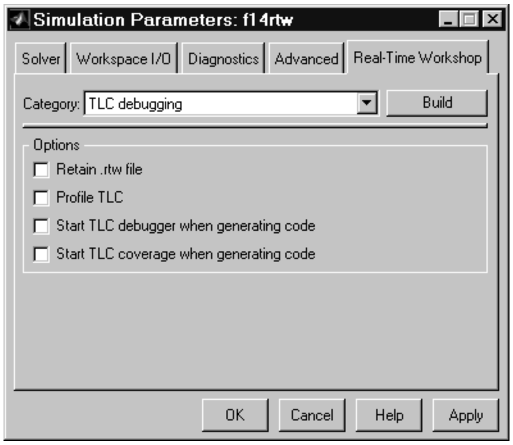
This menu selects a string to be added to the variable names used when logging data to MAT-files. You can select a prefix (`rt_`), suffix (`_rt`), or choose to have no modifier. The Real-Time Workshop prepends or appends the string chosen to the variable names for system outputs, states, and simulation time specified in the Workspace I/O page.

See “Workspace I/O Options and Data Logging” on page 3-18 for information on MAT-file data logging.

External Mode Option

Selecting this option turns on generation of code to support external mode communication between host and target systems. This option is available for GRT and other targets supporting external mode. For information see Chapter 5, “External Mode.”

TLC Debugging Options



Category: TLC debugging

Options
- [ ] Retain .rtw file
- [ ] Profile TLC
- [ ] Start TLC debugger when generating code
- [ ] Start TLC coverage when generating code

The TLC Debugging options are of interest to those who are writing TLC code. These options are summarized here; refer to the *Target Language Compiler Reference Manual* for details. The TLC Debugging options are:

- **Retain .rtw file**

  Normally, the build process deletes the *model*`.rtw` file from the build directory at the end of the build. When **Retain .rtw file** is selected, *model*`.rtw` is not deleted. This option is useful if you are modifying the target files, in which case you will need to look at the *model*`.rtw` file.

- **Profile TLC**

  When this option is selected, the TLC profiler analyzes the performance of TLC code executed during code generation, and generates a report. The report is in HTML format and can be read by your Web browser.

- **Start TLC debugger when generating code**

  This option starts the TLC debugger during code generation.This option is equivalent to entering the `-dc` argument into the **System Target File** field on the Real-Time Workshop page.

- **Start TLC coverage when generating code**

  When this option is selected, the Target Language Compiler generates a report containing statistics indicating how many times each line of TLC code is hit during code generation.

  This option is equivalent to entering the `-dg` argument into the **System Target File** field on the Real-Time Workshop page.

Real-Time Workshop Submenu

- The **Tools** menu of the Simulink window contains a **Real-Time Workshop** submenu. The submenu items are:
  - **Options**: Open the Real-Time Workshop page of the **Simulation Parameters** dialog.
  - **Build Model**: Initiate code generation and build process; equivalent to clicking the **Build** button in the Real-Time Workshop page.
  - **Build Subsystem**: Generate code and build an executable from a subsystem; enabled only when a subsystem is selected. See “Generating Code and Executables from Subsystems” on page 3-49.
  - **Generate S-Function**: Generate code and build an S-function from a subsystem; enabled only when a subsystem is selected. See “Automated S-Function Generation” on page 10-12.

Simulation Parameters and Code Generation

This section discusses how the simulation parameters of your model interact with Real-Time Workshop code generation. Only simulation parameters that affect code generation are mentioned here. For a full description of simulation parameters, see the *Using Simulink* manual.

This discussion is organized around the following pages of the **Simulation Parameters** dialog box:

- Solver page
- Workspace I/O page
- Diagnostics page
- Advanced Page

To view these pages, choose **Simulation parameters** from the **Simulation** menu. When the dialog box opens, click the appropriate tab.

Solver Options

Solver Type. If you are using the S-Function Target, you can specify either a fixed-step or a variable-step solver. All other targets require a fixed-step solver.

Mode. Real-Time Workshop supports both single- and multitasking modes. See Chapter 7, “Models with Multiple Sample Rates” for full details.

Start and Stop Times. The stop time must be greater than or equal to the start time. If the stop time is zero, or if the total simulation time (`Stop - Start`) is less than zero, the generated program runs for one step. If the stop time is set to `inf`, the generated program runs indefinitely.

Note that when using the GRT or Tornado targets, you can override the stop time when running a generated program from the DOS or UNIX command line. To override the stop time that was set during code generation, use the `-tf` switch.

```
modelname -tf n
```

The program will run for `n` seconds. If `n = inf`, the program will run indefinitely. See “Part 3: Running the External Mode Target Program” on page 5-11 for an example of the use of this option.

Note Certain blocks have a dependency on absolute time. If you are designing a program that is intended to run indefinitely (Stop time = `inf`), you must not use these blocks. See Appendix A for a list of blocks that depend on absolute time.

Workspace I/O Options and Data Logging

This section discusses several different methods by which a Real-Time Workshop generated program can save data to a MAT-file for later analysis. These methods include:

- Using the Workspace I/O page to define and log workspace return variables
- Logging data from Scope and To Workspace blocks
- Logging data using To File blocks

"Tutorial 2: Data Logging" on page 1-49 is an exercise designed to give you hands-on experience with data logging features of the Real-Time Workshop.

Note Data logging is available only for targets that have access to a file system.

Logging States, Time, and Outputs via the Workspace I/O Page

The Workspace I/O page enables a generated program to save system states, outputs, and simulation time at each model execution time step. The data is written to a MAT-file, named (by default) *model*`.mat`.

Before using this data logging feature, you should learn how to configure a Simulink model to return output to the MATLAB workspace. This is discussed in the *Using Simulink* manual.

For each workspace return variable that you define and enable, Real-Time Workshop defines a MAT-file variable. For example, if your model saves simulation time to the workspace variable `tout`, your generated program will log the same data to a variable named (by default) `rt_tout`.

The Real-Time Workshop logs the following data:

- All root Outport blocks

  The default MAT-file variable name for system outputs is `rt_yout`.

  The sort order of the `rt_yout` array is based on the port number of the Outport block, starting with 1.
- All continuous and discrete states in the model

  The default MAT-file variable name for system states is `rt_xout`.
- Simulation time

  The default MAT-file variable name for simulation time is `rt_tout`.

Real-Time Workshop data logging follows the Workspace I/O **Save options**: (**Limit data points**, **Decimation**, and **Format**).

Overriding the Default MAT-File Name. The MAT-file name defaults to *model*`.mat`. To specify a different filename:

1 Choose **Simulation parameters** from the **Simulation** menu. The dialog box opens. Click the **Real-Time Workshop** tab.

2 Append the following option to the existing text in the **Make command** field.

```
OPTS="-DSAVEFILE=filename"
```

Overriding Default MAT-File Variable Names. By default, the Real-Time Workshop prepends the string `rt_` to the variable names for system outputs, states, and simulation time to form MAT-file variable names. To change this prefix:

1 Choose **Simulation parameters** from the **Simulation** menu. The dialog box opens. Click the **Real-Time Workshop** tab.

2 Select the target-specific code generation options item from the **Category** menu.

3 Select a prefix(`rt_`) or suffix (`_rt`) from the **MAT-file variable name modifier** field, or choose none for no prefix.

Logging Data with Scope and To Workspace Blocks

The Real-Time Workshop also logs data from these sources:

- All Scope blocks that have the `save data to workspace` option enabled

  You must specify the variable name and data format in each Scope block's dialog box.
- All To Workspace blocks in the model

  You must specify the variable name and data format in each To Workspace block's dialog box.

The variables are written to *model*`.mat`, along with any variables logged from the Workspace I/O page.

Logging Data with To File Blocks

You can also log data to a To File block. The generated program creates a separate MAT-file (distinct from *model*`.mat`) for each To File block in the model. The file contains the block's time and input variable(s). You must specify the filename, variable name(s), decimation, and sample time in the To File block's dialog box.

Note that the To File block cannot be used in DOS real-time targets because of limitations of the DOS target.

Data Logging Differences in Single- and Multitasking Models

When logging data in singletasking and multitasking systems, you will notice differences in the logging of:

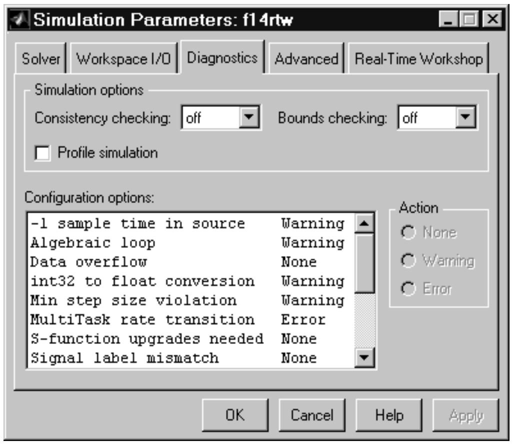
- Noncontinuous root Outport blocks
- Discrete states

In multitasking mode, the logging of states and outputs is done after the first task execution (and not at the end of the first time step). In singletasking mode, the Real-Time Workshop logs states and outputs after the first time step.

See "Data Logging In Single- and Multitasking Model Execution" on page 6–14 for more details on the differences between single- and multitasking data logging.

Note The rapid simulation target (`rsim`) provides enhanced logging options. See Chapter 11, “Real-Time Workshop Rapid Simulation Target” for more information.

Diagnostics Page Options



Simulation options

Consistency checking: off   Bounds checking: off

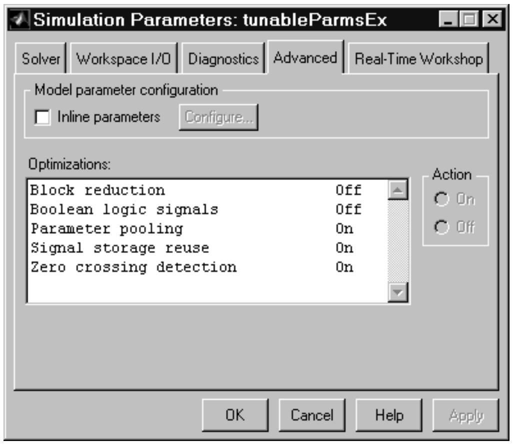
☐ Profile simulation

Configuration options:

```
-1 sample time in source     Warning
Algebraic loop               Warning
Data overflow                None
int32 to float conversion    Warning
Min step size violation      Warning
MultiTask rate transition    Error
S-function upgrades needed   None
Signal label mismatch        None
```

The Diagnostics page specifies what action should be taken when various model conditions such as unconnected ports are encountered. You can specify whether to ignore a given condition, issue a warning, or raise an error. If an error condition is encountered during a build, the build is terminated. The Diagnostics page is fully documented in the *Using Simulink* manual.

Advanced Options Page



```
Block reduction              Off
Boolean logic signals        Off
Parameter pooling            On
Signal storage reuse         On
Zero crossing detection      On
```

The Advanced page includes several options that affect the performance of generated code. The Advanced page has two sections. Options in the **Model parameter configuration** section let you specify how block parameters are represented in generated code, and how they are interfaced to externally written code. Options in the **Optimizations** section help you to optimize both memory usage and code size and efficiency.

Note that the **Zero crossing detection** option affects only simulations with variable step solvers. Therefore, this option is not applicable to code generation. See *Using Simulink* for further information on the **Zero crossing detection** option.

Inline Parameters Option

Selecting this option has two effects:

1. The Real-Time Workshop uses the numerical values of model parameters, instead of their symbolic names, in generated code.

   If the value of a parameter is a workspace variable, or an expression including one or more workspace variables, the variable or expression is evaluated at code generation time. The hard-coded result value appears in the generated code. An inlined parameter, since it has in effect been transformed into a constant, is no longer tunable. That is, it is not visible to externally-written code, and its value cannot be changed at run-time.

2 The **Configure** button becomes enabled. Clicking the **Configure** button opens the **Model Parameter Configuration** dialog.

   The **Model Parameter Configuration** dialog lets you remove individual parameters from inlining and declare them to be tunable variables (or global constants). When you declare a parameter tunable, the Real-Time Workshop generates a storage declaration that allows the parameter to be interfaced to externally-written code. This enables your hand-written code to change the value of the parameter at run-time.

   The **Model Parameter Configuration** dialog lets you improve overall efficiency by inlining most parameters, while at the same time retaining the flexibility of run-time tuning for selected parameters.

   See "Parameters: Storage, Interfacing, and Tuning" on page 3-51 for further information on interfacing parameters to externally-written code.

Inline parameters also instructs Simulink to propagate constant sample times. Simulink computes the output signals of blocks that have constant sample times once during model startup. This improves performance, since such blocks do not compute their outputs at every time step of the model.

Selecting **Inline parameters** also interacts with other code generation parameters as follows:

- When **Inline parameters** is selected, the **Inline invariant signals** code generation option becomes available. See "Inline Invariant Signals Option" on page 3-12.
- You cannot inline parameters when using external mode. External mode requires that all parameters be tunable. See Chapter 5, "External Mode."
- The **Parameter pooling** option is used only when **Inline parameters** is selected. See "Parameter Pooling Option" on page 3-24.

Block Reduction Option

When this option is selected, Simulink collapses certain groups of blocks into a single, more efficient block, or removes them entirely. This results in faster model execution during simulation and in generated code. The appearance of the source model does not change. The types of block reduction optimizations currently supported are:

- Accumulator folding: Simulink recognizes certain constructs as accumulators, and reduces them to a single block. For a detailed example, see “Accumulators” on page 8-15.
- Unnecessary type conversion blocks are removed. For example, an `int` type conversion block whose input and output are of type `int` is redundant and will be removed.

Boolean Logic Signals Option

By default, Simulink does not signal an error when it detects that double signals are connected to blocks that prefer Boolean input. This ensures compatibility with models created by earlier versions of Simulink that support only double data type. You can enable strict Boolean type checking by selecting the **Boolean logic signals** option.

Selecting this option is recommended. Generated code will require less memory, because a Boolean signal typically requires one byte of storage while a double signal requires eight bytes of storage.

Parameter Pooling Option

Parameter pooling occurs when multiple block parameters refer to storage locations that are separately defined but structurally identical. The optimization is similar to that of a C compiler that encounters declarations such as

```
int a[] = {1,2,3};
int b[] = {1,2,3};
```

In such a case, an optimizing compiler would collapse `a` and `b` into a single text location containing the values `1,2,3` and initialize `a` and `b` from the same code.

To understand the effect of parameter pooling in the Real-Time Workshop, consider the following scenario.

Assume that the MATLAB workspace variables a and b are defined as follows.

```
a = [1:1000]; b = [1:1000];
```

Suppose that a and b are used as vectors of input and output values in two Look-Up Table blocks in a model. Figure 3-4 shows the model.

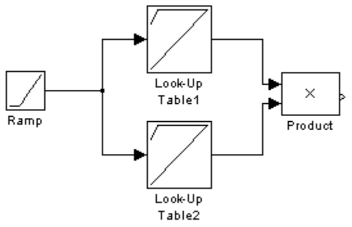
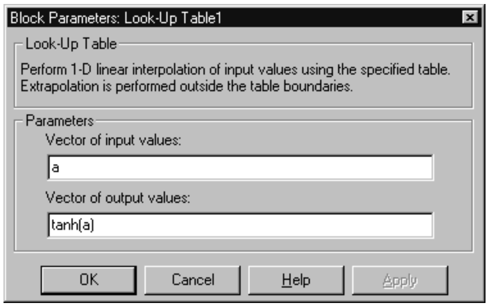


Figure 3-4: Model with Pooled Storage for Look-Up Table Blocks

Figure 3-5 shows the use of `a` and `b` as a parameters of `Look-Up Table1`and `Look-Up Table2`.



Block Parameters: Look-Up Table1

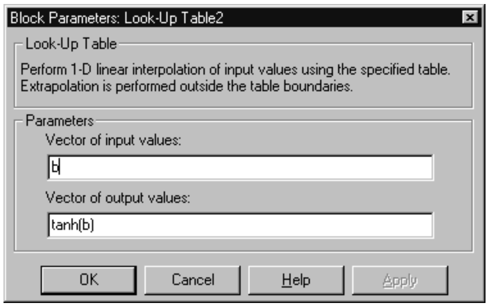
Look-Up Table

Perform 1-D linear interpolation of input values using the specified table. Extrapolation is performed outside the table boundaries.

Parameters

Vector of input values: a

Vector of output values: tanh(a)



Block Parameters: Look-Up Table2

Look-Up Table

Perform 1-D linear interpolation of input values using the specified table. Extrapolation is performed outside the table boundaries.

Parameters

Vector of input values: b

Vector of output values: tanh(b)

Figure 3-5: Pooled Storage in Look-Up Table Blocks

If **Parameter pooling** is on, pooled storage is used for the input/output data of the Look-Up Table blocks. The following code fragment shows the definition of the global parameter structure of the model (`rtP`). The input data references to `a` and `b` are pooled in the field `rtP.p2`. Likewise, while the output data references (expressions including `a` and `b`) are pooled in the field `rtP.p3`.

```
typedef struct Parameters_tag {
real_T p2[1000]; /* Variable: p2
                  * External Mode Tunable: no
                  * Referenced by blocks:
                  * <Root>/Look-Up Table1
                  * <Root>/Look-Up Table2
                  */
real_T p3[1000]; /* Expression: tanh(a)
                  * External Mode Tunable: no
                  * Referenced by blocks:
                  * <Root>/Look-Up Table1
                  * <Root>/Look-Up Table2
                  */
} Parameters;
```

If **Parameter pooling** is off, separate arrays are declared for the input/output data of the Look-Up Table blocks. Twice the amount of storage is used.

```
typedef struct Parameters_tag {
  real_T root_Look_Up_Table1_XData[1000];
  real_T root_Look_Up_Table1_YData[1000];
  real_T root_Look_Up_Table2_XData[1000];
  real_T root_Look_Up_Table2_YData[1000];
} Parameters;
```

The **Parameter pooling** option has the following advantages:

- Reduces ROM size
- Reduces RAM size for all compilers (`rtP` is a global vector)
- Speeds up code generation by reducing the size of *model*`.rtw`
- Can speed up execution

Note that the generated parameter names consist of the letter `p` followed by a number generated by the Real-Time Workshop. Comments indicate what parameters are pooled.

Note The **Parameter pooling** option affects code generation only when **Inline parameters** is on.

Signal Storage Reuse Option

This option instructs the Real-Time Workshop to reuse signal memory. This reduces the memory requirements of your real-time program. Selecting this option is recommended. Disabling **Signal storage reuse** makes all block outputs global and unique, which in many cases significantly increases RAM and ROM usage.

For further details on the **Signal storage reuse** option, see “Signals: Storage, Optimization, and Interfacing” on page 3-65.

Note Selecting **Signal storage reuse** also enables the **Local block outputs** option in the **General code generation options** category of the Real-Time Workshop page. See “Local Block Outputs Option” on page 3-12.

Tracing Generated Code Back to Your Simulink Model

The Real-Time Workshop writes system/block identification tags in the generated code. The tags are designed to help you identify the block, in your source model, that generated a given line of code. Tags are located in comment lines above each line of generated code.

The tag format is `<system>/block_name`, where:

- `system` is either:
  - the string `'root'`, or
  - a unique system number assigned by Simulink
- `block_name` is the name of the block.

The following code fragment illustrates a tag comment adjacent to a line of code generated by a Gain block at the root level of the source model.

```
/* Gain Block: <Root>/Gain */
rtb_temp3 *= (rtP.root_Gain_Gain);
```

The following code fragment illustrates a tag comment adjacent to a line of code generated by a Gain block within a subsystem one level below the root level of the source model.

```
/* Gain Block: <S1>/Gain */
rtB.temp0 *= (rtP.s1_Gain_Gain);
```

In addition to the tags, the Real-Time Workshop documents the tags for each model in comments in the generated header file *model*.h. The following illustrates such a comment, generated from a source model, foo. foo has a subsystem Outer with a nested subsystem Inner.

```
/* Here is the system hierarchy for this model.
 *
 * <Root> : foo
 * <S1>   : foo/Outer
 * <S2>   : foo/Outer/Inner
 */
```

To trace a tag back to the generating block:

1 Open the source model.

2 Close any other model windows that are open.

3 Use the MATLAB hilite_system command to view the desired system and block.

As an example, consider the model foo mentioned above. If foo is open,

```
hilite_system('<S1>')
```

opens the subsystem Outer and

```
hilite_system('<S2>/Gain1')
```

opens the subsystem Outer and selects and highlights the Gain block Gain1 within that subsystem.

Other Interactions Between Simulink and the Real-Time Workshop

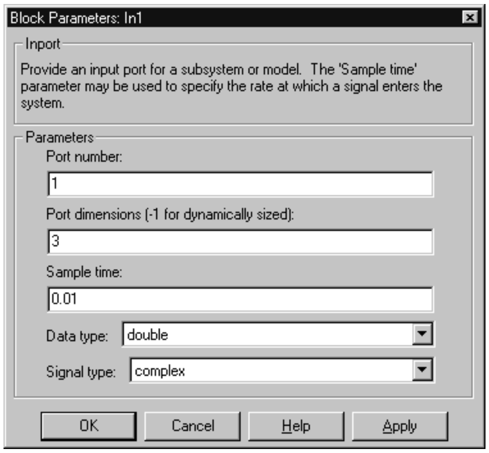
The Simulink engine propagates data from one block to the next along signal lines. The data propagated are:

- Data type
- Line widths

• Sample times

The first stage of code generation is compilation of the block diagram. This compile stage is analogous to that of a C program. The C compiler carries out type checking and pre-processing. Similarly, Simulink verifies that input/output data types of block ports are consistent, line widths between blocks are of the correct thickness, and the sample times of connecting blocks are consistent.

The Simulink engine typically derives signal attributes from a source block. For example, the Inport block's parameters dialog box specifies the signal attributes for the block.


Block Parameters: In1

Inport

Provide an input port for a subsystem or model. The 'Sample time' parameter may be used to specify the rate at which a signal enters the system.

Parameters

Port number: 1

Port dimensions (-1 for dynamically sized): 3

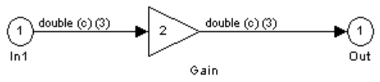
Sample time: 0.01

Data type: double

Signal type: complex

In this example, the Inport block has a port width of 3, a sample time of .01 seconds, the data type is double, and the signal is complex.

This figure shows the propagation of the signal attributes associated with the Inport block through a simple block diagram.



In this example, the Gain and Outport blocks inherit the attributes specified for the Inport block.

Sample Time Propagation

Inherited sample times in source blocks (e.g., a root inport) can sometimes lead to unexpected and unintended sample time assignments. Since a block may specify an inherited sample time, information available at the outset is often insufficient to compile a block diagram completely. In such cases, the Simulink engine propagates the known or assigned sample times to those blocks that have inherited sample times but which have not yet been assigned a sample time. Thus, Simulink continues to fill in the blanks (the unknown sample times) until sample times have been assigned to as many blocks as possible. Blocks that still do not have a sample time are assigned a default sample time according to the following rules:

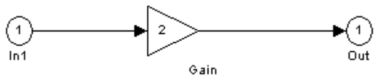
1. If the current system has at least one rate in it, the block is assigned the fastest rate.

2. If no rate exists and the model is configured for a variable-step solver, the block is assigned a continuous sample time (but fixed in minor time steps). Note that the Real-Time Workshop (with the exception of the S-Function Target) does not currently support variable-step solvers.

3. If no rate exists and the model is configured for a fixed-step solver, the block is assigned a discrete sample time of $(T_f - T_i)/50$, where T_i is the simulation start time and T_f is the simulation stop time. If T_f is infinity, the default sample time is set to 0.2.

To ensure a completely deterministic model (one where no sample times are set using the above rules), you should explicitly specify the sample time of all your source blocks. Source blocks include root inport blocks and any blocks without input ports. You do not have to set subsystem input port sample times. You may want to do so, however, when creating modular systems.

An unconnected input implicitly sources ground. For ground blocks and ground connections, the default sample time is derived from destination blocks or the default rule.

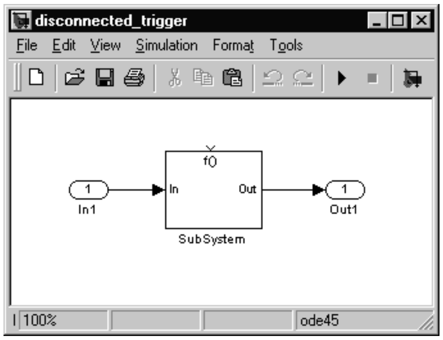


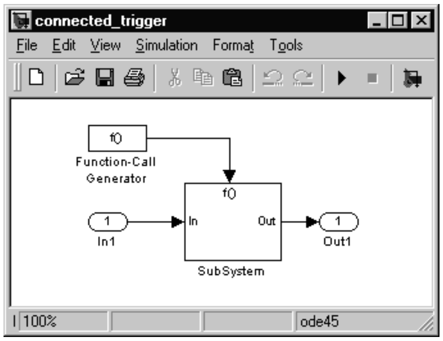
All blocks have an inherited sample time (T_s = -1). They will all be assigned a sample time of (T_f - T_i)/50.

Block Execution Order

Once Simulink compiles the block diagram, it creates a *model*`.rtw` file (analogous to an object file generated from a C file). The *model*`.rtw` file contains all the connection information of the model, as well as the necessary signal attributes. Thus, the timing engine in the Real-Time Workshop knows when blocks with different rates should be executed.

You cannot override this execution order by directly calling a block (in hand-written code) in a model. For example, the `disconnected_trigger` model below will have its trigger port source to ground, which may lead to all blocks inheriting a constant sample time. Calling the trigger function, `f()`, directly from hand-code will not work correctly and should never be done. Instead, you should use a function-call generator to properly specify the rate at which `f()` should be executed, as shown in the `connected_trigger` model below.





Instead of the function-call generator, you could use any other block that can drive the trigger port. Then, you should call the model's main entry point to execute the trigger function.

For multirate models, a common use of the Real-Time Workshop is to build individual models separately and then hand-code the I/O between the models. This approach places the burden of data consistency between models on the developer of the models. Another approach is to let Simulink and the Real-Time Workshop ensure data consistency between rates and generate multirate code for use in a multitasking environment. The Real-Time Workshop Interrupt Template and VxWorks Support libraries provide blocks which allow synchronous and asynchronous data flow. For a description of the Real-Time Workshop libraries, see Chapter 14, "Custom Code Blocks" and Chapter 15, "Asynchronous Support." For more information on multi-rate code generation, see Chapter 7, "Models with Multiple Sample Rates."

Selecting a Target Configuration

The process of generating target-specific code is controlled by a configuration of:

- A system target file
- A template makefile
- A `make` command

The System Target File Browser lets you specify such a configuration in a single step, choosing from a wide variety of ready-to-run configurations.

The System Target File Browser

To select a target configuration using the System Target File Browser:

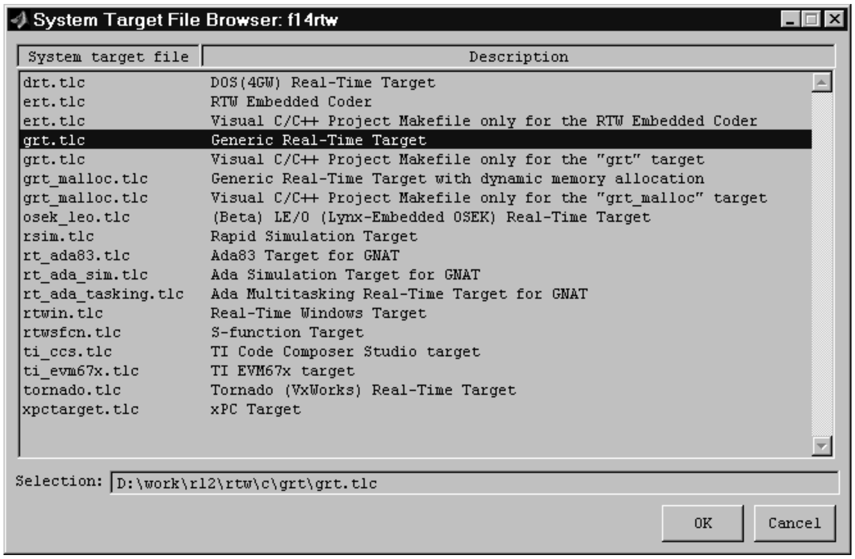
1. Click the **Real-Time Workshop** tab of the **Simulation Parameters** dialog box. The Real-Time Workshop page activates.

2. Select **Target configuration** from the **Category** menu.

3. Click the **Browse** button next to the **System target file** field. This opens the System Target File Browser. The browser displays a list of all currently available target configurations. When you select a target configuration, the Real-Time Workshop automatically chooses the appropriate system target file, template makefile, and `make` command.

   Figure 3-6 shows the System Target File Browser with the generic real-time target selected.

4. Double-click on the desired entry in the list of available configurations. Alternatively, you can select the desired entry in the list and click **OK**.



System target file	Description
drt.tlc	DOS(4GW) Real-Time Target
ert.tlc	RTW Embedded Coder
ert.tlc	Visual C/C++ Project Makefile only for the RTW Embedded Coder
grt.tlc	Generic Real-Time Target
grt.tlc	Visual C/C++ Project Makefile only for the "grt" target
grt_malloc.tlc	Generic Real-Time Target with dynamic memory allocation
grt_malloc.tlc	Visual C/C++ Project Makefile only for the "grt_malloc" target
osek_leo.tlc	(Beta) LE/O (Lynx-Embedded OSEK) Real-Time Target
rsim.tlc	Rapid Simulation Target
rt_ada83.tlc	Ada83 Target for GNAT
rt_ada_sim.tlc	Ada Simulation Target for GNAT
rt_ada_tasking.tlc	Ada Multitasking Real-Time Target for GNAT
rtwin.tlc	Real-Time Windows Target
rtwsfcn.tlc	S-function Target
ti_ccs.tlc	TI Code Composer Studio target
ti_evm67x.tlc	TI EVM67x target
tornado.tlc	Tornado (VxWorks) Real-Time Target
xpctarget.tlc	xPC Target

Selection: D:\work\r12\rtw\c\grt\grt.tlc

Figure 3-6: The System Target File Browser

5 When you choose a target configuration, the Real-Time Workshop automatically chooses the appropriate system target file, template makefile, and make command for the selected target, and displays them in the Real-Time Workshop page.

Available Targets

Table 3-10 lists all the supported system target files and their associated code formats, template makefiles, and make commands. The table also gives references to relevant manuals or chapters in this book.

Table 3-1: Targets Available from the System Target File Browser

Target/Code Format	System Target File	Template Makefile	Make Command	Relevant Chapters
RTW Embedded Coder (PC or UNIX)	`ert.tlc`	`ert_default_tmf`	`make_rtw`	9 and 4
RTW Embedded Coder for Watcom	`ert.tlc`	`ert_watc.tmf`	`make_rtw`	9 and 4
RTW Embedded Coder for Visual C/ C++	`ert.tlc`	`ert_vc.tmf`	`make_rtw`	9 and 4
RTW Embedded Coder for Visual C/ C++ Project Makefile	`ert.tlc`	`ert_msvc.tmf`	`make_rtw`	9 and 4
RTW Embedded Coder for Borland	`ert.tlc`	`ert_bc.tmf`	`make_rtw`	9 and 4
RTW Embedded Coder for LCC	`ert.tlc`	`ert_lcc.tmf`	`make_rtw`	9 and 4
RTW Embedded Coder for UNIX	`ert.tlc`	`ert_unix.tmf`	`make_rtw`	9 and 4
Generic Real-Time for PC/UNIX	`grt.tlc`	`grt_default_tmf`	`make_rtw`	4
Generic Real-Time for Watcom	`grt.tlc`	`grt_watc.tmf`	`make_rtw`	4
Generic Real-Time for Visual C/C++	`grt.tlc`	`grt_vc.tmf`	`make_rtw`	4

Table 3-1: Targets Available from the System Target File Browser (Continued)

Target/Code Format	System Target File	Template Makefile	Make Command	Relevant Chapters
Generic Real-Time for Visual C/C++ Project Makefile	`grt.tlc`	`grt_msvc.tmf`	`make_rtw`	4
Generic Real-Time for Borland	`grt.tlc`	`grt_bc.tmf`	`make_rtw`	4
Generic Real-Time for LCC	`grt.tlc`	`grt_lcc.tmf`	`make_rtw`	4
Generic Real-Time for UNIX	`grt.tlc`	`grt_unix.tmf`	`make_rtw`	4
Generic Real-Time (dynamic) for PC/ UNIX	`grt_malloc.tlc`	`grt_malloc_default_ tmf`	`make_rtw`	4
Generic Real-Time (dynamic) for Watcom	`grt_malloc.tlc`	`grt_malloc_watc.tmf`	`make_rtw`	4
Generic Real-Time (dynamic) for Visual C/C++	`grt_malloc.tlc`	`grt_malloc_vc.tmf`	`make_rtw`	4
Generic Real-Time (dynamic) for Visual C/C++ Project Makefile	`grt_malloc.tlc`	`grt_malloc_msvc.tmf`	`make_rtw`	4
Generic Real-Time (dynamic) for Borland	`grt_malloc.tlc`	`grt_malloc_bc.tmf`	`make_rtw`	4
Generic Real-Time (dynamic) for LCC	`grt_malloc.tlc`	`grt_malloc_lcc.tmf`	`make_rtw`	4
Generic Real-Time (dynamic) for UNIX	`grt_malloc.tlc`	`grt_malloc_unix.tmf`	`make_rtw`	4

Table 3-1: Targets Available from the System Target File Browser (Continued)

Target/Code Format	System Target File	Template Makefile	Make Command	Relevant Chapters
Rapid Simulation Target (default for PC or UNIX)	`rsim.tlc`	`rsim_default_tmf`	`make_rtw`	11
Rapid Simulation Target for Watcom	`rsim.tlc`	`rsim_watc.tmf`	`make_rtw`	11
Rapid Simulation Target for Visual C/C++	`rsim.tlc`	`rsim_vc.tmf`	`make_rtw`	11
Rapid Simulation Target for Borland	`rsim.tlc`	`rsim_bc.tmf`	`make_rtw`	11
Rapid Simulation Target for LCC	`rsim.tlc`	`rsim_lcc.tmf`	`make_rtw`	11
Rapid Simulation Target for UNIX	`rsim.tlc`	`rsim_unix.tmf`	`make_rtw`	11
Ada Simulation Target for GNAT	`rt_ada_sim.tlc`	`gnat_sim.tmf`	`make_rtw -ada`	16
Ada Real-Time Multitasking Target for GNAT	`rt_ada_tasking .tlc`	`gnat_tasking.tmf`	`make_rtw -ada`	16
S-Function Target for PC or UNIX	`rtwsfcn.tlc`	`rtwsfcn_default_tmf`	`make_rtw`	4
S-Function Target for Watcom	`rtwsfcn.tlc`	`rtwsfcn_watc.tmf`	`make_rtw`	4
S-Function Target for Visual C/C++	`rtwsfcn.tlc`	`rtwsfcn_vc.tmf`	`make_rtw`	4
S-Function Target for Borland	`rtwsfcn.tlc`	`rtwsfcn_bc.tmf`	`make_rtw`	4

Table 3-1: Targets Available from the System Target File Browser (Continued)

Target/Code Format	System Target File	Template Makefile	Make Command	Relevant Chapters
S-Function Target for LCC	`rtwsfcn.tlc`	`rtwsfcn_lcc.tmf`	`make_rtw`	4
Tornado (VxWorks) Real-Time Target	`tornado.tlc`	`tornado.tmf`	`make_rtw`	12
Windows 95/98/NT Real-Time Target for Watcom	`rtwin.tlc`	`win_watc.tmf`	`make_rtw`	*Real-Time Windows Target User's Guide*
Windows 95/98/NT Real-Time Target for Visual C/C++	`rtwin.tlc`	`win_vc.tmf`	`make_rtw`	*Real-Time Windows Target User's Guide*
Texas Instruments EVM67x Target	`evm67x.tlc`	`evm67x.tmf`	`make_rtw`	*Texas Instruments DSP Developer's Kit User's Guide*
Texas Instruments Code Composer Studio Target	`ccs.tlc`	`ccs.tmf`	`make_rtw`	*Texas Instruments DSP Developer's Kit User's Guide*
xPC Target for Watcom C/C++ or Visual C/C++	`xpctarget.tlc`	`xpc_default_tmf`	`make_xpc`	*xPC Target User's Guide*

Table 3-1: Targets Available from the System Target File Browser (Continued)

Target/Code Format	System Target File	Template Makefile	Make Command	Relevant Chapters
DOS (4GW)	`drt.tlc`	`drt_watc.tmf`	`make_rtw`	13 and 4
LE/O (Lynx embedded OSEK) Real-Time Target	`osek_leo.tlc`	`osek_leo.tmf`	`make_rtw MAT_FILE=1 RUN=1 LEO_NODE=osek`	Readme file in *matlabroot*/`rtw/c/osek_leo`

Note The LE/O and DOS targets are included as examples only.

Nonvirtual Subsystem Code Generation

The Real-Time Workshop allows you to control generation of code at the subsystem level, for any nonvirtual subsystem. The categories of nonvirtual subsystems are:

- *Conditionally executed* subsystems: execution depends upon a control signal. These include triggered subsystems, enabled subsystems, subsystems that are both triggered and enabled, and function call subsystems. See *Using Simulink* for information on conditionally executed subsystems.
- *Atomic* subsystems: Any virtual subsystem can be declared atomic (and therefore nonvirtual) via the **Treat as atomic unit** option in the **Block Parameters** dialog.

See *Using Simulink* for further information on virtual subsystems and atomic subsystems.

You can control the code generated from nonvirtual subsystems as follows:

- You can instruct the Real-Time Workshop to generate separate functions, within separate code files, for selected nonvirtual systems. You can control both the names of the functions and of the code files generated from nonvirtual subsystems.
- You can generate inlined code from selected nonvirtual subsystems within your model. When you inline a nonvirtual subsystem, a separate function call is not generated for the subsystem.

Nonvirtual Subsystem Code Generation Options

For any nonvirtual subsystem, you can choose the following code generation options from the **RTW system code** pop-up menu in the subsystem **Block parameters** dialog:

- `Auto`: This is the default option. See “Auto Option” below.
- `Inline`: This option explicitly directs the Real-Time Workshop to inline the subsystem.
- `Function`: This option explicitly directs the Real-Time Workshop to generate a separate function and file for the subsystem. In this case you can choose further options to control the naming of the generated function and file.

The sections below discuss the `Auto`, `Inline`, and `Function` options.

Auto Option

In the current release, the `Auto` option causes the Real-Time Workshop to inline the subsystem, unless it is a function-call subsystem with multiple callers. In that case, a function is generated.

In a future release, the `Auto` option will also create functions when multiple instances of a subsystem are detected. If there are multiple instances or multiple callers of the subsystem, the subsystem will not be inlined. To take advantage of this capability, you should choose the `Auto` option. Choose `Inline` or `Function` when you want specifically to inline a subsystem or generate a separate function and code module.

To use the `Auto` option:

1. Select the subsystem block. Then select **Subsystem parameters** from the Simulink **Edit** menu. The **Block Parameters** dialog opens, as shown in Figure 3-7.

   Alternatively, you can open the **Block Parameters** dialog by:

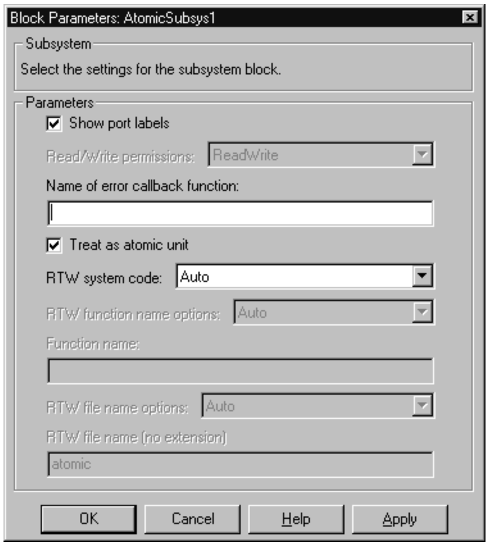
   - Shift-double-clicking on the Subsystem block
   - Right-clicking on the Subsystem block and selecting **Block parameters** from the pop-up menu.

2. If the subsystem is virtual, select **Treat as atomic unit** as shown in Figure 3-7. This makes the subsystem nonvirtual, and the **RTW system code** option becomes enabled.

   If the system is already nonvirtual, the **RTW system code** option is already enabled.

3. Select **Auto** from the **RTW system code** pop-up menu as shown in Figure 3-7.

4. Click **Apply** and close the dialog.


Block Parameters: AtomicSubsys1

Subsystem

Select the settings for the subsystem block.

Parameters

[✓] Show port labels

Read/Write permissions: ReadWrite

Name of error callback function:

[✓] Treat as atomic unit

RTW system code: Auto

RTW function name options: Auto

Function name:

RTW file name options: Auto

RTW file name (no extension)

atomic

Figure 3-7: Auto Code Generation Option for a Nonvirtual Subsystem

Inline Option

As noted above, subsystem code can be inlined only if the subsystem is nonvirtual.

Exceptions to Inlining. Note that there are certain cases in which the Real-Time Workshop will not inline a nonvirtual subsystem, even though the **Inline** option is selected. These cases are:

- If the subsystem is a function-call subsystem that is called by a noninlined S-function, the **Inline** option is ignored. Noninlined S-functions make such calls via function pointers; therefore the function-call subsystem must generate a function with all arguments present.
- In a feedback loop involving function-call subsystems, the Real-Time Workshop will force one of the subsystems to be generated as a function instead of inlining it. The Real-Time Workshop selects the subsystem to be generated as a function based on the order in which the subsystems are sorted internally.

- If a subsystem is called from an S-function block that sets the option `SS_OPTION_FORCE_NONINLINED_FCNCALL` to `TRUE`, it will not be inlined. This may be the case when user-defined Asynchronous Interrupt blocks or Task Synchronization blocks are required. Such blocks must be generated as functions. The VxWorks Asynchronous Interrupt and Task Synchronization blocks, shipped with the Real-Time Workshop, use the `SS_OPTION_FORCE_NONINLINED_FCNCALL` option.

To generate inlined subsystem code:

1. Select the subsystem block. Then select **Subsystem parameters** from the Simulink **Edit** menu. The **Block Parameters** dialog opens, as shown in Figure 3-8.

   Alternatively, you can open the **Block Parameters** dialog by:

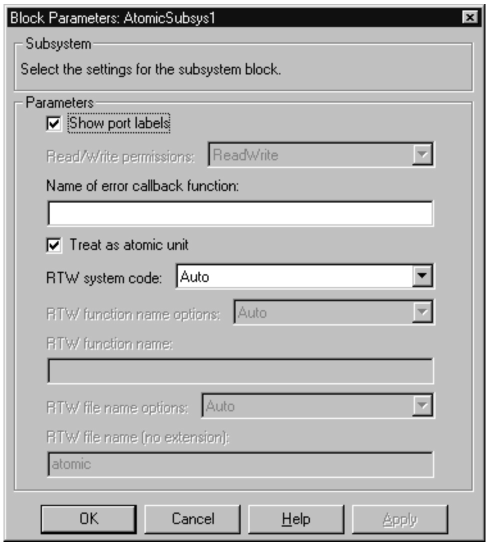
   - Shift-double-clicking on the Subsystem block
   - Right-clicking on the Subsystem block and selecting **Block parameters** from the pop-up menu.

2. If the subsystem is virtual, select **Treat as atomic unit** as shown in Figure 3-8. This makes the subsystem atomic, and the **RTW system code** pop-up menu becomes enabled.

   If the system is already nonvirtual, the **RTW system code** menu is already enabled.

3. Select **Inline** from the **RTW system code** menu as shown in Figure 3-8.

4. Click **Apply** and close the dialog.


Block Parameters: AtomicSubsys1

Subsystem
Select the settings for the subsystem block.

Parameters
☑ Show port labels
Read/Write permissions: ReadWrite
Name of error callback function:
☑ Treat as atomic unit
RTW system code: Auto
RTW function name options: Auto
RTW function name:
RTW file name options: Auto
RTW file name (no extension): atomic

Figure 3-8: Inlined Code Generation for a Nonvirtual Subsystem

When you generate code from your model, the Real-Time Workshop writes inline code within *model*.c to perform subsystem computations. You can identify this code by system/block identification tags, such as the following.

```
/* Atomic SubSystem Block: <Root>/AtomicSubsys1 */
```

See “Tracing Generated Code Back to Your Simulink Model” on page 3-28 for further information on system/block identification tags.

Function Option

This option lets you direct the Real-Time Workshop to generate a separate function and file for the subsystem. When you select the `Function` option, two additional options are enabled:

- The **RTW function name options** menu lets you control the naming of the generated function.
- The **RTW file name options** menu lets you control the naming of the generated file.

Figure 3-9 shows the **Block Parameters** dialog with the `Function` option selected.

RTW Function Name Options Menu. This menu offers the following choices:

- `Auto`: the Real-Time Workshop assigns a unique function name using the default naming convention: *model_systemid*`()`, where *systemid* is a sequential identifier (`s0, s1,...s`*n*) assigned by Simulink.
- `UseSubSystemName`: the Real-Time Workshop uses the subsystem name as the filename.
- `UserSpecified`: When this option is selected, the **RTW function name** text entry field is enabled. Enter any legal function name. Note that the function name must be unique.

RTW File Name Options Menu. This menu offers the following choices:

- `Auto`: the Real-Time Workshop assigns a unique filename using the default naming convention: *model_systemid*`.c`, where *systemid* is a sequential identifier (`s0, s1,...s`*n*) assigned by Simulink.
- `UseSubSystemName`: the Real-Time Workshop uses the subsystem name as the filename.
- `UseFunctionName`: the Real-Time Workshop uses the function name (as specified by the **RTW function name** options) as the filename.
- `UserSpecified`: When this option is selected, the **RTW file name (no extension)** text entry field is enabled. Enter any filename desired, but do not include the `.c` (or any other) extension. Note that this filename should be unique.

To generate a separate subsystem function and file:

1. Select the subsystem block. Then select **Subsystem parameters** from the Simulink **Edit** menu. The **Block Parameters** dialog opens, as shown in Figure 3-9.

   Alternatively, you can open the **Block Parameters** dialog by:

   - Shift-double-clicking on the Subsystem block
   - Right-clicking on the Subsystem block and selecting **Block parameters** from the pop-up menu.

2 If the subsystem is virtual, select **Treat as atomic unit** as shown in Figure 3-9. This makes the subsystem atomic, and the **RTW system code** menu becomes enabled.

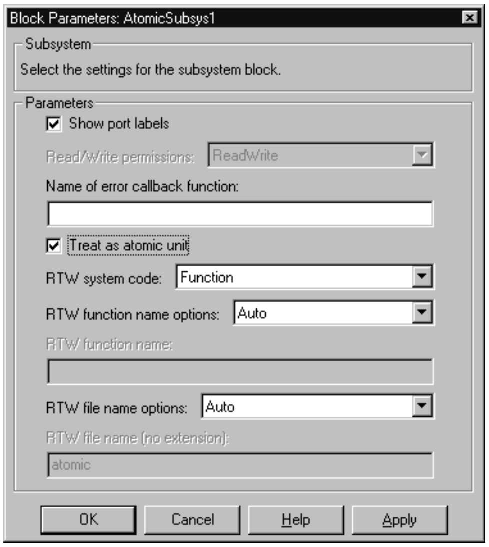
If the system is already nonvirtual, the **RTW system code** menu is already enabled.

3 Select **Function** from the **RTW system code** menu as shown in Figure 3-9.

4 Set the function name, using the **RTW function name** options described in "RTW Function Name Options Menu" on page 3-46.

5 Set the filename, using the **RTW file name** options described in "RTW File Name Options Menu" on page 3-46.

6 Click **Apply** and close the dialog.



Block Parameters: AtomicSubsys1

Subsystem

Select the settings for the subsystem block.

Parameters

☑ Show port labels

Read/Write permissions: ReadWrite

Name of error callback function:

☑ Treat as atomic unit

RTW system code: Function

RTW function name options: Auto

RTW function name:

RTW file name options: Auto

RTW file name (no extension): atomic

Figure 3-9: Subsystem Function Code Generation with Default Naming Options

Modularity of Subsystem Code

Note that code generated from nonvirtual subsystems, when written to separate files, is not completely independent of the generating model. For example, subsystem code may reference global data structures of the model. Each subsystem code file contains appropriate include directives and comments explaining the dependencies.

Generating Code and Executables from Subsystems

The Real-Time Workshop can generate code and build an executable from any subsystem within a model. The code generation and build process uses the code generation and build parameters of the root model.

To generate code and build an executable from a subsystem:

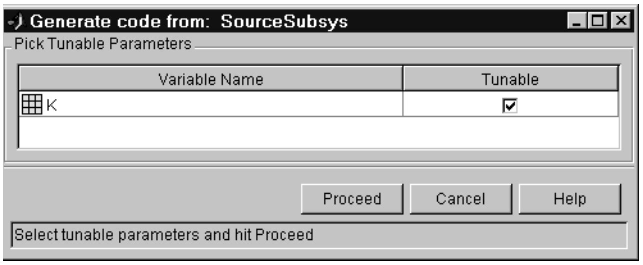
1 Set up the desired code generation and build parameters in the **Simulation Parameters** dialog, just as you would for code generation from a model.

2 Select the desired subsystem block.

3 Right-click on the subsystem block and select **Build Subsystem** from the **Real-Time Workshop** submenu of the subsystem block's context menu.

   Alternatively, you can select **Build Subsystem** from the **Real-Time Workshop** submenu of the **Tools** menu. This menu item is enabled when a subsystem is selected in the current model.

4 A window displaying a list of the subsystem parameters opens. You can declare any parameter to be tunable by selecting its check box in the **Tunable** column.)



Variable Name	Tunable
K	✓

Select tunable parameters and hit Proceed

   In the illustration above, the parameter K is declared tunable.

5 After selecting tunable parameters, click the **Proceed** button. This initiates the code generation and build process.

6 The build process displays status messages in the MATLAB command window. When the build completes, the generated executable is in your

working directory. The name of the generated executable is *subsystem*.exe (PC) or *subsystem* (UNIX), where *subsystem* is the name of the source subsystem block.

The generated code is in a build subdirectory, named *subsystem*_*target*_rtw, where *subsystem* is the name of the source subsystem block and *target* is the name of the target configuration.

Note You can generate subsystem code using any target configuration available in the System Target File Browser. However, if the S-function target is selected, **Build Subsystem** behaves identically to **Generate S-function**. (See "Automated S-Function Generation" on page 10-12.)

Parameters: Storage, Interfacing, and Tuning

Simulink external mode (see Chapter 5, "External Mode") offers a quick and easy way to monitor signals and modify parameter values while generated model code executes. However, external mode may not be appropriate for your application in some cases. Some targets (such as the Real-Time Workshop Embedded Coder) do not support external mode. In other cases, you may want your existing code to access parameters and signals of a model directly, rather than using the external mode communications mechanism.

This section discusses how the Real-Time Workshop generates parameter storage declarations, and how you can generate the storage declarations you need to interface block parameters to your code.

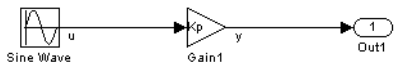
Storage of Nontunable Parameters

By default, block parameters are not tunable in the generated code. In the default case, the Real-Time Workshop has control of parameter storage declarations and the symbolic naming of parameters in the generated code.

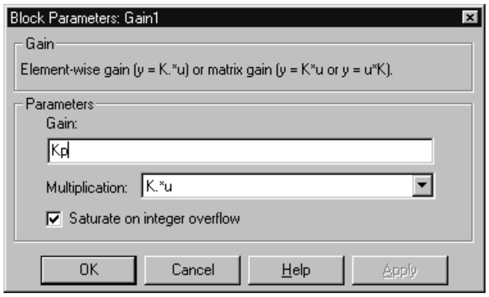
Nontunable parameters are stored as fields of a model-specific global parameter data structure called `rtP`. The Real-Time Workshop initializes each field of `rtP` to the value of the corresponding block parameter at code generation time.

When the **Inline parameters** option is on, block parameters are evaluated at code generation time, and their values appear as constants in the generated code. (A vector parameter, however, may be represented as an array of constants within `rtP`.)

As an example of nontunable parameter storage, consider this model.



The workspace variable `Kp` sets the gain of the `Gain1` block.



Block Parameters: Gain1

Gain

Element-wise gain (y = K.*u) or matrix gain (y = K*u or y = u*K).

Parameters

Gain:

Kp

Multiplication: K.*u

[✓] Saturate on integer overflow

Assume that `Kp` is nontunable, and has a value of 5.0. Table 3-2 shows the variable declarations and the code generated for `Kp` in the noninlined and inlined cases.

Notice that the generated code does not preserve the symbolic name `Kp`. The noninlined code represent the gain of the `Gain1` block as `rtP.Gain1_Gain`.

Table 3-2: Nontunable Parameter Storage Declarations and Code

Inline Parameters	Generated Variable Declaration and Code

Off	`typedef struct Parameters_tag {`
	`  real_T Sine_Wave_Amp;`
	`  real_T Sine_Wave_Freq;`
	`  real_T Sine_Wave_Phase;`
	`  real_T Gain1_Gain;`
	`} Parameters;`
	`.`
	`.`
	`Parameters rtP = {`
	` 1.0 , /*Sine_Wave_Amp :'<Root>/Sine Wave' */`
	` 1.0 , /*Sine_Wave_Freq:'<Root>/Sine Wave' */`
	` 0.0 , /*Sine_Wave_Phase:'<Root>/Sine Wave'*/`
	` 5.0   /*Gain1_Gain : '<Root>/Gain1' */`
	`};`
	`.`
	`.`
	`rtb_y = rtB.u * (rtP.Gain1_Gain);`

On	`rtb_y = rtB.u * (5.0);`

Tunable Parameter Storage

A *tunable* parameter is a block parameter whose value can be changed at run-time. A tunable parameter is inherently noninlined. A *tunable expression* is an expression that contains one or more tunable parameters.

When you declare a parameter tunable, you control whether or not the parameter is stored within `rtP`. You also control the symbolic name of the parameter in the generated code.

When you declare a parameter tunable, you specify:

- The *storage class* of the parameter.

  In the Real-Time Workshop, the storage class property of a parameter specifies how the Real-Time Workshop declares the parameter in generated code.

  Note that the term “storage class,” as used in the Real-Time Workshop, is not synonymous with the term *storage class specifier,* as used in the C language.
- A *storage type qualifier*, such as `const` or `volatile`. This is simply an string that is included in the variable declaration, without error checking.
- (Implicitly) the symbolic name of the variable or field in which the parameter is stored. The Real-Time Workshop derives variable and field names from the names of tunable parameters.

The Real-Time Workshop generates a variable or `struct` storage declaration for each tunable parameter. Your choice of storage class controls whether the parameter is declared as a member of `rtP` or as a separate global variable.

You can use the generated storage declaration to make the variable visible to your code. You can also make variables declared in your code visible to the generated code. You are responsible for properly linking your code to generated code modules.

You can use tunable parameters or expressions in your root model and in masked or unmasked subsystems, subject to certain restrictions (See “Tunable Expressions” on page 3-61.)

To declare tunable parameters, you must first enable the **Inline parameters** option. You then use the **Model Parameter Configuration** dialog to remove individual parameters from inlining and declare them to be tunable. This allows you to improve overall efficiency by inlining most parameters, while at the same time retaining the flexibility of run-time tuning for selected parameters.

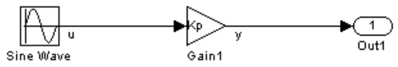
The mechanics of declaring tunable parameters is discussed in “Using the Model Parameter Configuration Dialog” on page 3-57.

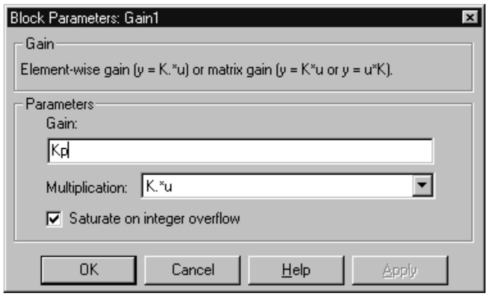
Storage Classes of Tunable Parameters

The Real-Time Workshop defines four storage classes for tunable parameters. You must declare a tunable parameter to have one of the following storage classes:

- `SimulinkGlobal(Auto)`: `SimulinkGlobal(Auto)` is the default storage class. The Real-Time Workshop stores the parameter as a member of `rtP`. Each member of `rtP` is initialized to the value of the corresponding workspace variable at code generation time.
- `ExportedGlobal`: The generated code instantiates and initializes the parameter and *model*`_export.h` exports it as a global variable. An exported global variable is independent of the `rtP` data structure. Each exported global variable is initialized to the value of the corresponding workspace variable at code generation time.
- `ImportedExtern`: *model*`.h` declares the parameter as an `extern` variable. Your code must supply the proper variable definition and initializer, if any.
- `ImportedExternPointer`: *model*`.h` declares the variable as an `extern` pointer. Your code must supply the proper pointer variable definition and initializer, if any.

As an example of how the storage class declaration affects the code generated for a parameter, consider the model shown below.




Block Parameters: Gain1

Gain

Element-wise gain (y = K.*u) or matrix gain (y = K*u or y = u*K).

Parameters

Gain:

Kp

Multiplication: K.*u

☑ Saturate on integer overflow

The workspace variable `Kp` sets the gain of the `Gain1` block. Assume that the value of `Kp` is 5.0. Table 3-3 shows the variable declarations and the code generated for the gain block when Kp is declared as a tunable parameter. An example is shown for each storage class.

Note that the symbolic name Kp is preserved in the variable and field names in the generated code.

Table 3-3: Tunable Parameter Storage Declarations and Code

Storage Class	Generated Variable Declaration and Code

SimulinkGlobal(Auto)	`typedef struct Parameters_tag {`
	`  real_T Kp;`
	`} Parameters;`
	`.`
	`.`
	`Parameters rtP = {`
	`  5.0`
	`};`
	`.`
	`.`
	`rtb_y = rtB.u * (rtP.Kp);`

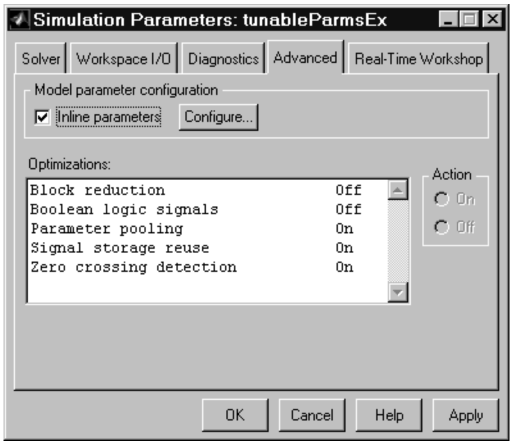
ExportedGlobal	`real_T Kp = 5.0;`
	`.`
	`.`
	`rtb_y = rtB.u * (Kp);`

ImportedExtern	`extern real_T Kp;`
	`.`
	`.`
	`rtb_y = rtB.u * (Kp);`

ImporteExternPointer	`extern real_T *Kp;`
	`.`
	`.`
	`rtb_y = rtB.u * ((*Kp));`

Using the Model Parameter Configuration Dialog

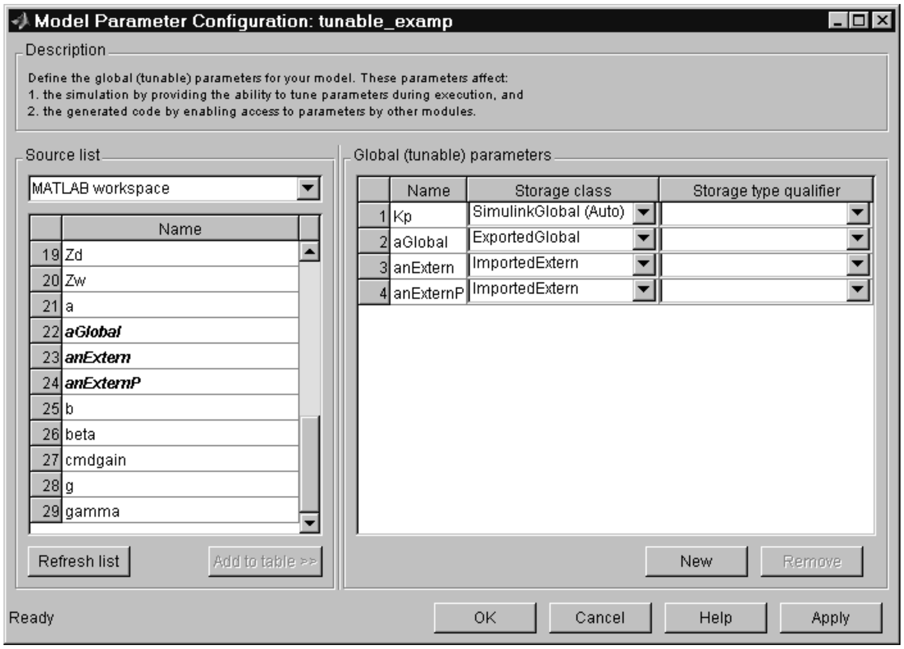
The **Model Parameter Configuration** dialog is available only when the **Inline parameters** option on the Advanced page is selected. Selecting this option activates the **Configure** button.



[✓] Inline parameters

```
Block reduction                  Off
Boolean logic signals            Off
Parameter pooling                On
Signal storage reuse             On
Zero crossing detection          On
```

Clicking on the **Configure** button opens the **Model Parameter Configuration** dialog.



Model Parameter Configuration: tunable_examp

Description

Define the global (tunable) parameters for your model. These parameters affect:
1. the simulation by providing the ability to tune parameters during execution, and
2. the generated code by enabling access to parameters by other modules.

Source list: MATLAB workspace

	Name
19	Zd
20	Zw
21	a
22	*aGlobal*
23	*anExtern*
24	*anExternP*
25	b
26	beta
27	cmdgain
28	g
29	gamma

Global (tunable) parameters

	Name	Storage class	Storage type qualifier
1	Kp	SimulinkGlobal (Auto)	
2	aGlobal	ExportedGlobal	
3	anExtern	ImportedExtern	
4	anExternP	ImportedExtern	

Ready

Figure 3-10: The Model Parameter Configuration Dialog

The **Model Parameter Configuration** dialog lets you select workspace variables and declare them to be tunable parameters in the current model. The dialog is divided into two panels:

- The **Global (tunable) parameters** panel displays and maintains a list of tunable parameters associated with the model.
- The **Source list** panel displays a list of workspace variables and lets you add them to the tunable parameters list.

To declare tunable parameters, you select one or more variables from the source list, add them to the **Global (tunable) parameters** list, and set their storage class and other attributes.

Source List Panel. The **Source list** panel displays a menu and a scrolling table of numerical workspace variables.

The menu lets you choose the source of the variables to be displayed in the list. Currently there is only one choice: **MATLAB workspace**. The source list displays names of the variables defined in the MATLAB base workspace.

Selecting one or more variables from the source list enables the **Add to table** button. Clicking **Add to table** adds selected variables to the tunable parameters list in the **Global (tunable) parameters** panel. This action is all that is necessary to declare tunable parameters.

In the source list, the names of variables that have been added to the tunable parameters list are displayed in italics (See Figure 3-10).

The **Refresh list** button updates the table of variables to reflect the current state of the workspace. If you define or remove variables in the workspace while the **Model Parameter Configuration** dialog is open, click the **Refresh list** button when you return to the dialog. The new variables are added to the source list.

Global (Tunable) Parameters Panel. The **Global (tunable) parameters** panel displays a scrolling table of variables that have been declared tunable in the current model, and lets you specify their attributes. The **Global (tunable) parameters** panel also lets you remove entries from the list, or create new tunable parameters.

You select individual variables and change their attributes directly in the table. The attributes are:

- **Storage class** of the parameter in the generated code. Select one of
  - `SimulinkGlobal(Auto)`
  - `ExportedGlobal`
  - `ImportedExtern`
  - `ImportedExternPointer`

  See "Storage Classes of Tunable Parameters" on page 3-55 for definitions.

- **Storage type qualifier** of the variable in the generated code. For variables with any storage class *except* `SimulinkGlobal(Auto)`, you can add a qualifier (such as `const` or `volatile`) to the generated storage declaration. To do so, you can select a predefined qualifier from the list, or add additional qualifiers to the list. Note that the code generator does not check the storage type qualifier for validity. The code generator includes the qualifier string in the generated code without syntax checking.

- **Name** of the parameter. This field is used only when creating a new tunable variable.

The **Remove** button deletes selected variables from the **Global (tunable) parameters** list.

The **New** button lets you create new tunable variables in the **Global (tunable) parameters** list. At a later time, you can add references to these variables in the model.

If the name you enter matches the name of an existing workspace variable in the **Source list**, that variable is declared tunable, and is displayed in italics in the **Source list**.

To define a new tunable variable, click the **New** button. This creates an empty entry in the table. Then, enter the name and attributes of the variable and click **Apply**.

Note If you edit the name of an existing variable in the list, you actually create a new tunable variable with the new name. The previous variable is removed from the list and loses its tunability (that is, it is inlined).

Declaring Tunable Variables

To declare an existing variable tunable:

1. Open the **Model Parameter Configuration** dialog.
2. In the **Source list** panel, click on the desired variable in the list to select it.
3. Click the **Add to table** button. The variable then appears in the table of tunable variables in the **Global (tunable) parameters** panel.
4. Click on the variable in the **Global (tunable) parameters** panel.
5. Select the desired storage class from the **Storage class** menu.
6. Optionally, select (or enter) a storage type qualifier.

7 Click **Apply**, or click **OK** to apply changes and close the dialog.

Tunable Expressions

The Real-time Workshop supports the use of tunable variables in expressions. An expression that contains one or more tunable parameters is called a *tunable expression*.

Currently, there are certain limitations on the use of tunable variables in expressions. When an expression described below as not supported is encountered during code generation, a warning is issued and a nontunable expression is generated in the code. The limitations on tunable expressions are:

- Complex expressions are not supported, except where the expression is simply the name of a complex variable.
- The use of certain operators or functions in expressions containing tunable operands is restricted. Restrictions are applied to four categories of operators or functions, classified in Table 3-4.

Table 3-4: Tunability Classification of Operators and Functions

Category	Operators or Functions
1	`+ - .* ./ < > <= >= == ~= & |`
2	`* /`
3	`abs, acos, asin, atan, atan2, boolean, ceil, cos, cosh, exp, floor, int8, int16, int32, log, log10, rem, sign, sin, sinh, sqrt, tan, tanh, uint8, uint16, uint32`
4	`: .^ ^ [] {} . \ .\ ' .' ; ,`

The rules applying to each category are as follows:

- Category 1 is unrestricted. These operators can be used in tunable expressions with any combination of scalar or vector operands.
- Category 2 operators can be used in tunable expressions where at least one operand is a scalar. That is, scalar/scalar and scalar/matrix operand combinations are supported, but not matrix/matrix.

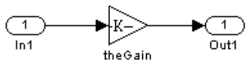
- Category 3 lists all functions that support tunable arguments. Tunable arguments passed to these functions retain their tunability. Tunable arguments passed to any other functions lose their tunability.
- Category 4 operators are not supported.

Note The “dot” (structure membership) operator is not supported. This means that expressions that include a structure member are not tunable.

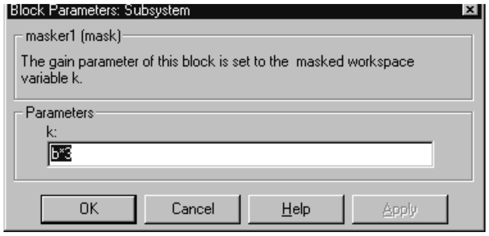
Tunable Expressions in Masked Subsystems

Tunable expressions are allowed in masked subsystems. You can use tunable parameter names or tunable expressions in a masked subsystem dialog. When referenced in lower-level subsystems, such parameters remain tunable.

As an example, consider the masked subsystem depicted below. The masked dialog variable `k` sets the gain parameter of `theGain`.



Suppose that the base workspace variable `b` is declared tunable with `SimulinkGlobal(Auto)` storage class. Figure 3-11 shows the tunable expression `b*3` in the subsystem’s mask dialog.


Block Parameters: Subsystem

masker1 (mask)

The gain parameter of this block is set to the masked workspace variable k.

Parameters

k:

b*3

Figure 3-11: Tunable Expression in Subsystem Mask Dialog

The Real-Time Workshop produces the following output computation for `theGain`.

```
/* Gain Block: <S1>/theGain */
rtb_temp0 *= (rtP.b * 3.0);
```

Note that `b` is represented as a member of the global parameters structure, `rtP`.

Limitations of Tunable Expressions in Masked Subsystems. Expressions that include variables that were declared or modified in mask initialization code are *not* tunable.

As an example, consider the subsystem above, modified as follows:

- The mask initialization code is

  ```
  t = 3 * k;
  ```

- The parameter `k` of the `myGain` block is `4 + t`.
- Workspace variable `b = 2`. The expression `b * 3` is plugged into the mask dialog as in Figure 3-11.

Since the mask initialization code can only run once, `k` is evaluated at code generation time as

```
4 + (3 * (2 * 3) )
```

The Real-Time Workshop inlines the result. Therefore, despite the fact that `b` was declared tunable, the code generator produces the following output computation for `theGain`.

```
/* Gain Block: <S1>/theGain */
rtb_temp0 *= (22.0);
```

Tunability of Linear Block Parameters

The following blocks have a `Realization` parameter that affects the tunability of their parameters:

- Transfer Fcn
- State-Space,
- Discrete Transfer Fcn,
- Discrete State-Space

- Discrete Filter

The `Realization` parameter must be set via the MATLAB `set_param` command, as in the following example.

```
set_param(gcb,'Realization','auto')
```

The following values are defined for the `Realization` parameter:

- `general`: The block's parameters are preserved in the generated code, permitting parameters to be tuned.
- `sparse`: The block's parameters are represented in the code by transformed values that increase the computational efficiency. Because of the transformation, the block's parameters are no longer tunable.
- `auto`: This setting is the default. A `general` realization is used if one or more of the block's parameters are tunable. Otherwise `sparse`, is used.

Note To tune the parameter values of a block of one of the above types without restriction during an external mode simulation, you must use set `Realization` to `general` .

Signals: Storage, Optimization, and Interfacing

The Real-Time Workshop offers a number of options that let you control how signals in your model are stored and represented in the generated code. This section discusses how you can use these options to:

- Control whether signal storage is declared in global memory space, or locally in functions (i.e., in stack variables).
- Control the allocation of stack space when using local storage.
- Ensure that particular signals are stored in unique memory locations by declaring them as *test points*.
- Reduce memory usage by instructing the Real-Time Workshop to store signals in reusable buffers.
- Control whether or not signals declared in generated code are interfaceable (visible) to externally written code. You can also specify that signals are to be stored in locations declared by externally written code.
- Preserve the symbolic names of signals in generated code by using signal labels.

The discussion in the following sections refers to code generated from `Signals_examp`, the model shown in Figure 3-12.

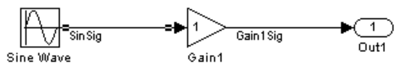


Figure 3-12: Signals_examp Model

Signal Storage Concepts

This section discusses structures and concepts you must understand in order to choose the best signal storage options for your application:

- The global block I/O data structure `rtB`
- The concept of signal *storage classes* as used in the Real-Time Workshop

rtB: the Global Block I/O Structure

By default, the Real-Time Workshop attempts to optimize memory usage by sharing signal memory and using local variables.

However, there are a number of circumstances in which it is desirable or necessary to place signals in global memory. For example:

- You may want a signal to be stored in a structure that is visible to externally written code.
- The number and/or size of signals in your model may exceed the stack space available for local variables.

In such cases, it is possible to override the default behavior and store selected (or all) signals in a model-specific *global block I/O data structure*. The global block I/O structure is called `rtB`.

The following code fragment illustrates how `rtB` is defined and declared in code generated (with signal storage optimizations off) from the `Signals_examp` model shown in Figure 3-12.

```
typedef struct BlockIO_tag {
  real_T SinSig;                        /* <Root>/Sine Wave */
  real_T Gain1Sig;                      /* <Root>/Gain1 */
} BlockIO;
.
.
.
/* Block I/O Structure */
BlockIO rtB;
```

Field names for signals stored in `rtB` are generated according to the rules described in “Symbolic Naming Conventions for Signals in Generated Code” on page 3-74.

Storage Classes for Signals

In the Real-Time Workshop, the *storage class* property of a signal specifies how the Real-Time Workshop declares and stores the signal. In some cases this specification is qualified by further options.

Note that in the context of the Real-Time Workshop, the term “storage class” is not synonymous with the term *storage class specifier*, as used in the C language.

Default Storage Class. `Auto` is the default storage class. `Auto` is the appropriate storage class for signals that you do not need to interface to external code. Signals with `Auto` storage class may be stored in local and/or shared variables, or in a global data structure. The form of storage depends on the **Signal storage reuse** and **Local block outputs** options, and on available stack space. See "Signals with Auto Storage Class" on page 3-68 for a full description of code generation options for signals with `Auto` storage class.

Explicitly Assigned Storage Classes. Signals with storage classes other than `Auto` are stored either as members of `rtB`, or in unstructured global variables, independent of `rtB`. These storage classes are appropriate for signals that you want to monitor and/or interface to external code.

The **Signal storage reuse** and **Local block outputs** optimizations do not apply to signals with storage classes other than `Auto`.

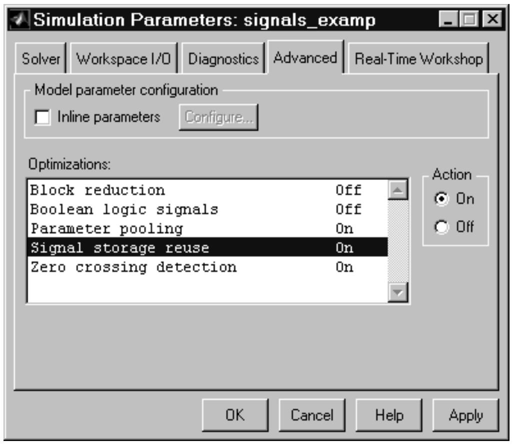
Use the **Signal Properties** dialog to assign these storage classes to signals:

- `SimulinkGlobal(Test Point)`: Test points are stored as fields of the `rtB` structure that are not shared or reused by any other signal. See "Declaring Test Points" on page 3-71 for further information.
- `ExportedGlobal`: The signal is stored in a global variable, independent of the `rtB` data structure. *model*`_export.h` exports the variable. Signals with `ExportedGlobal` storage class must have unique signal names. See "Interfacing Signals to External Code" on page 3-72 for further information.
- `ImportedExtern`: *model*`.h` declares the signal as an `extern` variable. Your code must supply the proper variable definition. Signals with `ImportedExtern` storage class must have unique signal names. See "Interfacing Signals to External Code" on page 3-72 for further information.
- `ImportedExternPointer`: *model*`.h` declares the signal as an `extern` pointer. Your code must supply the proper pointer variable definition. Signals with `ImportedExtern` storage class must have unique signal names. See "Interfacing Signals to External Code" on page 3-72 for further information.

Signals with Auto Storage Class

This section discusses options that are available for signals with `Auto` storage class. These options let you control signal memory reuse and choose local or global (`rtB`) storage for signals.

The **Signal storage reuse** option controls signal memory reuse. This option is on the Advanced page of the **Simulation Parameters** dialog box.

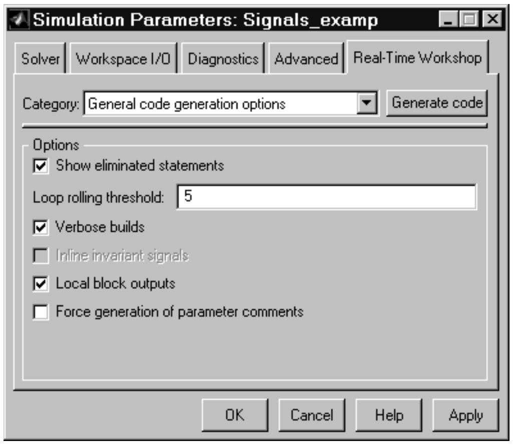


☐ Inline parameters

```
Block reduction          Off
Boolean logic signals    Off
Parameter pooling        On
Signal storage reuse     On
Zero crossing detection  On
```

Action: [✓ On] Off

The **Local block outputs** option determines whether signals are stored as members of `rtB`, or as local variables in functions. This option is in the **General code generation options** category of the Real-Time Workshop page.



Category: General code generation options

☑ Show eliminated statements

Loop rolling threshold: 5

☑ Verbose builds

☐ Inline invariant signals

☑ Local block outputs

☐ Force generation of parameter comments

By default, both **Signal storage reuse** and **Local block outputs** are on.

Note that these options interact. When the **Signal storage reuse** option is on:

- Signal memory is reused whenever possible.
- The **Local block outputs** option is enabled. This lets you choose whether reusable signal variables are declared as local variables in functions, or as members of `rtB`.

The following code examples illustrate the effects of the **Signal storage reuse** and **Local block outputs** options. The examples were generated from the `Signals_examp` model (see Figure 3-12).

The first example illustrates maximal signal storage optimization, with both **Signal storage reuse** and **Local block outputs** on (the default). The output signals from the Sine Wave and Gain blocks reuse `rtb_temp0`, a variable local to the `MdlOutputs` function.

```
/* local block i/o variables */
real_T rtb_temp0;
/* Sin Block: <Root>/Sine Wave */
rtb_temp0 = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) *
ssGetT(rtS) + (rtP.Sine_Wave_Phase));
/* Gain Block: <Root>/Gain1 */
rtb_temp0 *= (rtP.Gain1_Gain);
```

If you are constrained by limited stack space, you can turn **Local block outputs** off and still benefit from memory reuse. The following example was generated with **Local block outputs** off and **Signal storage reuse** on. The output signals from the Sine Wave and Gain blocks reuse `rtB.temp0`, a member of `rtB`.

```
rtB.temp0 = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) *
ssGetT(rtS) + (rtP.Sine_Wave_Phase));
/* Gain Block: <Root>/Gain1 */
rtB.temp0 *= (rtP.Gain1_Gain);
```

When the **Signal storage reuse** option is off, signal storage is not reused, and the **Local block outputs** option is disabled. This makes all block outputs global and unique, as in the following code fragment.

```
/* Sin Block: <Root>/Sine Wave */
rtB.SinSig = (rtP.Sine_Wave_Amp) *
sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));
rtB.Gain1Sig = rtB.SinSig * (rtP.Gain1_Gain);
```

In large models, disabling **Signal storage reuse** can significantly increase RAM and ROM usage. Therefore, this approach is not recommended.

Table 3-5 summarizes the possible combinations of the **Signal storage reuse** and **Local block outputs** options.

Table 3-5: Global, Local and Reusable Signal Storage Options

	Signal storage reuse ON	**Signal storage reuse OFF**
Local Block Outputs ON	Reuse signals in local memory (fully optimized)	N/A
Local Block Outputs OFF	Reuse signals in `rtB` structure	Individual signal storage in `rtB` structure

Controlling Stack Space Allocation

When the **Local block outputs** option is on, the use of stack space is constrained by the following TLC variables:

- `MaxStackSize`: the total allocation size of local variables that are declared by all functions in the entire model may not exceed `MaxStackSize` (in bytes). `MaxStackSize` can be any positive integer. If the total size of local variables exceeds this maximum, the Target Language Compiler will allocate the remaining variables in global, rather than local, memory.
- `MaxStackVariableSize`: limits the size of any local variable declared in a function to `N` bytes, where `N>0`. A variable whose size exceeds `MaxStackVariableSize` will be allocated in global, rather than local, memory.

You can change the values of these variables in your system target file if necessary. See “Assigning Target Language Compiler Variables” on page 3-93 for further information.

Declaring Test Points

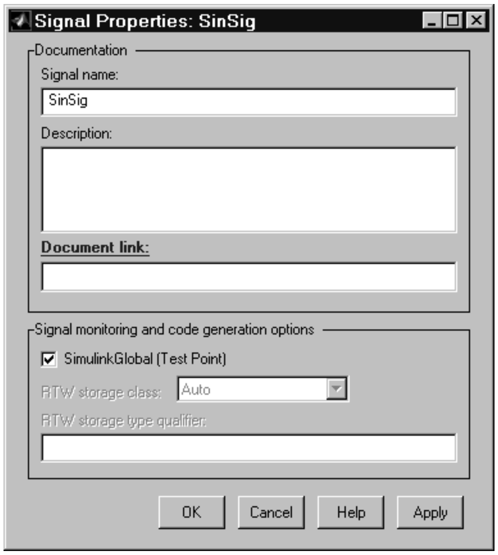
A *test point* is a signal that is stored in a unique location that is not shared or reused by any other signal. *Test-pointing* is the process of declaring a signal to be a test point.

Test points are stored as members of the `rtB` structure, even when the **Signal storage reuse** and **Local block outputs** option are selected. Test-pointing lets you override these options for individual signals. Therefore, you can test-point selected signals, without losing the benefits of optimized storage for the other signals in your model.

To declare a test point, use the Simulink **Signal Properties** dialog box as follows:

1 In your Simulink block diagram, select the line that carries the signal. Then select **Signal properties** from the **Edit** menu of your model. This opens the **Signal properties** dialog box.

  Alternatively, you can right-click the line that carries the signal, and select **Signal properties** from the pull-down menu.


Signal Properties: SinSig

Documentation

Signal name: SinSig

Description:

Document link:

Signal monitoring and code generation options

☑ SimulinkGlobal (Test Point)

RTW storage class: Auto

RTW storage type qualifier:

2 Check the **SimulinkGlobal (Test Point)** option.

3 Click **Apply**.

For an example of storage declarations and code generated for a test point, see Table 3-6, Signal Properties Options and Generated Code, on page 3-76.

Interfacing Signals to External Code

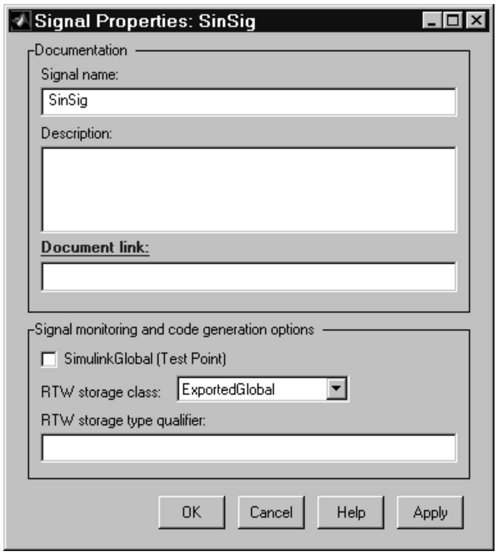
The Simulink **Signal Properties** dialog lets you interface selected signals to externally written code. In this way, your hand-written code has access to such signals for monitoring or other purposes. To interface a signal to external code, use the **Signal Properties** dialog box to assign one of the following storage classes to the signal:

- `ExportedGlobal`
- `ImportedExtern`
- `ImportedExternPointer`

Set the storage class as follows:

1 In your Simulink block diagram, select the line that carries the signal.Then select **Signal Properties** from the **Edit** menu of your model. This opens the **Signal Properties** dialog box.

Alternatively, you can right-click the line that carries the signal, and select **Signal properties** from the pull-down menu.


Signal Properties: SinSig
Documentation
Signal name:
SinSig
Description:
Document link:
Signal monitoring and code generation options
SimulinkGlobal (Test Point)
RTW storage class:
ExportedGlobal
RTW storage type qualifier:
OK
Cancel
Help
Apply

2 Deselect the **SimulinkGlobal (Test Point)** option if necessary. This enables the **RTW storage class** field.

3 Select the desired storage class (`ExportedGlobal`, `ImportedExtern`, or `ImportedExternPointer`) from the **RTW storage class** menu.

4 *Optional*: For storage classes other than Auto and SimulinkGlobal, you can enter a storage type qualifier such as const or volatile in the **RTW storage type qualifier** field. Note that the Real-Time Workshop does not check this string for errors; whatever you enter is included in the variable declaration.

5 Click **Apply**.

Note You can also interface test points and other signals that are stored as members of rtB to your code. To do this, your code must know the address of the rtB structure where the data is stored, and other information. This information is not automatically exported. The Real-Time Workshop provides C and Target Language Compiler APIs that give your code access to rtB and other data structures. See "Interfacing Parameters and Signals" on page 17-65 for further information.

Symbolic Naming Conventions for Signals in Generated Code

When signals have a storage class other than Auto, the Real-Time Workshop preserves symbolic information about the signals or their originating blocks in the generated code.

For labelled signals, field names in rtB derive from the signal names. In the following example, the field names rtB.SinSig and rtB.Gain1Sig derive from the corresponding labeled signals in the Signals_examp model (shown in Figure 3-12).

```
typedef struct BlockIO_tag {
  real_T SinSig;                        /* <Root>/Sine Wave */
  real_T Gain1Sig;                      /* <Root>/Gain1 */
} BlockIO;
```

For unlabeled signals, rtB field names derive from the name of the source block or subsystem. The naming format is

```
rtB.system#_BlockName_outport#
```

where system# is a unique system number assigned by Simulink, BlockName is the name of the source block, and outport# is a port number. The port number (`outport#`) is used only when the source block or subsystem has multiple output ports.

When a signal has `Auto` storage class, the Real-Time Workshop controls generation of variable or field names without regard to signal labels.

Summary of Signal Storage Class Options

Table 3-6 shows, for each signal storage class option, the variable declaration and the code generated for Sine Wave output (`SinSig`) of the model shown in Figure 3-12.

Table 3-6: Signal Properties Options and Generated Code

Storage Class	Declaration	Code
Auto (with storage optimizations on)	`real_T rtb_temp0;` (declared in *model*`_common.h`)	`rtb_temp0 = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`

Test point	`typedef struct BlockIO_tag {`
	`  real_T SinSig;`
	`  real_T Gain1Sig;`
	`} BlockIO;`
	`.`
	`.`
	`BlockIO rtB;`
		`rtB.SinSig = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`

Exported Global	`extern real_T SinSig;` (declared in *model*`_export.h`	`SinSig = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`
Imported Extern	`extern real_T SinSig;` (declared in *model*`_common.h`)	`SinSig = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`
Imported Extern Pointer	`extern real_T *SinSig;` (declared in *model*`_common.h`)	`*(SinSig) = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`

C API for Parameter Tuning and Signal Monitoring

The Real-Time Workshop includes support for development of a C application program interface (API) for tuning parameters and monitoring signals independent of external mode. See “Interfacing Parameters and Signals” in Chapter 17 for information.

Target Language Compiler API for Parameter Tuning and Signal Monitoring

The Real-Time Workshop includes support for development of a Target Language Compiler API for tuning parameters and monitoring signals independent of external mode. See “Interfacing Parameters and Signals” in Chapter 17 for information.

Parameter Tuning via MATLAB Commands

The **Model Parameter Configuration** dialog is the recommended way to see or set the attributes of tunable parameters. However, you can also use MATLAB `get_param` and `set_param` commands.

The following commands return the tunable parameters and/or their attributes:

- `get_param(gcs, 'TunableVars')`
- `get_param(gcs, 'TunableVarsStorageClass')`
- `get_param(gcs, 'TunableVarsTypeQualifier')`

The following commands declare tunable parameters or set their attributes:

- `set_param(gcs, 'TunableVars', str)`

  The argument `str` (string) is a comma-separated list of variable names.
- `set_param(gcs, 'TunableVarsStorageClass', str)`

  The argument `str` (string) is a comma-separated list of storage class settings.

  The valid storage class settings are:

  - `Auto`
  - `ExportedGlobal`
  - `ImportedExtern`
  - `ImportedExternPointer`

- `set_param(gcs, 'TunableVarsTypeQualifier', str)`

  The argument `str` (string) is a comma-separated list of storage type qualifiers.

The following example declares the variable `k1` to be tunable, with storage class `ExportedGlobal` and type qualifier `const`.

```
set_param(gcs, 'TunableVars', 'k1')
set_param(gcs, 'ExportedGlobal')
set_param(gcs, 'TunableVarsTypeQualifier','const')
```

Simulink Data Objects and Code Generation

Prerequisites

Before using Simulink data objects with the Real-Time Workshop, please read the following:

- The discussion of Simulink data objects in *Using Simulink*
- "Parameters: Storage, Interfacing, and Tuning" on page 3-51
- "Signals: Storage, Optimization, and Interfacing" on page 3-65

Overview

Within the class hierarchy of Simulink data objects, Simulink provides two classes that are designed as base classes for signal and parameter storage. These are:

- `Simulink.Parameter`: Objects that are instances of the `Simulink.Parameter` class or any class derived from `Simulink.Parameter` are called *parameter objects*.
- `Simulink.Signal`: Objects that are instances of the `Simulink.Signal` class or any class derived from `Simulink.Signal` are called *signal objects*.

The `RTWInfo` properties of parameter and signal objects are used by the Real-Time Workshop during code generation. These properties let you assign storage classes and storage type qualifiers to the objects, thereby controlling how the generated code stores and represents signals and parameters.

The Real-Time Workshop also writes information about the properties of parameter and signal objects to the *model*`.rtw` file. This information, formatted as `ObjectProperties` records, is accessible to Target Language Compiler programs. For general information on `ObjectProperties` records, see "Object Property Information in the model.rtw File" on page 3-88.

The general procedure for using Simulink data objects in code generation is as follows:

1. Define a subclass of one of the built-in `Simulink.Data` classes.
   - For parameters, define a subclass of `Simulink.Parameter`.
   - For signals, define a subclass of `Simulink.Signal`.

2. Instantiate parameter or signal objects from your subclass and set their properties appropriately, using the Simulink Explorer.
3. Use the objects as parameters or signals within your model.
4. Generate code and build your target executable.

The following sections describe the relationship between Simulink data objects and code generation in the Real-Time Workshop.

Parameter Objects

This section discusses how to use parameter objects in code generation.

Configuring Parameter Objects for Code Generation

In configuring parameter objects for code generation, you use the following code generation and parameter object properties:

- The **Inline parameters** option (see “Parameters: Storage, Interfacing, and Tuning” on page 3-51).
- Parameter object properties:
  - `Value`. This property is the numeric value of the object, used as an initial (or inlined) parameter value in generated code.
  - `RTWInfo.StorageClass`. This property controls the generated storage declaration and code for the parameter object.
  - `RTWInfo.TypeQualifier`. This property is a string included as a prefix in the generated storage declaration.

  Other parameter object properties (such as user-defined properties of classes derived from `Simulink.Parameter`) do not affect code generation.

Note If **Inline parameters** is off (the default), the `RTWInfo.StorageClass` and `RTWInfo.TypeQualifier` parameter object properties are ignored in code generation.

Effect of Storage Classes on Code Generation for Parameter Objects

The Real-Time Workshop generates code and storage declarations based on the `RTWInfo.StorageClass` property of the parameter object. The logic is as follows:

- If the storage class is `'Auto'` (the default), the parameter object is inlined (if possible), using the `Value` property.
- For storage classes other than `'Auto'`, the parameter object is handled as a tunable parameter.
  - A global storage declaration is generated. You can use the generated storage declaration to make the variable visible to your hand-written code.

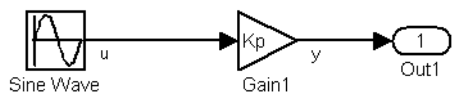
You can also make variables declared in your hand-written code visible to the generated code.

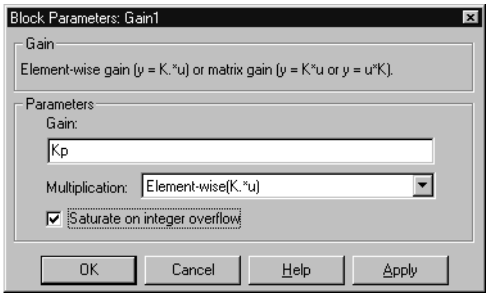
- The symbolic name of the parameter object is preserved in the generated code.

See Table 3-7 for examples of code generated for each possible setting of `RTWInfo.StorageClass`.

Example of Parameter Object Code Generation

In this section, we use the Gain block computations of the model shown in Figure 3-13 as an example of how the Real-Time Workshop generates code for a parameter object.




Block Parameters: Gain1

Gain

Element-wise gain (y = K.*u) or matrix gain (y = K*u or y = u*K).

Parameters

Gain:

Kp

Multiplication: Element-wise(K.*u)

[✓] Saturate on integer overflow

Figure 3-13: Model Using Parameter Object Kp As Block Parameter

In this model, `Kp` sets the gain of the `Gain1` block.

To configure a parameter object such as `Kp` for code generation:

1 Define a subclass of `Simulink.Parameter`. In this example, the parameter object is an instance of the example class `UserDefined.Parameter`, which is

provided with Simulink. For the definition of `UserDefined.Parameter`, see the directory *matlabroot*`/toolbox/simulink/simdemos/@UserDefined`.

2 Instantiate a parameter object from your subclass. The following example instantiates `Kp` as a parameter object of class `UserDefined.Parameter`.

```
Kp = UserDefined.Parameter;
```

Make sure that the name of the parameter object matches the desired block parameter in your model. This ensures that Simulink can associate the parameter name with the correct object. For example, in the model of Figure 3-13, the Gain block parameter `Kp` resolves to the parameter object `Kp`.

3 Set the object properties.

- Set the `Value` property, for example:

  ```
  Kp.Value = 5.0;
  ```

- Set the `RTWInfo.StorageClass` property, for example:

  ```
  Kp.RTWInfo.StorageClass = 'ExportedGlobal';
  ```

- *Optional*: if the `RTWInfo.StorageClass` property is not `Auto`, you can assign a storage type qualifier to the `RTWInfo.TypeQualifier` property, for example:

  ```
  Kp.RTWInfo.StorageClass = 'const';
  ```

Table 3-7 shows the variable declarations for `Kp` and the code generated for the Gain block in the model shown in Figure 3-13, with **Inline parameters** on. An example is shown for each possible setting of `RTWInfo.StorageClass`.

Table 3-7: Code Generation from Parameter Objects (Inline Parameters ON)

StorageClass Property	Generated Variable Declaration and Code
Auto	`rtB.y = rtB.u * (5.0);`

Simulink Global	`typedef struct Parameters_tag {`
	`  real_T Kp;`
	`.`
	`.`
	`Parameters rtP = {`
	`  5.0`
	`};`
	`.`
	`.`
	`rtB.y = rtB.u * (rtP.Kp);`

Exported Global	`real_T Kp = 5.0;`
	`.`
	`.`
	`rtB.y = rtB.u * (Kp);`

Imported Extern	` extern real_T Kp;`
	`.`
	`.`
	`rtB.y = rtB.u * (Kp);`

Imported Extern Pointer	`extern real_T *Kp;`
	`.`
	`.`
	`rtB.y = rtB.u * ((*Kp));`

Signal Objects

This section discusses how to use signal objects in code generation.

Configuring Signal Objects for Code Generation

In configuring signal objects for code generation, you use the following code generation options and signal object properties:

- The **Signal storage reuse** code generation option (see “Signals: Storage, Optimization, and Interfacing” on page 3-65).
- The **Local block outputs** code generation option (see “Signals: Storage, Optimization, and Interfacing” on page 3-65).
- Signal object properties:
  - `RTWInfo.StorageClass`. The storage classes defined for signal objects, and their effect on code generation, are the same for model signals and signal objects (see “Storage Classes for Signals” on page 3–66).
  - `RTWInfo.TypeQualifier`. This property is a storage type qualifier. The string is included as a prefix in the generated storage declaration. The syntax of the string is not checked for validity.

  Other signal object properties (such as user-defined properties of classes derived from `Simulink.Signal`) do not affect code generation.

Effect of Storage Classes on Code Generation for Signal Objects

The way in which the Real-Time Workshop uses storage classes to determine how signals are stored is the same with and without signal objects. However, if a signal’s label resolves to a signal object, the object’s `RTWInfo.StorageClass` property is used in place of the port configuration of the signal.

The default storage class is `Auto`. If the storage type is `Auto`, the Real-Time Workshop follows the **Signal storage reuse** and **Local block outputs** code generation options to determine whether signal objects are stored in reusable and/or local variables. Make sure that these options are set correctly for your application.

To generate a a test point or externally-interfaceable signal storage declaration, use an explicit `RTWInfo.StorageClass` assignment. For example, setting the storage class to `SimulinkGlobal`, as in the following command, is equivalent to declaring a signal as a test point.

```
SinSig.RTWInfo.StorageClass = 'SimulinkGlobal';
```

Example of Signal Object Code Generation

The discussion and code examples in this section refers to the model shown in Figure 3-14.

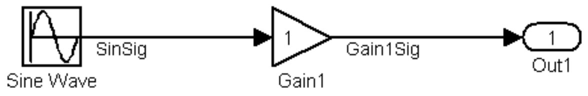


Figure 3-14: Example Model With Signal Object

To configure a signal object, you must first create it and associate it with a labelled signal in your model. To do this:

1 Define a subclass of `Simulink.Signal`. In this example, the signal object is an instance of the example class `UserDefined.Signal`, which is provided with Simulink. For the definition of `UserDefined.Signal`, see the directory *matlabroot*`/toolbox/simulink/simdemos/@UserDefined`.

2 Instantiate a signal object from your subclass. The following example instantiates `SinSig`, a signal object of class `UserDefined.Signal`.

```
SinSig = UserDefined.Signal;
```

Make sure that the name of the signal object matches the label of the desired signal in your model. This ensures that Simulink can resolve the signal label to the correct object. For example, in the model shown in Figure 3-14, the signal label `SinSig` would resolve to the signal object `SinSig`.

3 Set the object properties as required:

- Assign the signal object's storage class by setting the `RTWInfo.StorageClass` property, for example,

```
SinSig.RTWInfo.StorageClass = 'ExportedGlobal';
```

- *Optional*: if the `RTWInfo.StorageClass` property is not `Auto`, you can assign a storage type qualifier to the `RTWInfo.TypeQualifier` property, for example,

```
SinSig.RTWInfo.StorageClass = 'const';
```

Table 3-8 shows, for each setting of RTWInfo.StorageClass, the variable declaration and the code generated for Sine Wave output (SinSig) of the model shown in Figure 3-14.

Table 3-8: Signal Properties Options and Generated Code

Storage Class	Declaration	Code
Auto (with storage optimizations on)	`real_T rtb_temp0;` (declared in *model*`_common.h`)	`rtb_temp0 = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`

Simulink Global	`typedef struct BlockIO_tag {`
	`  real_T SinSig;`
	`  real_T Gain1Sig;`
	`} BlockIO;`
	`.`
	`.`
	`BlockIO rtB;`
		`rtB.SinSig = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`

Exported Global	`extern real_T SinSig;` (declared in *model*`_export.h`	`SinSig = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`
Imported Extern	`extern real_T SinSig;` (declared in *model*`_common.h`)	`SinSig = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`
Imported Extern Pointer	`extern real_T *SinSig;` (declared in *model*`_common.h`)	`*(SinSig) = (rtP.Sine_Wave_Amp) * sin((rtP.Sine_Wave_Freq) * ssGetT(rtS) + (rtP.Sine_Wave_Phase));`

Object Property Information in the model.rtw File

During code generation, the Real-Time Workshop writes property information about signal and parameter objects to the *model*`.rtw` file. An `ObjectProperties` record is written for each parameter or signal that meets certain conditions. These conditions are described in “ObjectProperties Records For Parameters” on page 3-88 and “ObjectProperties Records For Signals” on page 3-89.

The `ObjectProperties` records contain all of the property information from the associated object. To access `ObjectProperties` records, you must write Target Language Compiler code (see “Accessing Object Property Information via TLC” on page 3-90).

ObjectProperties Records For Parameters

An `ObjectProperties` record is included in the in the `ModelParameters` section of *model*`.rtw` file for each parameter, under the following conditions:

1 **Inline parameters** is on.

2 The parameter resolves to a `Simulink.Parameter` object (or to a parameter object that comes from a class derived from the `Simulink.Parameter` class).

3 The parameter’s `RTWInfo.StorageClass` is set to anything but `'Auto'`.

The following is an example of an `ObjectProperties` record for a parameter.

```
ModelParameters {
  ...
  Parameter {
    Identifier           Kp
    Tunable              yes
    ...
    Value                [5.0]
    Dimensions           [1, 1]
    ObjectProperties {
        RTWInfo {
        StorageClass "SimulinkGlobal"
        TypeQualifier ""
      }
      Value 5.0
      ...
    }
  }
}
```

ObjectProperties Records For Signals

An `ObjectProperties` record is included in the `BlockOutputs` section of the *`model`*`.rtw` file for each signal which meets the following conditions:

1 The signal resolves to a `Simulink.Signal` object (or to an object that comes from a class derived from the `Simulink.Signal` class).

2 The signal's symbol is preserved in the generated code. The symbol is preserved if:

   - The signal's `RTWInfo.StorageClass` should be set to anything but `'Auto'`.
   - The signal label must be a valid variable name.
   - The signal label must be unique throughout the model.

Note If the signal is configured to be an unstructured global variable in the generated code, its validity and uniqueness are enforced and its symbol is always preserved.

The following is an example of an `ObjectProperties` record for a signal.

```
BlockOutputs {
  ...
    BlockOutput {
      Identifier               SinSig
      ...
      SigLabel "SinSig"
      ObjectProperties {
        RTWInfo {
           StorageClass "SimulinkGlobal"
          TypeQualifier ""
        }
      ...
    }
  }
}
```

Accessing Object Property Information via TLC

This section provides sample code to illustrate how to access object property information from the *model*`.rtw` file using TLC code. For more information on TLC and the *model*`.rtw` file, see the *Target Language Compiler Reference Guide*.

Accessing Parameter Object Property Records. The following code fragment iterates over the ModelParameters section of the *model*.rtw file and extracts information from any parameter ObjectProperties records encountered.

```
%with CompiledModel.ModelParameters
  %foreach modelParamIdx = NumParameters
    %assign thisModelParam = ModelParameter[modelParamIdx]
    %assign paramName = thisModelParam.Identifier
    %if EXISTS("thisModelParam.ObjectProperties")
      %with thisModelParam.ObjectProperties
        %assign valueInObject = Value
        %with RTWInfo
          %assign storageClassInObject  = StorageClass
          %assign typeQualifierInObject = TypeQualifier
        %endwith
        %% **********************************
        %% Access user-defined properties here
        %% **********************************
        %if EXISTS("MY_PROPERTY_NAME")
          %assign userDefinedPropertyName = MY_PROPERTY_NAME
        %endif
        %% **********************************
      %endwith
    %endif
  %endforeach
%endwith
```

Accessing Signal Object Property Records. The following code fragment iterates over the `BlockOutputs` section of the *model*`.rtw` file and extracts information from any signal `ObjectProperties` records encountered.

```
%with CompiledModel.BlockOutputs
  %foreach blockOutputIdx = NumBlockOutputs
    %assign thisBlockOutput = BlockOutput[blockOutputIdx]
    %assign signalName = thisBlockOutput.Identifier
    %if EXISTS("thisBlockOutput.ObjectProperties")
      %with thisBlockOutput.ObjectProperties
        %with RTWInfo
          %assign storageClassInObject  = StorageClass
          %assign typeQualifierInObject = TypeQualifier
        %endwith \
        %% ***********************************\
        %% Access user-defined properties here\
        %% ***********************************
        %if EXISTS("MY_PROPERTY_NAME")
          %assign userDefinedPropertyName = MY_PROPERTY_NAME
        %endif
        %% ***********************************
      %endwith
    %endif
  %endforeach
%endwith
```

Using Object Properties to Export ASAP2 Files

The ASAM-ASAP2 Data Definition Target provides special signal and parameter subclasses that support exporting of signal and parameter object property information to ASAP2 data files. For information about the ASAP2 target and its associated classes and TLC files, see “Generating ASAP2 Files” in the Real-Time Workshop online documentation.

Configuring the Generated Code via TLC

This section covers features of the Real-Time Workshop Target Language Compiler that help you to fine-tune your generated code. To learn more about TLC, read the *Target Language Compiler Reference Guide*.

Target Language Compiler Variables and Options

The Target Language Compiler supports extended code generation variables and options in addition to those included in the code generation options categories of the Real-Time Workshop page.

There are two ways to set TLC variables and options:

- Assigning TLC variables in the system target file
- Entering TLC options or variables into the **System Target File** field on the Real-Time Workshop page

Assigning Target Language Compiler Variables

The `%assign` statement lets you assign a value to a TLC variable, as in

```
%assign MaxStackSize = 4096
```

This is also known as creating a *parameter name/parameter value pair*.

The `%assign` statement is described in the *Target Language Compiler Reference Guide*. It is recommended that you write your `%assign` statements in the `Configure RTW code generation settings` section of the system target file.

The following table lists the code generation variables you can set with the `%assign` statement.

Table 3-9: Target Language Compiler Optional Variables

Variable	Description
`MaxStackSize=`*`N`*	When **Local block outputs** is enabled, the total allocation size of local variables that are declared by all functions in the entire model may not exceed `MaxStackSize` (in bytes). `N` can be any positive integer.
`MaxStackVariableSize=`*`N`*	When **Local block outputs** is enabled, this limits the size of any local variable declared in a function to `N` bytes, where `N>0`. A variable whose size exceeds `MaxStackVariableSize` will be allocated in global, rather than local, memory
`FunctionInlineType=` `"`*`mode`*`"`	Controls how functions are inlined. There are two modes: • `CodeInsertion` • `PragmaInline` Using `CodeInsertion`, the code is actually inserted where the function call would have been made. `PragmaInline` directs the Target Language Compiler to declare the function when the appropriate compiler directive occurs.
`PragmaInlineString=` *`"string"`*	If `FunctionInlineType` is set to `PragmaInline`, this should be set to the directive that your compiler uses for inlining a function (for example, for Microsoft Visual C/C++, `"__inline"`).

Table 3-9: Target Language Compiler Optional Variables (Continued)

Variable	Description
`WarnNonSaturatedBlocks=`*`value`*	Flag to control display of overflow warnings for blocks that have saturation capability, but have it turned off (unchecked) in their dialog. These are the options: • `0` — no warning is displayed • `1` — displays one warning for the model during code generation • `2` — displays one warning that contains a list of all offending blocks
`BlockIOSignals=`*`value`*	Supports monitoring signals in a running model. See “Signal Monitoring via Block Outputs” on page 17-65. Setting the variable causes the *`model`*`_bio.c` file to be generated. These are the options: • `0` — deactivates this feature • `1` — creates *`model`*`_bio.c`
`ParameterTuning=`*`value`*	Setting the variable to `1` causes a parameter tuning file (*`model`*`_pt.c`) to be generated. *`model`*`_pt.c` contains data structures that enable a running program to access model parameters independent of external mode. See “Parameter Tuning via model_pt.c” on page 17-71.

Setting Target Language Compiler Options

You can enter TLC options directly into the **System target file** field in the **Target configuration** category of the Real-Time Workshop page, by appending the options and arguments to the system target filename. This is equivalent to invoking the Target Language Compiler with options on the MATLAB command line.

The common options are shown in the table below.

Table 3-10: Target Language Compiler Options

Option	Description
`–I`*path*	Adds *path* to the list of paths in which to search for target files (`.tlc` files).
`–m[`*N*`|a]`	Maximum number of errors to report when an error is encountered (default is 5). For example, `–m3` specifies that at most three errors will be reported. To report all errors, specify `–ma`.
`–d[g|n|o]`	Specifies debug mode (`generate`, `normal`, or `off`). Default is `off`. When `–dg` is specified, a `.log` file is create for each of your TLC files. When debug mode is enabled (i.e., `generate` or `normal`), the Target Language Compiler displays the number of times each line in a target file is encountered.
`–a`*Variable*`=`*val*	Equivalent to the TLC statement `%assign Variable = val` Note: It is recommended that you use `%assign` statements in the TLC files, rather than the `-a` option.

Making an Executable

After completing code generation, the build process determines whether or not to continue and compile and link an executable program. This decision is governed by the following parameters:

- **Generate code only** option

  When this option is selected, the build process always omits the make phase.
- Makefile-only target

  The Visual C/C++ Project Makefile versions of the `grt`, `grt_malloc`, and Real-Time Workshop Embedded Coder target configurations generate a Visual C/C++ project makefile (*model*`.mak`). To build an executable, you must open *model*`.mak` in the Visual C/C++ IDE and compile and link the model code.
- `HOST` template makefile variable

  The template makefile variable `HOST` identifies the type of system upon which your executable is intended to run. The `HOST` variable can take on one of three possible values: `PC`, `UNIX`, or `ANY`.

  By default, `HOST` is set to `UNIX` in template makefiles designed for use with UNIX (such as `grt_unix.tmf`), and to `PC` in the template makefiles designed for use with development systems for the PC (such as `grt_vc.tmf`).

  If Simulink is running on the same type of system as that specified by the `HOST` variable, then the executable is built. Otherwise:

  - If `HOST = ANY`, an executable is still built. This option is useful when you want to cross-compile a program for a system other than the one Simulink is running on.
  - Otherwise, processing stops after generating the model code and the makefile; the following message is displayed on the MATLAB command line.

  ```
  ### Make will not be invoked - template makefile is for a different host
  ```

Directories Used in the Build Process

The Real-Time Workshop creates output files in two directories during the build process:

- The working directory

  If an executable is created, it is written to your working directory. The executable is named *model*`.exe` (on PC) or *model* (on UNIX).
- The build directory

  The build process creates a subdirectory, called the build directory, within your working directory. The build directory name is *model*`_`*target*`_rtw`, where *model* is the name of the source model and *target* is the name of the chosen target. The build directory stores generated source code and all other files created during the build process (except the executable).

The build directory always contains the generated code modules *model*`.c`, *model*`.h`, and *model*`_export.h`, and the generated makefile *model*`.mk`.

Depending upon the target and code generation and build options selected, additional files in the build directory may include:

- *model*`.rtw`
- Object (`.obj`) files
- Code modules generated from subsystems
- TLC profiler report files
- Block I/O (*model*`_bio.c`) and parameter tuning (*model*`_pt.c`) information files
- Real-Time Workshop project (*model*`.tmw`) files

Choosing and Configuring Your Compiler

The Real-Time Workshop build process depends upon the correct installation of one or more supported compilers. Note that *compiler*, in this context, refers to a development environment containing a linker and make utility, in addition to a high-level language compiler.

The build process also requires the selection of a template makefile. The template makefile determines which compiler will be run, during the make phase of the build, to compile the generated code.

This section discusses how to install a compiler and choose an appropriate template makefile, on both Windows and UNIX systems.

Choosing and Configuring Your Compiler on Windows

On Windows, you must install one or more supported compilers, In addition, you must define an environment variable associated with each compiler.Make sure that your compiler is installed as described in“Third-Party Compiler Installation on Windows” on page -xxii.

You can select a template makefile that is specific to your compiler. For example, `grt_bc.tmf` designates the Borland C/C++ compiler, and `grt_vc.tmf` designates the Visual C/C++ compiler.

Alternatively, you can choose a default template makefile that will select the default compiler for your system. The default compiler is the compiler MATLAB uses to build MEX-files. You can set up the default compiler by using the MATLAB `mex` command as shown below.

```
mex –setup
```

See the *MATLAB Application Program Interface Guide* for information on the `mex` command.

Default template makefiles are named *target*`_default_tmf`. For example, the default template makefile for the generic real-time (GRT) target is `grt_default_tmf`.

The build process uses the following logic to locate a compiler for the generated code:

1 If a specific compiler is named in the template makefile, the build process uses that compiler.

2 If the template makefile designates a default compiler (as in `grt_default_tmf`), the build process uses the same compiler as those used for building C MEX-files.

3 If no default compiler is established, the build process examines environment variables which define the path to installed compilers, and selects the first compiler located. The variables are searched in the following order:

   - `MSDevDir` or `DEVSTUDIO` (defining the path to the Microsoft Visual C/C++)
   - `WATCOM` (defining the path to the Watcom C/C++ compiler)
   - `BORLAND` (defining the path to the Borland C/C++ compiler)

4 If none of the above environment variables is defined, the build process selects the `lcc` compiler, which is shipped and installed with MATLAB.

Compile/Build Options for Visual C/C++. The Real-Time Workshop offers two sets of template makefiles designed for use with Visual C/C++.

To compile under Visual C/C++ and build an executable within the Real-Time Workshop build process, use one of the *target*`_vc.tmf` template makefiles:

- `ert_vc.tmf`
- `grt_malloc_vc.tmf`
- `grt_vc.tmf`
- `rsim_vc.tmf`

Alternatively, you can choose to create a project makefile (*model*`.mak`) suitable for use with the Visual C/C++ IDE. In this case, you must compile and link your code within the Visual C/C++ environment. To create a Visual C/C++ project makefile, choose one of the Visual C/C++ Project Makefile versions of the `grt`, `ert`, or `grt_malloc` target configurations. These configurations use the *target*`_msvc.tmf` template makefiles:

- `ert_msvc.tmf`
- `grt_malloc_msvc.tmf`
- `grt_msvc.tmf`

Choosing and Configuring Your Compiler On UNIX

On UNIX, the Real-Time Workshop build process uses the default compiler. `cc` is the default on all platforms except SunOS, where `gcc` is the default.

You should choose the UNIX-specific template makefile that is appropriate to your target. For example, `grt_unix.tmf` is the correct template makefile to build a generic real-time program under UNIX.

Available Compiler/Makefile/Target Configurations

To determine which template makefiles are appropriate for your compiler and target, see the table “Targets Available from the System Target File Browser” on page 3-36.

Template Makefiles and Make Options

The Real-Time Workshop includes a set of built-in template makefiles that are designed to build programs for specific targets.

There are two types of template makefiles:

- *Compiler-specific* template makefiles are designed for use with a particular compiler or development system.

  By convention, compiler-specific template makefiles are named according to the target and compiler (or development system). For example, `grt_vc.tmf` is the template makefile for building a generic real-time program under Visual C/C++; `ert_lcc.tmf` is the template makefile for building a Real-Time Workshop Embedded Coder program under the LCC compiler.
- *Default* template makefiles make your model designs more portable, by choosing the correct compiler-specific makefile and compiler for your installation. "Choosing and Configuring Your Compiler" on page 3-99 describes the operation of default template makefiles in detail.

  Default template makefiles are named *target*`_default_tmf`. For example, `grt_default_tmf` is the default template makefile for building a generic real-time program; `ert_default_tmf` is the default template makefile building a Real-Time Workshop Embedded Coder program.

You can supply options to makefiles via arguments to the **Make command** field of the **Target configuration** category of the Real-Time Workshop page. Append the arguments after `make_rtw` (or `make_xpc` or other `make` command), as in the following example.

```
make_rtw OPTS="-DMYDEFINE=1"
```

The syntax for `make` command options differs slightly for different compilers.

Compiler-Specific Template Makefiles

This section documents the available compiler-specific template makefiles and common options you can use with each.

Template Makefiles for UNIX

- `ert_unix.tmf`

- `grt_malloc_unix.tmf`
- `grt_unix.tmf`
- `rsim_unix.tmf`
- `rtwsfcn_unix.tmf`

The template makefiles for UNIX platforms are designed to be used with GNU Make. These makefile are set up to conform to the guidelines specified in the IEEE Std 1003.2-1992 (POSIX) standard.

You can supply options via arguments to the `make` command.

- `OPTS` — User-specific options, for example,

  ```
  make_rtw OPTS="-DMYDEFINE=1"
  ```

- `OPT_OPTS` — Optimization options. The default optimization option is `-O`. To turn off optimization and add debugging symbols, specify the `-g` compiler switch in the `make` command, for example,

  ```
  make_rtw OPT_OPTS="-g"
  ```

For additional options, see the comments at the head of each template makefile.

Template Makefiles for Visual C/C++

The Real-Time Workshop offers two sets of template makefiles designed for use with Visual C/C++.

To build an executable within the Real-Time Workshop build process, use one of the *target*`_vc.tmf` template makefiles:

- `ert_vc.tmf`
- `grt_malloc_vc.tmf`
- `grt_vc.tmf`
- `rsim_vc.tmf`
- `rtwsfcn_vc.tmf`

You can supply options via arguments to the `make` command.

- `OPTS` — User-specific options, for example,

  ```
  make_rtw OPTS="-DMYDEFINE=1"
  ```

- `OPT_OPTS` — Optimization options. The default optimization option is `-Ot`. To turn off optimization and add debugging symbols, specify the `-Zd` compiler switch in the `make` command.

  ```
  make_rtw OPT_OPTS="-Zd"
  ```

For additional options, see the comments at the head of each template makefile.

To create a Visual C/C++ project makefile (*model*`.mak`) without building an executable, use one of the *target*`_msvc.tmf` template makefiles:

- `ert_msvc.tmf`
- `grt_malloc_msvc.tmf`
- `grt_msvc.tmf`

These template makefiles are designed to be used with `nmake`, which is bundled with Visual C/C++.

You can supply the following options via arguments to the `nmake` command:

- `OPTS` — User-specific options, for example,

  ```
  make_rtw OPTS="/D MYDEFINE=1"
  ```

For additional options, see the comments at the head of each template makefile.

Template Makefiles for Watcom C/C++

Note As of this printing, the Watcom C compiler is no longer available from the manufacturer. The Real-Time Workshop continues to ship Watcom-related template makefiles at this time. However, this policy may be subject to change in the future.

- `drt_watc.tmf`
- `ert_watc.tmf`
- `grt_malloc_watc.tmf`
- `grt_watc.tmf`
- `rsim_watc.tmf`

- `rtwsfcn_watc.tmf`
- `win_watc.tmf`

The Real-Time Workshop provides template makefiles to create an executable for Windows 95, Windows 98, and Windows NT using Watcom C/C++. These template makefiles are designed to be used with `wmake`, which is bundled with Watcom C/C++.

You can supply options via arguments to the `make` command. Note that the location of the quotes is different from the other compilers and make utilities discussed in this chapter:

- `OPTS` — User specific options, for example,

  ```
  make_rtw "OPTS=-DMYDEFINE=1"
  ```

- `OPT_OPTS` — Optimization options. The default optimization option is `-oxat`. To turn off optimization and add debugging symbols, specify the `-d2` compiler switch in the `make` command, for example,

  ```
  make_rtw "OPT_OPTS=-d2"
  ```

For additional options, see the comments at the head of each template makefile.

Template Makefiles for Borland C/C++

- `ert_bc.tmf`
- `grt_bc.tmf`
- `grt_malloc_bc.tmf`
- `rsim_bc.tmf`
- `rtwsfcn_bc.tmf`

The Real-Time Workshop provides template makefiles to create an executable for Windows 95, Windows 98, and Windows NT using Borland C/C++.

You can supply these options via arguments to the `make` command:

- `OPTS` — User-specific options, for example,

  ```
  make_rtw OPTS="-DMYDEFINE=1"
  ```

- `OPT_OPTS` — Optimization options. Default is none. To turn off optimization and add debugging symbols, specify the `-v` compiler switch in the `make` command.

  ```
  make_rtw OPT_OPTS="-v"
  ```

For additional options, see the comments at the head of each template makefile.

Template Makefiles for LCC

- `ert_lcc.tmf`
- `grt_lcc.tmf`
- `grt_malloc_lcc.tmf`
- `rsim_lcc.tmf`
- `rtwsfcn_lcc.tmf`

The Real-Time Workshop provides template makefiles to create an executable for Windows 95, Windows 98, and Windows NT using LCC compiler Version 2.4 and GNU Make (`gmake`).

You can supply options via arguments to the `make` command:

- `OPTS` — User-specific options, for example,

  ```
  make_rtw OPTS="-DMYDEFINE=1"
  ```

- `OPT_OPTS` — Optimization options. Default is none. To enable debugging, specify `-g4` in the `make` command:

  ```
  make_rtw OPT_OPTS="-g4"
  ```

For additional options, see the comments at the head of each template makefile.

Template Makefile Structure

The detailed structure of template makefiles is documented in “Template Makefiles” on page 17-25. This information is provided for those who want to customize template makefiles.

Generated Code Formats

Introduction

The Real-Time Workshop provides five different *code formats*. Each code format specifies a framework for code generation suited for specific applications.

The five code formats and corresponding application areas are:

- Real-time: Rapid prototyping
- Real-time `malloc`: Rapid prototyping
- S-function: Creating proprietary S-function `.dll` or MEX-file objects, code reuse, and speeding up your simulation
- Embedded C: Deeply embedded systems
- Ada

Note Generation of Ada code requires the Real-Time Workshop Ada Coder, a separate product. See Chapter 16, "Real-Time Workshop Ada Coder" for more information.

This chapter discusses the relationship of code formats to the available target configurations, and factors you should consider when choosing a code format and target. This chapter also summarizes the real-time, real-time `malloc`, S-function, and embedded C code formats.

Choosing a Code Format for Your Application

Your choice of code format is the most important code generation option. The code format specifies the overall framework of the generated code and determines its style.

When you choose a target, you implicitly choose a code format. Typically, the system target file will specify the code format by assigning the TLC variable `CodeFormat`. The following example is from `ert.tlc`.

```
%assign CodeFormat = "Embedded-C"
```

If the system target file does not assign `CodeFormat`, the default is `RealTime` (as in `grt.tlc`).

If you are developing a custom target, you must consider which code format is best for your application and assign `CodeFormat` accordingly.

Choose the real-time or real-time `malloc` code format for rapid prototyping. If your application does not have significant restrictions in code size, memory usage, or stack usage, you may want to continue using the generic real-time (GRT) target throughout development. The real-time format is the most comprehensive code format and supports almost all the built-in blocks.

If your application demands that you limit source code size, memory usage, or maintain a simple call structure, then you should choose the Real-Time Workshop Embedded Coder target, which uses the embedded C format.

Finally, you should choose the S-function format if you are not concerned about RAM and ROM usage and want to:

- Use a model as a component, for scalability
- Create a proprietary S-function `.dll` or MEX-file object
- Interface the generated code using the S-function C API
- Speed up your simulation

Table 4-1 summarizes the various options available for each code format/target available in the Real-Time Workshop.

Table 4-1: Features Supported by Real-Time Workshop Targets and Code Formats

Feature	GRT	Real-Time malloc	RTW Embedded Coder	DOS	Ada	Tornado	S-Func	RSIM	RT Win	xPC	TI DSP
Static memory allocation	X		X	X	X	X	X		X	X	X
Dynamic memory allocation		X				X	X	X			
Continuous time	X	X		X		X	X	X	X	X	
C MEX S-functions (noninline)	X	X		X		X	X	X	X	X	X
Any S-function (inlined)	X	X	X	X	X	X	X	X	X	X	X
Optimized for min. RAM/ROM usage			X		X						
Supports external mode	X	X				X			X	X	X
Intended for rapid prototyping	X	X		X		X			X	X	X
Intended for production code			X		X					X	X
Batch parameter tuning and Monte Carlo methods								X			

Table 4-1: Features Supported by Real-Time Workshop Targets and Code Formats (Continued)

Feature	GRT	Real-Time malloc	RTW Embedded Coder	DOS	Ada	Tornado	S-Func	RSIM	RT Win	xPC	TI DSP
Executes in hard real time	X	X	X	X	X	X			X	X	X
Non real-time executable included	X	X	X		X			X			
Multiple instances of one model (if no Stateflow blocks in model)		X					X				

Real-Time Code Format

The real-time code format (corresponding to the generic real-time target) is useful for rapid prototyping applications. If you want to generate real-time code while iterating model parameters rapidly, you should begin the design process with the generic real-time target. The real-time code format supports:

- Continuous time
- Continuous states
- C MEX S-functions (inlined and noninlined)

For more information on inlining S-functions, see the *Target Language Compiler Reference Guide*.

The real-time code format declares memory statically, that is, at compile time.

Unsupported Blocks

The real-time format does not support the following built-in blocks:

- Functions & Tables
  - MATLAB Fcn
  - S-Function — M-file and Fortran S-functions, C MEX S-functions that call into MATLAB.

System Target Files

- `drt.tlc` — DOS real-time target
- `grt.tlc` — generic real-time target
- `osek_leo.tlc` — Lynx-Embedded OSEK target
- `rsim.tlc` — rapid simulation target
- `tornado.tlc` — Tornado (VxWorks) real-time target

Template Makefiles

- `drt.tmf`
- `grt`
  - `grt_bc.tmf` — Borland C

  - grt_vc.tmf — Visual C
  - grt_watc.tmf — Watcom C
  - grt_lcc.tmf — LCC compiler
  - grt_unix.tmf — UNIX host
- osek_leo.tmf
- rsim
  - rsim_bc.tmf — Borland C
  - rsim_vc.tmf — Visual C
  - rsim_watc.tmf — Watcom C
  - rsim_lcc.tmf — LCC compiler
  - rsim_unix.tmf — UNIX host
- tornado.tmf
- win_watc.tmf

Real-Time malloc Code Format

The real-time `malloc` code format (corresponding to the generic real-time `malloc` target) is very similar to the real-time code format. The differences are:

- Real-time `malloc` declares memory dynamically.
- Real-time `malloc` allows you to multiply instance the same model with each instance maintaining its own unique data.
- Real-time `malloc` allows you to combine multiple models together in one executable. For example, to integrate two models into one larger executable, real-time `malloc` maintains a unique instance of each of the two models. If you do not use the real-time `malloc` format, the Real-Time Workshop will not necessarily create uniquely named data structures for each model, potentially resulting in name clashes.

  `grt_malloc_main.c`, the main routine for the generic real-time `malloc` (`grt_malloc`) target, supports one model by default. See"Combining Multiple Models" on page 17–82 for information on modifying `grt_malloc_main` to support multiple models. `grt_malloc_main.c` is located in the directory `matlabroot/rtw/c/grt_malloc`.

Unsupported Blocks

The real-time `malloc` format does not support the following built-in blocks:

- Functions & Tables
  - MATLAB Fcn
  - S-Function — M-file and Fortran S-functions, C MEX S-functions that call into MATLAB.

System Target Files

- `grt_malloc.tlc`
- `tornado.tlc` — Tornado (VxWorks) real-time target

Template Makefiles

- `grt_malloc`
  - `grt_malloc_bc.tmf` — Borland C

  - `grt_malloc_vc.tmf` — Visual C
  - `grt_malloc_watc.tmf` — Watcom C
  - `grt_malloc_lcc.tmf` — LCC compiler
  - `grt_malloc_unix.tmf` — UNIX host
- `tornado.tmf`

S-Function Code Format

The S-function code format (corresponding to the S-Function Target) generates code that conforms to the Simulink C MEX S-function API. Using the S-Function Target, you can build an S-function component and use it as an S-Function block in another model.

The S-function code format is also used by the Simulink Accelerator to create the Accelerator MEX-file.

In general you should not use the S-function code format in a system target file. However, you may need to do special handling in your inlined TLC files to account for this format. You can check the TLC variable `CodeFormat` to see if the current target is a MEX-file. If `CodeFormat = "S-Function”` and the TLC variable `Accelerator` is set to 1, the target is a Simulink Accelerator MEX-file.

See Chapter 10, “The S-Function Target” for further information.

Embedded C Code Format

The embedded C code format corresponds to the Real-Time Workshop Embedded Coder target. This code format includes a number of memory-saving and performance optimizations. See Chapter 9, “Real-Time Workshop Embedded Coder” for full details.

External Mode

Introduction

External mode allows two separate systems — a *host* and a *target* — to communicate. The host is the computer where MATLAB and Simulink are executing. The target is the computer where the executable created by the Real-Time Workshop runs.

The host (Simulink) transmits messages requesting the target to accept parameter changes or to upload signal data. The target responds by executing the request. External mode communication is based on a *client/server* architecture, in which Simulink is the client and the target is the server.

External mode lets you:

- Modify, or *tune*, block parameters in real time. In external mode, whenever you change parameters in the block diagram, Simulink automatically downloads them to the executing target program. This lets you tune your program's parameters without recompiling. In external mode, the Simulink model becomes a graphical front end to the target program.
- View and log block outputs in many types of blocks and subsystems. You can monitor and/or store signal data from the executing target program, without writing special interface code. You can define the conditions under which data is uploaded from target to host. For example, data uploading could be triggered by a selected signal crossing zero in a positive direction. Alternatively, you can manually trigger data uploading.

External mode works by establishing a communication channel between Simulink and the Real-Time Workshop generated code. This channel is implemented by a low-level *transport layer* that handles physical transmission of messages. Both Simulink and the generated model code are independent of this layer. The transport layer and the code directly interfacing to the transport layer are isolated in separate modules that format, transmit, and receive messages and data packets.

This design makes it possible for different targets to use different transport layers. For example, the GRT, GRT `malloc`, and Tornado targets support host/target communication via TCP/IP, whereas the xPC Target supports both RS232 (serial) and TCP/IP communication. The Real-Time Windows Target implements external mode communication via shared memory.

This chapter discusses the following topics:

- “Tutorial: Getting Started with External Mode Using GRT” on page 5–4 covers the basics of how to use external mode on a single computer.
- “Using the External Mode User Interface” on page 5–16 covers all elements of the external mode user interface in detail.
- “External Mode Compatible Blocks and Subsystems” on page 5–32 discusses the types of blocks that you can use to receive and view signals in external mode.
- “Signal Viewing Subsystems” on page 5-32 shows how you can use subsystems to encapsulate processing and viewing of signals received from the target system. This feature can help you generate a smaller and more efficient target program.
- “Overview of External Mode Communications” on page 5-36 summarizes the communications process between Simulink and the target program.
- “The TCP/IP Implementation” on page 5-38 discusses the TCP/IP-based implementation of the external mode protocol that is provided by the Real-Time Workshop. This section includes information on bundled targets that support this implementation, and on how to build and run target programs that support the TCP/IP implementation.
- “Limitations of External Mode” on page 5-45 discusses limitations on the use of external mode that are imposed by the structure of the model.

Additional Reading

“Creating an External Mode Communication Channel” on page 17–73 contains advanced information for those who want to implement their own external mode communications layer. You may want to read it for additional insight into the architecture and code structure of external mode communications.

Chapter 12, “Targeting Tornado for Real-Time Applications” discusses the use of external mode in the VxWorks Tornado environment.

Tutorial: Getting Started with External Mode Using GRT

This section provides step-by-step instructions for getting started with external mode. This tutorial assumes you have basic familiarity with MATLAB and Simulink. In addition, you should read Chapter 1, “Introduction to the Real-Time Workshop.” Read “Getting Started: Basic Concepts and Tutorials” on page 1-37 and do the “Quick Start Tutorials” on page 1-40 before proceeding.

The example presented uses the generic real-time (GRT) target. The example does not require any hardware other than the computer on which you run Simulink and the Real-Time Workshop. The generated executable in this example runs on the host computer under a separate process from MATLAB and Simulink. This technique is called *self-targeting*.

The procedures for building, running, and testing your programs are almost identical in UNIX and PC environments. The discussion notes differences where applicable.

For a more thorough description of external mode, including a discussion of all the options available, see “Using the External Mode User Interface” on page 5-16.

Part 1: Setting Up the Model

In this part of the tutorial, you create a simple model, `ext_example`, and a directory called `ext_mode_example` to store the model and the generated executable:

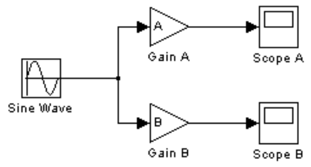
1. Create the directory from the MATLAB command line by typing

   ```
   mkdir ext_mode_example
   ```

2. Make `ext_mode_example` your working directory.

   ```
   cd ext_mode_example
   ```

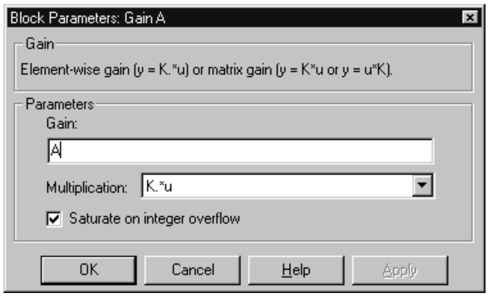
3 Create a model in Simulink with a Sine Wave block for the input signal, two Gain blocks in parallel, and two Scope blocks. The model is shown below. Label the Gain and Scope blocks as shown.



4 Define and assign two variables `A` and `B` in the MATLAB workspace as follows.

```
A = 2; B = 3;
```

5 Open Gain block A and sets its Gain parameter to the variable `A` as shown below.


Block Parameters: Gain A

Gain

Element-wise gain (y = K.*u) or matrix gain (y = K*u or y = u*K).

Parameters

Gain:

A

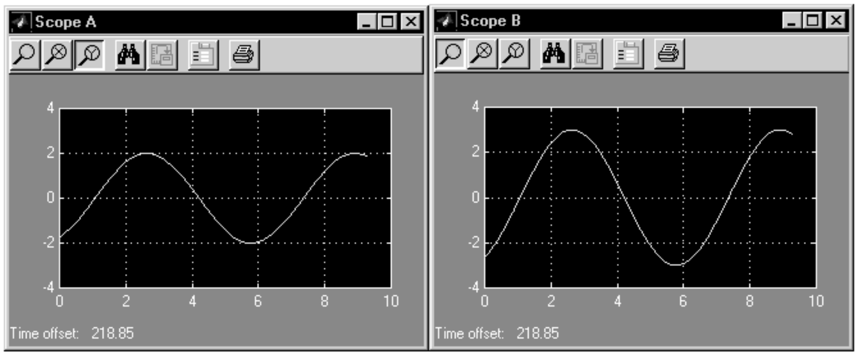
Multiplication: K.*u

☑ Saturate on integer overflow

6 Similarly, open Gain block B and sets its Gain parameter to the variable `B`.

When the target program is built and connected to Simulink in external mode, new gain values can be downloaded to the executing target program by assigning new values to workspace variables `A` and `B`, or by editing the values in the block parameter dialog boxes.

7 Verify correct operation of the model. Open the Scope blocks and run the model. Given that `A=2` and `B=3`, the output should look like this.



8 From the **File** menu, choose **Save As**. Save the model as `ext_example.mdl`.

Part 2: Building the Target Executable

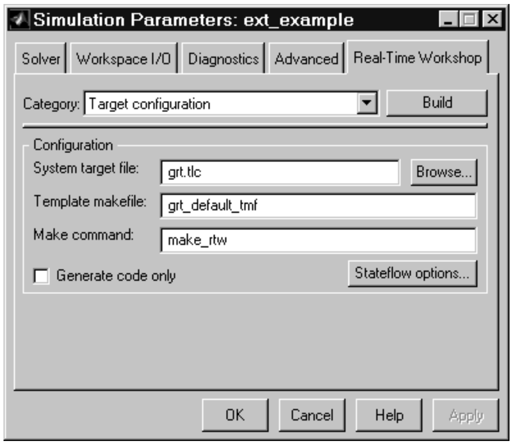
In this section, you set up the model and code generation parameters required for an external mode compatible target program. Then you generate code and build the target executable.

9 Open the **Simulation Parameters** dialog box. On the Solver page, set the **Solver options Type** to `Fixed-step`, and the algorithm to `discrete (no continuous states)`. Set **Fixed-step Size** to `0.01`. Leave the other parameters at their default values.

10 On the Workspace I/O page, deselect **Time** and **Output**. In this exercise, data will not be logged to the workspace or to a MAT-file.

11 On the Real-Time Workshop page, select **Target configuration** from the **Category** menu.

By default, the GRT target should be selected, as shown in this picture.



Category: Target configuration

System target file: grt.tlc

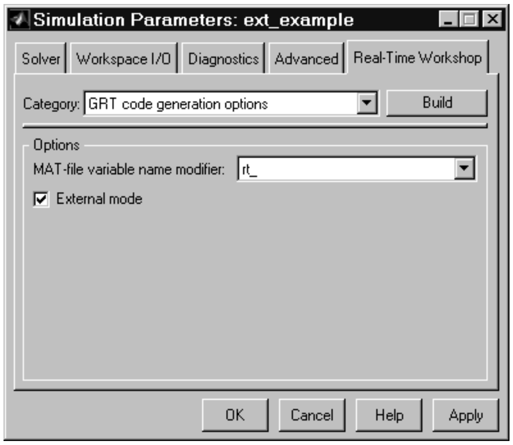
Template makefile: grt_default_tmf

Make command: make_rtw

☐ Generate code only

If the GRT target is *not* selected, click the **Browse** button and select the GRT target from the System Target File Browser. Then click **OK** to close the browser. Return to the Real-Time Workshop page and click **Apply**.

12 Select **GRT code generation options** from the **Category** menu and check the **External mode** option.This enables generation of external mode support code.



Simulation Parameters: ext_example

Category: GRT code generation options

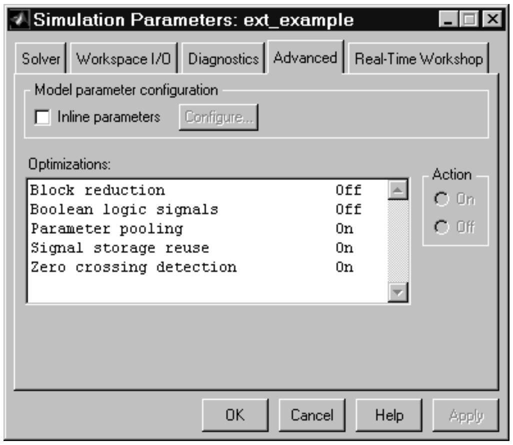
MAT-file variable name modifier: rt_

[✓] External mode

13 Click **Apply**.

14 On the Advanced page, make sure that the **Inline parameters** option is deselected. External mode does not currently support the **Inline parameters** option.

The Advanced page should look like this picture.

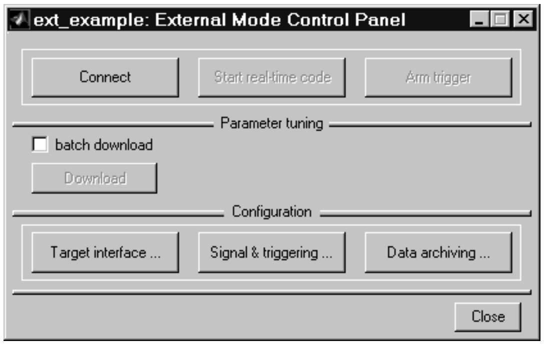


☐ Inline parameters

Optimizations:

```
Block reduction              Off
Boolean logic signals        Off
Parameter pooling            On
Signal storage reuse         On
Zero crossing detection      On
```

15 From the **Tools** menu, select **External Mode Control Panel**. The **External Mode Control Panel**, lets you configure host/target communications, signal monitoring, and data archiving. It also lets you connect to the target program and start and stop execution of the model code.



☐ batch download

The top four buttons are for use after the target program has been launched. The three lower buttons open three separate dialog boxes:

- The **Target interface** button opens the **External Target Interface** dialog box, which configures the external mode communications channel.
- The **Signal & triggering** button opens the **External Signal & Triggering** dialog box, which configures which signals are viewed and how signals are acquired.
- The **Data archiving** button opens the **External Data Archiving** dialog box. Data archiving lets you save data sets generated by the target program for future analysis. This example does not use data archiving. See "Data Archiving" on page 5-28 for more information.

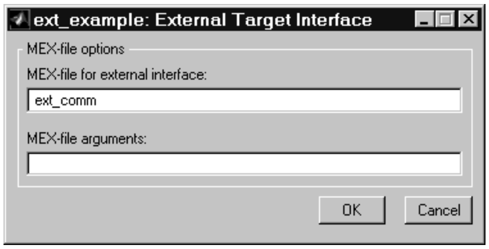
16 Click the **Target interface** button to open the **External Target Interface** dialog box. This dialog box configures the external mode interface options.

The **MEX-file for external interface** field specifies the name of a MEX-file that supports host/target communications on the host side. The default is `ext_comm`, a MEX-file provided by the Real-Time Workshop. `ext_comm` supports communication via the TCP/IP communications protocol.

The **MEX-file arguments** field lets you specify arguments, such as a TCP/IP server port number, to be passed to the external interface program. Note that these arguments are specific to the external interface file you are using. For information on these arguments, see "The External Interface MEX-File" on page 5-40.

This exercise uses the default arguments. Leave the **MEX-file arguments** field blank.

The **External Target Interface** dialog box should appear as shown below.


ext_example: External Target Interface

MEX-file options

MEX-file for external interface: ext_comm

MEX-file arguments:

17 Click **OK** to close the **External Target Interface** dialog box and return to the **External Mode Control Panel**.

18 Close the **External Mode Control Panel**.

19 Save the model.

20 Return to the Real-Time Workshop page. Click **Build** to generate code and create the target program. The content of the succeeding messages depends on your compiler and operating system.The final message is

```
### Successful completion of Real-Time Workshop build procedure
for model: ext_example
```

In the next section, you will run the `ext_example` executable and use Simulink as an interactive front end to the running target program.

Part 3: Running the External Mode Target Program

The target executable, `ext_example`, is now in your working directory. In this section, you run the target program and establish communication between Simulink and the target.

The **External Signal & Triggering** dialog box displays a list of all the blocks in your model that support external mode signal monitoring and logging. The **External Signal & Triggering** dialog box also lets you configure which signals are viewed and how they are acquired and displayed. You can reconfigure the **External Signal & Triggering** dialog box while the target program runs.

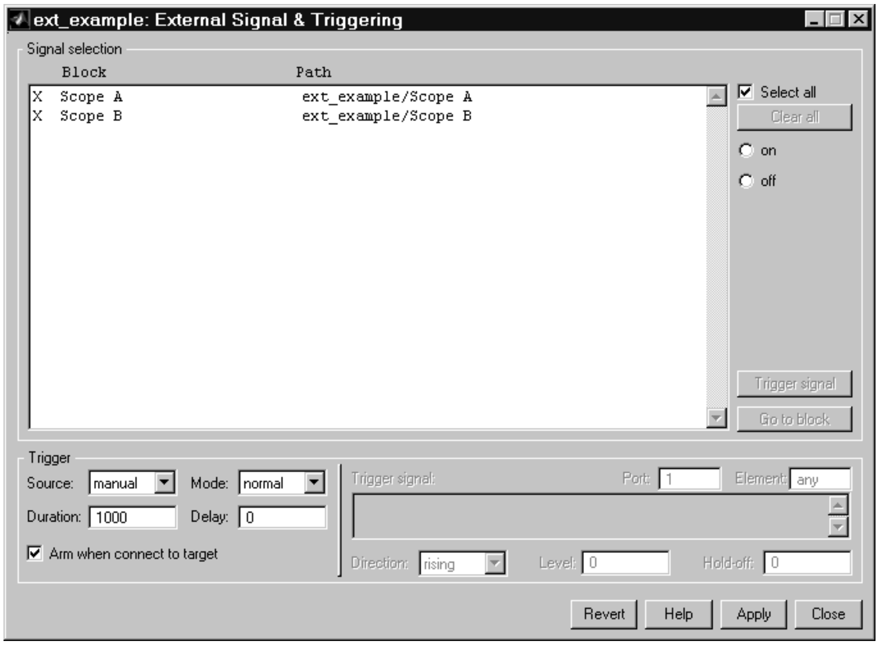
In this exercise you will observe and use the default settings of the **External Signal & Triggering** dialog box.

21 From the **Tools** menu, select **External Mode Control Panel**.

22 In the **External Mode Control Panel**, click the **Signal & triggering** button.

23 The **External Signal & Triggering** dialog box opens. The default configuration of the **External Signal & Triggering** dialog box is designed to ensure that all signals are selected for monitoring. The default configuration also ensures that signal monitoring will begin as soon as the host and target

programs have connected. The figure below shows the default configuration for `ext_example`.


ext_example: External Signal & Triggering

Signal selection

Block	Path
X Scope A	ext_example/Scope A
X Scope B	ext_example/Scope B

Trigger

Source: manual
Mode: normal
Duration: 1000
Delay: 0
Arm when connect to target: selected
Trigger signal:
Port: 1
Element: any
Direction: rising
Level: 0
Hold-off: 0

24 Make sure that the **External Signal and Triggering** dialog box is set to the defaults as shown:

- **Select all** check box is selected. All signals in the **Signal selection** list are are marked with an `X` in the **Block** column.)
- **Trigger Source**: `manual`
- **Trigger Mode**: `normal`
- **Duration**: `1000`
- **Delay**: `0`
- **Arm when connect to target**: selected

Click **Close** and return to the **External Mode Control Panel**. Close the **External Mode Control Panel**.

25 To run the target program, you must open an MS-DOS command prompt (on UNIX systems, an Xterm window). At the command prompt, type

```
ext_example -tf inf -w
```

and the target program begins execution.

The -tf switch overrides the stop time set for the model in Simulink. The inf value directs the model to run indefinitely. The model code will run until the target program receives a stop message from Simulink.

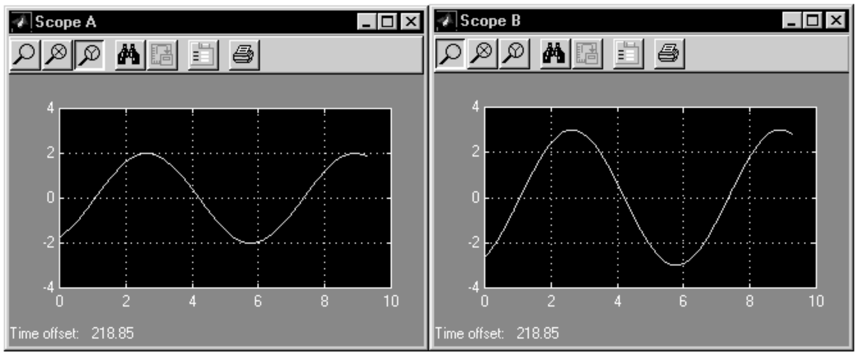
The -w switch instructs the target program to enter a wait state until it receives a **Start real-time code** message from the host. This switch is required if you want to view data from time step 0 of the target program execution, or if you want to modify parameters before the target program begins execution of model code.

26 Open Scope blocks A and B. At this point, no signals are visible on the scopes. When you connect Simulink to the target program and begin model execution, the signals generated by the target program will be visible on the scope displays.

27 The model must be in external mode before communication between the model and the target program can begin. To enable external mode, select **External** from the simulation mode menu in the toolbar of the Simulink window. Alternatively, you can select **External** from the **Simulation** menu.

28 Reopen the **External Mode Control Panel** and click **Connect**. This initiates a handshake between Simulink and the target program. When Simulink and the target are connected, the **Start real-time code** button becomes enabled, and the caption of the **Connect** button changes to **Disconnect**.

29 Click the **Start real-time code** button.You should see the outputs of Gain blocks A and B on the two scopes in your model. With A=2 and B=3, the output looks like this.



Having established communication between Simulink and the running target program, you can tune block parameters in Simulink and observe the effects the parameter changes have on the target program. You will do this in the next section.

Part 4: Tuning Parameters

You can change the gain factor of either Gain block by assigning new values to the variables A or B in the MATLAB workspace. When you change block parameter values in the workspace during a simulation, you must explicitly update the block diagram with these changes. When the block diagram is updated, the new values are downloaded to the target program. To tune the variables A and B:

30 In the MATLAB command window, assign new values to both variables, for example

```
A = 0.5;B = 3.5;
```

31 Activate the ext_example model window. Select **Update Diagram** from the **Edit** menu, or type **Ctrl+D**. As soon as Simulink has updated the block parameters, the new gain values are downloaded to the target program, and the effect of the gain change becomes visible on the scopes.

You can also enter gain values directly into the Gain blocks. To do this:

32 Open the dialog box for Gain block A or B in the model.

33 Enter a new numerical value for the gain and click **Apply**. As soon as you click **Apply**, the new value is downloaded to the target program and the effect of the gain change becomes visible on the scope.

   Similarly, you can change the frequency, amplitude, or phase of the sine wave signal by opening the dialog box for the Sine Wave block and entering a new numerical value in the appropriate field.

   Note, however, that you cannot change the sample time of the Sine Wave block. Block sample times are part of the structural definition of the model and are part of the generated code. Therefore, if you want to change a block sample time, you must stop the external mode simulation and rebuild the executable.

34 To simultaneously disconnect host/target communication and end execution of the target program, pull down the **Simulation** menu and select **Stop real-time code**.

Using the External Mode User Interface

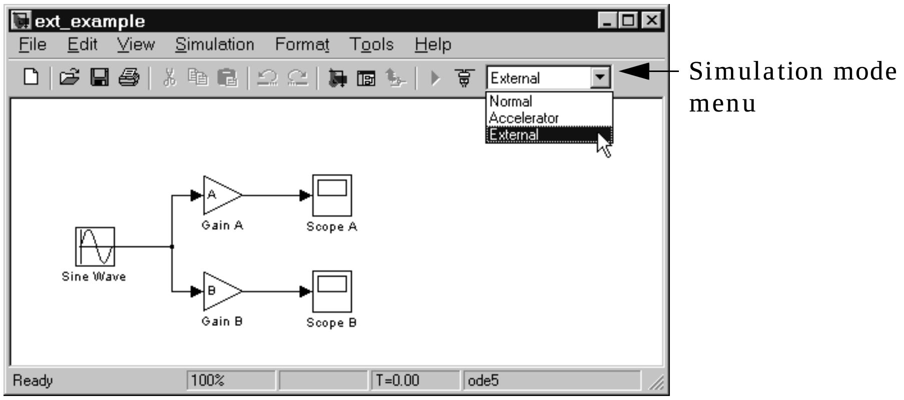
This section discusses the elements of the Simulink and Real-Time Workshop user interface that control the operation of external mode. These elements include:

- External mode related menu items in **Simulation** and **Tools** menus and in the Simulink toolbar.
- **External Mode Control Panel**
- **External Target Interface** dialog box
- **External Signal & Triggering** dialog box
- **External Data Archiving** dialog box

External Mode Related Menu and Toolbar Items

To communicate with a target program, the model must be operating in external mode. The **Simulation** menu and the toolbar provide two ways to enable external mode:

- Select **External** from the **Simulation** menu.
- Select **External** from the simulation mode menu in the toolbar. The simulation mode menu is shown in this picture.



Once external mode is enabled, you can use the **Simulation** menu or the toolbar to connect to and control the target program.

Note You can enable external mode, and simultaneously connect to the target system, by using the **External Mode Control Panel**. See "External Mode Control Panel" on page 5-21.

Simulation Menu

When Simulink is in external mode, the upper section of the **Simulation** menu contains external mode options. Initially, Simulink is disconnected from the target program, and the menu displays the options shown in this picture.

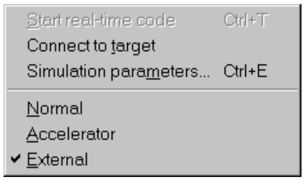


Figure 5-1: Simulation Menu External Mode Options (Host Disconnected from Target)

The **Connect to target** option establishes communication with the target program. When a connection is established, the target program may be executing model code, or it may be awaiting a command from the host to start executing model code.

If the target program is executing model code, the **Simulation** menu contents change, as shown in this picture.

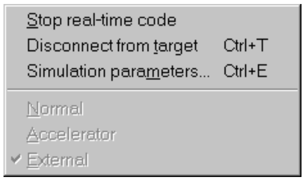

Stop real-time code
Disconnect from target Ctrl+T
Simulation parameters... Ctrl+E
Normal
Accelerator
✓ External

Figure 5-2: Simulation Menu External Mode Options (Target Executing Model Code)

The **Disconnect from target** option disconnects Simulink from the target program, which continues to run. The **Stop real-time code** option terminates execution of the target program and disconnects Simulink from the target system.

If the target program is in a wait state, the **Start real-time code** option is enabled, as shown in this picture. The **Start real-time code** option instructs the target program to begin executing the model code.

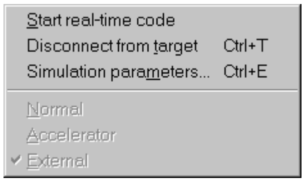

Start real-time code
Disconnect from target Ctrl+T
Simulation parameters... Ctrl+E
Normal
Accelerator
✓ External

Figure 5-3: Simulation Menu External Mode Options (Target Awaiting Start Command)

Toolbar Controls

The Simulink toolbar controls, shown in Figure 5-4, let you control the same external mode functions as the **Simulation** menu. Simulink displays external mode icons to the left of the Simulation mode menu. Initially, the toolbar displays a **Connect to target** icon and a disabled **Start real-time code** button (shown in Figure 5-4). Click on the **Connect to target** icon to connect Simulink to the target program.

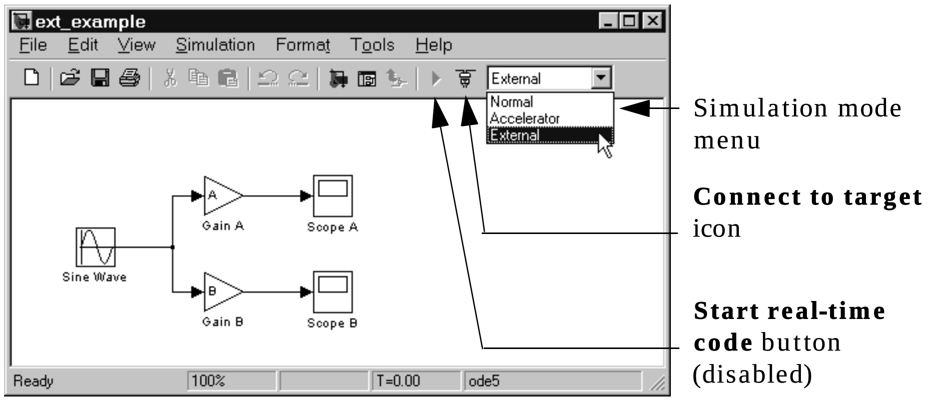


Figure 5-4: External Mode Toolbar Controls (Host Disconnected from Target)

When a connection is established, the target program may be executing model code, or it may be awaiting a command from the host to start executing model code.

If the target program is executing model code, the toolbar displays a **Stop real-time code** button and a **Disconnect from target** icon (shown in Figure 5-5). Click on the **Stop real-time code** button to command the target program to stop executing model code and disconnect Simulink from the target system. Click on the **Disconnect from target** icon to disconnect Simulink from the target program while leaving the target program running.

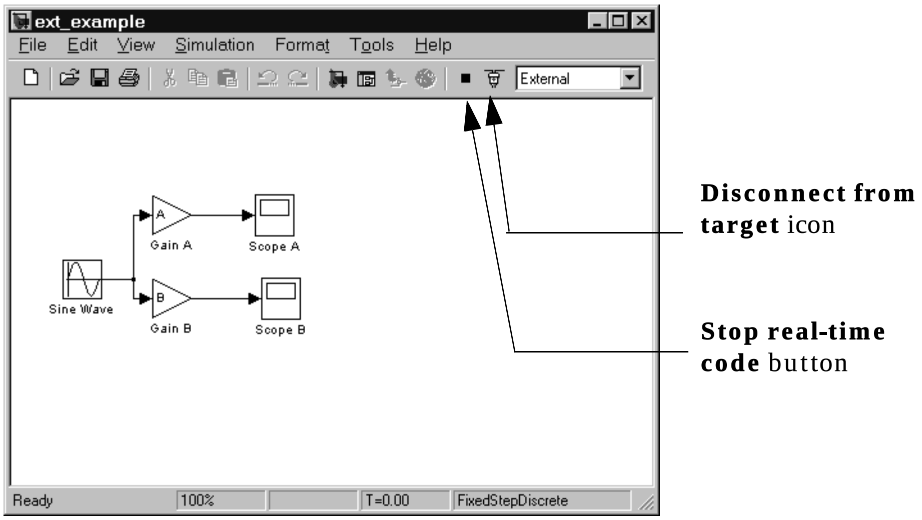

ext_example
File Edit View Simulation Format Tools Help
External
Gain A
Scope A
Sine Wave
Gain B
Scope B
Ready
100%
T=0.00
FixedStepDiscrete
Disconnect from target icon
Stop real-time code button

Figure 5-5: External Mode Toolbar Controls (Target Executing Model Code)

If the target program is in a wait state, the toolbar displays a **Start real-time code** button and a **Disconnect from target** icon (shown in Figure 5-6). Click on the **Start real-time code** button to instruct the target program to start executing model code. Click on the **Disconnect from target** icon to disconnect Simulink from the target program.

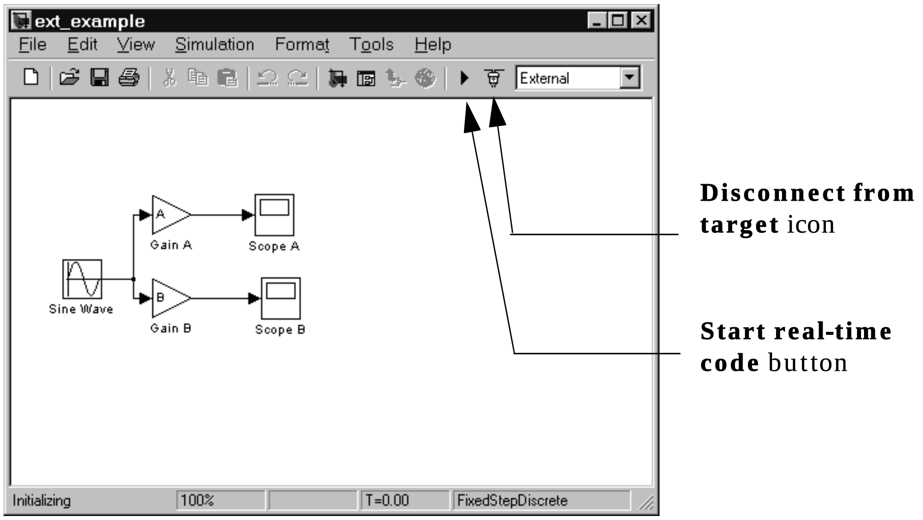

ext_example
File Edit View Simulation Format Tools Help
External
Sine Wave
Gain A
Gain B
Scope A
Scope B
Initializing
100%
T=0.00
FixedStepDiscrete
Disconnect from target icon
Start real-time code button

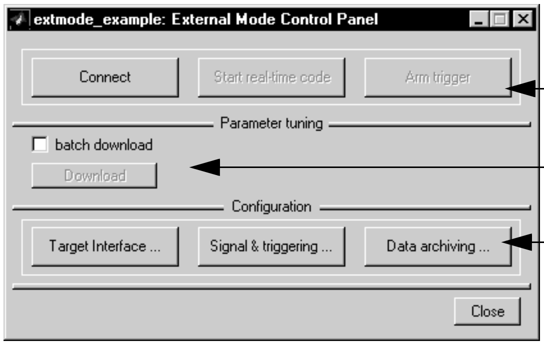
Figure 5-6: External Mode Toolbar Controls (Target in Wait State)

External Mode Control Panel

The **External Mode Control Panel** provides centralized control of all external mode features, including:

- Host/target connection, disconnection, and target program start/stop functions, and enabling of external mode
- Arming and disarming the data upload trigger
- External mode communications configuration
- Timing of parameter downloads
- Selection of signals from the target program to be viewed and monitored on the host
- Configuration of data archiving features

Select **External mode control panel** from the Simulink **Tools** menu to open the **External Mode Control Panel**.



These buttons control the connection between host and manual arming of the data uploading trigger.

This check box and button control the timing of parameter downloads.

These buttons open dialog boxes that configure external mode target interface, signal properties, and data archiving.

The following sections describe the features supported by the **External Mode Control Panel**.

Connection and Start/Stop Controls

The **External Mode Control Panel** performs the same connect/disconnect and start/stop functions found in the **Simulation** menu and the Simulink toolbar (see “External Mode Related Menu and Toolbar Items” on page 5-16.)

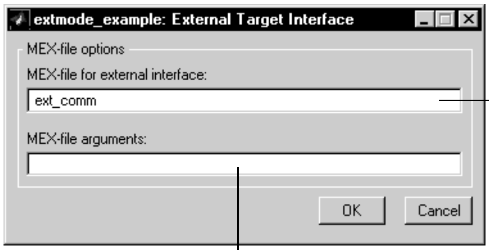
The **Connect**/**Disconnect** button connects to or disconnects from the target program. The button text changes in accordance with the state of the connection.

Note that if external mode is not enabled at the time the **Connect** button is clicked, the **External Mode Control Panel** enables external mode automatically.

The **Start**/**Stop real-time code** button commands the target to start or terminate model code execution. The button is disabled until a connection to the target is established. The button text changes in accordance with the state of the target program.

Target Interface Dialog Box

Pressing the **Target Interface** button activates the **External Target Interface** dialog box.


extmode_example: External Target Interface
MEX-file options
MEX-file for external interface:
ext_comm
MEX-file arguments:
OK
Cancel

Specify name of external interface MEX-file here. Default is ext_comm.

Enter optional arguments to the external interface MEX-file here.

The **External Target Interface** dialog box lets you specify the name of a MEX-file that implements host/target communications. This is known as the external interface MEX-file. The fields of the **External Target Interface** dialog box are:

- **MEX-file for external interface**: Name of the external interface MEX-file. The default is ext_comm, the TCP/IP-based external interface file provided for use with the GRT and Tornado targets

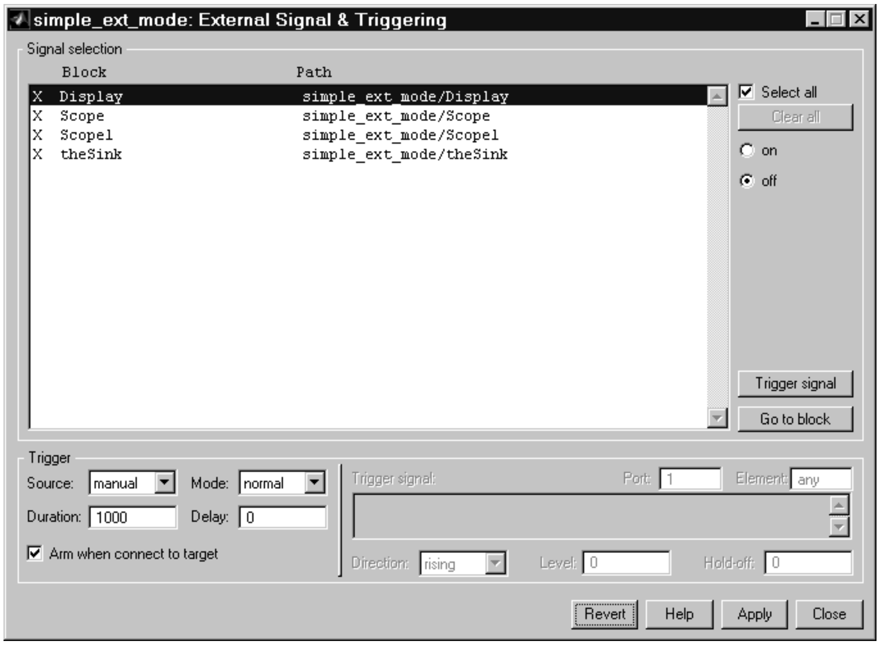
  The external interface MEX-file for the Real-Time Windows Target is win_tgt.

  Custom or third-party targets may use a different external interface MEX-file.
- **MEX-file arguments**: Arguments for the external interface MEX-file. For example, ext_comm allows three optional arguments: the network name of your target, the verbosity level, and a TCP/IP server port number.

See “The External Interface MEX-File” on page 5-40 for details on ext_comm and its arguments.

External Signal & Triggering Dialog Box

Clicking the **Signal & triggering** button activates the **External Signal & Triggering** dialog box.



simple_ext_mode: External Signal & Triggering

Signal selection

```
    Block                          Path
X   Display                        simple_ext_mode/Display
X   Scope                          simple_ext_mode/Scope
X   Scope1                         simple_ext_mode/Scope1
X   theSink                        simple_ext_mode/theSink
```

[✓] Select all

on [✓ off]

Trigger

Source: manual Mode: normal

Duration: 1000 Delay: 0

[✓] Arm when connect to target

Trigger signal: Port: 1 Element: any

Direction: rising Level: 0 Hold-off: 0

Figure 5-7: Default Settings of the External Signal & Triggering Dialog Box

The **External Signal & Triggering** dialog box displays a list of all blocks and subsystems in your model that support external mode signal uploading. See "External Mode Compatible Blocks and Subsystems" on page 5-32 for information on which types of blocks are external mode compatible.

The **External Signal & Triggering** dialog box lets you select which signals are collected from the target system and viewed in external mode. It also lets you select a signal that triggers uploading of data when certain signal conditions are met, and define the triggering conditions.

Default Operation

Figure 5-7 shows the default settings of the **External Signal and Triggering** dialog box. The default operation of the **External Signal and Triggering** dialog box is designed to simplify monitoring the target program. If you use the default settings, you do not need to preconfigure signals and triggers. Simply start the target program and connect the Simulink model to it. All external mode compatible blocks will be selected and the trigger will be armed. Signal uploading will begin immediately upon connection to the target program.

The default configuration is:

- **Arm when connect to target**: on
- **Trigger Mode**: normal
- **Trigger Source**: manual
- **Select all**: on

Signal Selection

All external mode compatible blocks in your model appear in the **Signal selection** list of the **External Signal & Triggering** dialog box. You use this list to select signals to be viewed. An X appears to the left of each selected block's name.

The **Select all** check box selects all signals. By default, **Select all** is on.

If **Select all** is off, you can select or deselect individual signals using the **on** and **off** radio buttons. To select a signal, click on the desired list entry and click the **on** radio button. To deselect a signal, click on the desired list entry and click the **off** radio button. Alternatively, you can double-click a signal in the list to toggle between selection and deselection.

The **Clear all** button deselects all signals.

Trigger Options

The **Trigger** panel located at the bottom left of the **External Signal & Triggering** dialog box contains options that control when and how signal data is collected (uploaded) from the target system. These options are:

- **Source**: `manual` or `signal`. Selecting `manual` directs external mode to start logging data when the **Arm trigger** button on the **External Mode Control Panel** is clicked.

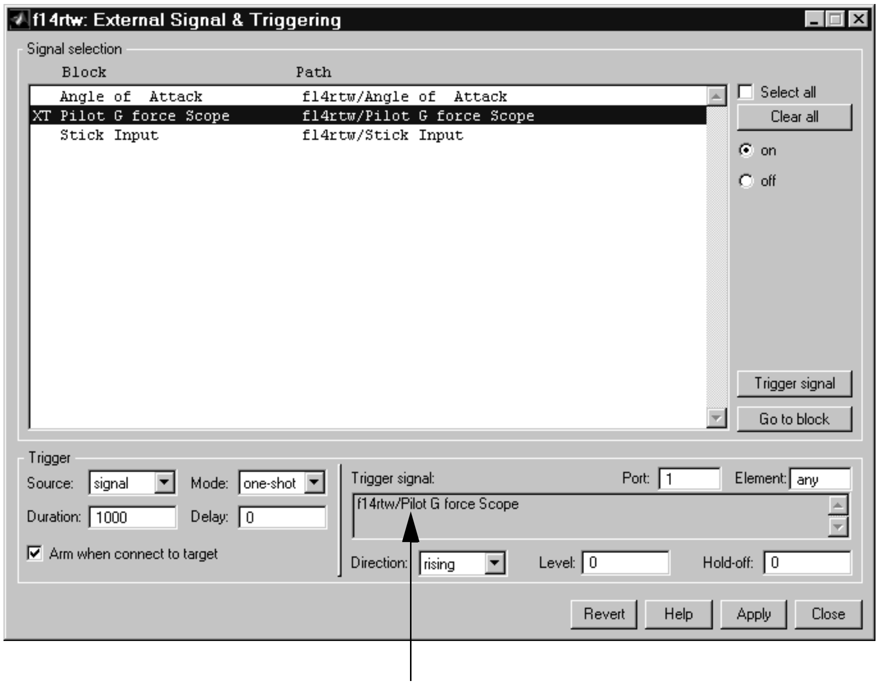
  Selecting `signal` tells external mode to start logging data when a selected trigger signal satisfies trigger conditions specified in the **Trigger signal** panel. When the trigger conditions are satisfied (that is, the signal crosses the trigger level in the specified direction) a *trigger event* occurs. If the trigger is *armed*, external mode monitors for the occurrence of a trigger event. When a trigger event occurs, data logging begins.
- **Arm when connect to target**: If this option is selected, external mode arms the trigger automatically when Simulink has connected to the target. If the trigger source is `manual`, uploading begins immediately. If the trigger mode is `signal`, monitoring of the trigger signal begins immediately, and uploading begins upon the occurrence of a trigger event.

  If **Arm when connect to target is not selected,** you must manually arm the trigger by clicking the **Arm trigger** button in the **External Mode Control Panel**.
- **Duration**: The number of base rate steps for which external mode logs data after a trigger event. For example, if the fastest rate in the model is 1 second and a signal sampled at 1 Hz is being logged for a duration of 10 seconds, then external mode will collect 10 samples. If a signal sampled at 2 Hz is logged, only 5 samples will be collected.
- **Mode**: `normal` or `one-shot`. In `normal` mode, external mode automatically rearms the trigger after each trigger event. In `one-shot` mode, external mode collects only one buffer of data each time you arm the trigger. See "Data Archiving" on page 5-28 for further details on the effect of the **Mode** setting.
- **Delay**: The delay represents the amount of time that elapses between a trigger occurrence and the start of data collection. The delay is expressed in base rate steps, and can be positive or negative. A negative delay corresponds to pretriggering. When the delay is negative, data from the time preceding the trigger is collected and uploaded.

Trigger Signal Selection

You can designate one signal as a trigger signal. To select a trigger signal, select `signal` from the **Trigger Source** menu. This activates the **Trigger signal** panel (see Figure 5-8). Then, click on the desired entry in the **Signal selection** list, and click the **Trigger signal** button.

When a signal is selected as a trigger, a `T` appears to the left of the block's name in the **Signal selection** list. In Figure 5-8, the `Pilot G force Scope` signal is the trigger. `Pilot G force Scope` is also selected for viewing, as indicated by the `X` to the left of the block name.



f14rtw: External Signal & Triggering

Signal selection

```
   Block                  Path
   Angle of  Attack       f14rtw/Angle of  Attack
XT Pilot G force Scope    f14rtw/Pilot G force Scope
   Stick Input            f14rtw/Stick Input
```

☐ Select all

Clear all

(● on) ○ off

Trigger signal

Go to block

Trigger

Source: signal   Mode: one-shot

Duration: 1000   Delay: 0

☑ Arm when connect to target

Trigger signal: Port: 1   Element: any

f14rtw/Pilot G force Scope

Direction: rising   Level: 0   Hold-off: 0

Revert   Help   Apply   Close

The **Trigger Signal** panel

Figure 5-8: Signals & Triggering Window with Trigger Selected

After selecting the trigger signal, you can define the trigger conditions in the **Trigger signal** panel, and set the **Port** and **Element** fields located on the right side of the **Trigger** panel.

Setting Trigger Conditions

Note The **Trigger signal** panel and the **Port** and **Element** fields of the **External Signal & Trigger** dialog box are enabled only when **Trigger source** is set to `signal`.

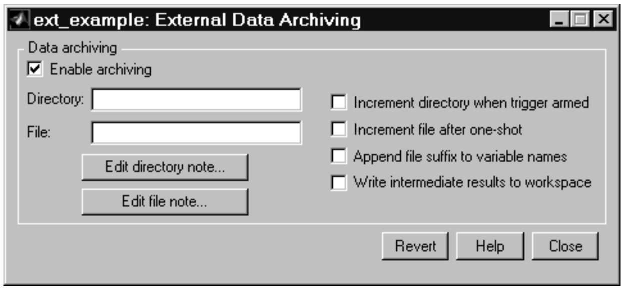
By default, any element of the first input port of the specified trigger block can cause the trigger to fire (i.e., Port 1, any element). You can modify this behavior by adjusting the **Port** and **Element** fields located on the right side of the Trigger panel. The **Port** field accepts a number or the keyword `last`. The **Element** field accepts a number or the keywords `any` and `last`.

The **Trigger Signal** panel defines the conditions under which a trigger event will occur. These are:

- **Level**: Specifies a threshold value. The trigger signal must cross this value in a designated direction to fire the trigger. By default, the level is 0.
- **Direction**: `rising`, `falling`, or `either`. This specifies the direction in which the signal must be travelling when it crosses the threshold value. The default is `rising`.
- **Hold-off**: Applies only to `normal` mode. Expressed in base rate steps, **Hold-off** is the time between the termination of one trigger event and the rearming of the trigger.

Data Archiving

Pressing the **Data Archiving** button of the **External Mode Control Panel** opens the **External Data Archiving** dialog box.



ext_example: External Data Archiving

Data archiving

☑ Enable archiving

Directory:

File:

☐ Increment directory when trigger armed

☐ Increment file after one-shot

☐ Append file suffix to variable names

☐ Write intermediate results to workspace

This panel supports the following features:

Directory Notes. Use this option to add annotations that pertain to a collection of related data files in a directory.

Pressing the **Edit directory note** button opens the MATLAB editor. Place comments that you want saved to a file in the specified directory in this window. By default, the comments are saved to the directory last written to by data archiving.

File Notes. Pressing **Edit file note** opens a file finder window that is, by default, set to the last file to which you have written. Selecting any MAT-file opens an edit window. Add or edit comments in this window that you want saved with your individual MAT-file.

Data Archiving. Clicking the **Enable Archiving** check box activates the automated data archiving features of external mode. To understand how the archiving features work, it is necessary to consider the handling of data when archiving is not enabled. There are two cases, one-shot and normal mode.

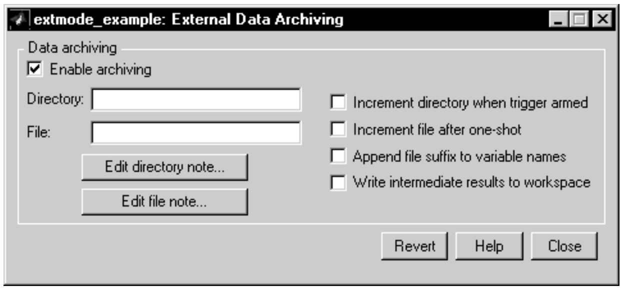
In one-shot mode, after a trigger event occurs, each selected block writes its data to the workspace just as it would at the end of a simulation. If another one-shot is triggered, the existing workspace data will be overwritten.

In normal mode, external mode automatically rearms the trigger after each trigger event. Consequently, you can think of normal mode as a series of one-shots. Each one-shot in this series, except for the last, is referred to as an *intermediate result*. Since the trigger can fire at any time, writing intermediate results to the workspace generally results in unpredictable overwriting of the workspace variables. For this reason, the default behavior is to write only the results from the final one-shot to the workspace. The intermediate results are discarded. If you know that sufficient time exists between triggers for inspection of the intermediate results, then you can override the default behavior by checking the **Write intermediate results to workspace** check box. Note that this option does not protect the workspace data from being overwritten by subsequent triggers.

The options in the **External Data Archiving** dialog box support automatic writing of logging results, including intermediate results, to disk. Data archiving provides the following settings:

- **Directory**: Specifies the directory in which data is saved. External mode appends a suffix if you select **Increment directory when trigger armed**.
- **File**: Specifies the filename in which data is saved. External mode appends a suffix if you select **Increment file after one-shot**.
- **Increment directory when trigger armed:** External mode uses a different directory for writing log files each time that you press the **Arm trigger** button. The directories are named incrementally; for example: `dirname1`, `dirname2`, and so on.
- **Increment file after one-shot**: New data buffers are saved in incremental files: `filename1`, `filename2`, etc. Note that this happens automatically in normal mode.
- **Append file suffix to variable names**: Whenever external mode increments filenames, each file contains variables with identical names. Choosing **Append file suffix to variable name** results in each file containing unique variable names. For example, external mode will save a variable named `xdata` in incremental files (`file_1`, `file_2`, etc.) as `xdata_1`, `xdata_2`, and so on. This is useful if you want to load the MAT-files into the workspace and compare variables in MATLAB. Without the unique names, each instance of `xdata` would overwrite the previous one in the MATLAB workspace.

This picture shows the **External Data Archiving** dialog box with archiving enabled.


extmode_example: External Data Archiving

Data archiving

[✓] Enable archiving

Directory:

File:

[ ] Increment directory when trigger armed

[ ] Increment file after one-shot

[ ] Append file suffix to variable names

[ ] Write intermediate results to workspace

Unless you select **Enable archiving**, entries for the **Directory** and **File** fields are not accepted.

Parameter Download Options

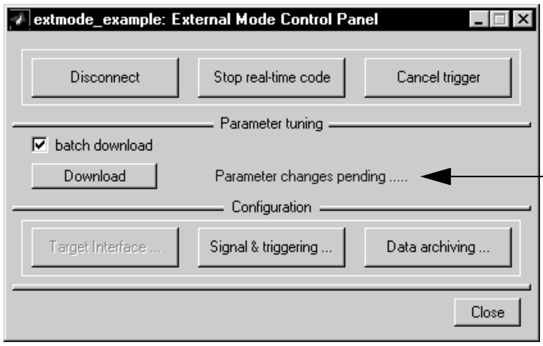
The **batch download** check box on the **External Mode Control Panel** enables or disables batch parameter changes.

By default, **batch download** is not enabled. When **batch download** is not enabled, changes made directly to block parameters are sent immediately to the target. Changes to MATLAB workspace variables are sent when an **Update diagram** is performed.

When **batch download** is enabled, the **Download** button is enabled. Changes made to block parameters are stored locally until you click the **Download** button. When you click the **Download** button, the changes are sent in a single transmission.

When parameter changes have been made and are awaiting batch download, the **External Mode Control Panel** displays the message **Parameter changes pending...** to the right of the download button. (See Figure 5-9.) This message disappears after Simulink receives notification from the target that the new parameters have been installed into the parameter vector of the target system.

Figure 5-9 shows the **External Mode Control Panel** with the batch download option activated.



Parameter changes pending... message appears if unsent parameter value changes are awaiting download.

Figure 5-9: External Mode Control Panel in Batch Download Mode

External Mode Compatible Blocks and Subsystems

Compatible Blocks

In external mode, you can use the following types of blocks to receive and view signals uploaded from the target program:

- Scope blocks
- Blocks in the Dials & Gauges Blockset
- Display blocks
- To Workspace blocks
- User-written S-Function blocks

  An external mode method has been added to the S-function API. This method allows user-written blocks to support external mode. See `*matlabroot*/simulink/simstruc.h`.
- XY Graph blocks

In addition to these types of blocks, you can designate certain subsystems as Signal Viewing Subsystems and use them to receive and view signals uploaded from the target program. See "Signal Viewing Subsystems" on page 5-32 for further information.

External mode compatible blocks and subsystems are selected, and the trigger is armed, via the **External Signal and Triggering** dialog box. For example, Figure 5-7 shows two Scope blocks, a Display block, and a Signal Viewing Subsystem (`theSink`). All of these are selected and the trigger is set to be armed when connected to the target program.

Signal Viewing Subsystems

A Signal Viewing Subsystem is an atomic subsystem that encapsulates processing and viewing of signals received from the target system. A Signal Viewing Subsystem runs only on the host, generating no code in the target system. Signal Viewing Subsystems run in all simulation modes — normal, accelerated, and external.

Signal Viewing Subsystems are useful in situations where you want to process or condition signals before viewing or logging them, but you do not want to perform these tasks on the target system. By using a Signal Viewing

Subsystem, you can generate smaller and more efficient code on the target system.

Like other external mode compatible blocks, Signal Viewing Subsystems are displayed in the **External Signal and Triggering** dialog box.

To declare a subsystem to be a Signal Viewing Subsystem:

1 Select the **Treat as atomic unit** option in the **Block Parameters** dialog box.

   See “Nonvirtual Subsystem Code Generation” on page 3-41 for further information on atomic subsystems.

2 Use the following `set_param` command to turn the `SimViewingDevice` property on.

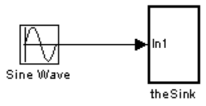
   ```
   set_param('blockname', 'SimViewingDevice','on')
   ```

   where `'blockname'` is the name of the subsystem.

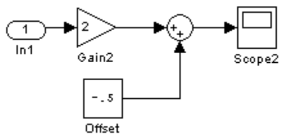
3 Make sure the subsystem meets the following requirements:

   - It must be a pure sink block. That is, it must contain no Outport blocks or Data Store blocks. It may contain Goto blocks only if the corresponding from blocks are contained within the subsystem boundaries.
   - It must have no continuous states.

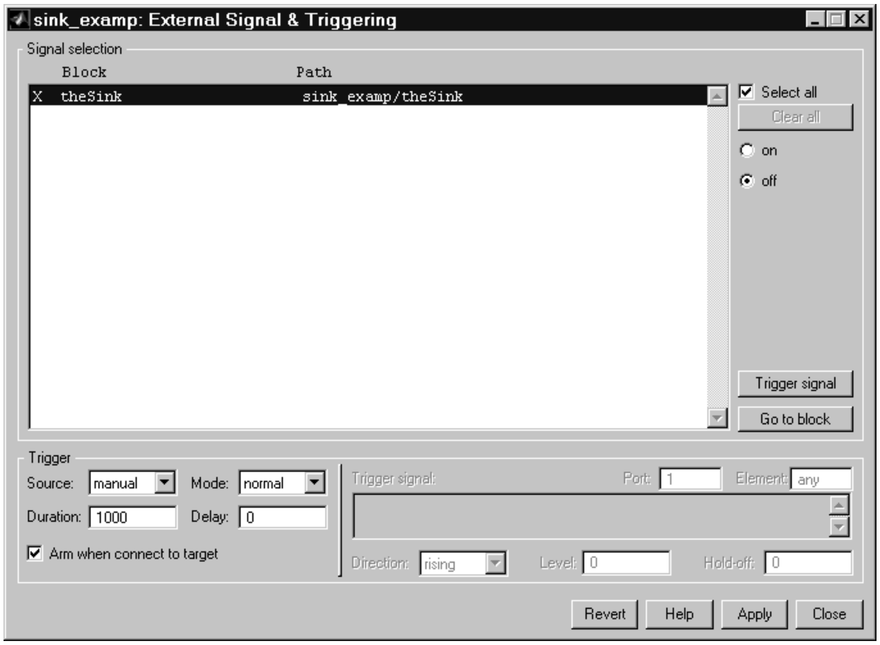
The model shown below, `sink_examp`, contains an atomic subsystem, `theSink`.



The subsystem `theSink`, shown below, applies a gain and an offset to its input signal, and displays it on a Scope block.



If theSink is declared as a Signal Viewing Subsystem, the generated target program includes only the code for the Sine Wave block. If theSink is selected and armed in the **External Signal and Triggering** dialog box (as shown in Figure 5-10), the target program uploads the sine wave signal to theSink during simulation.You can then modify the parameters of the blocks within theSink and observe their effect upon the uploaded signal.


sink_examp: External Signal & Triggering

Signal selection

	Block	Path
X	theSink	sink_examp/theSink

Select all [✓]
on [✓ off]

Trigger
Source: manual
Mode: normal
Duration: 1000
Delay: 0
Arm when connect to target [✓]

Trigger signal:
Port: 1
Element: any
Direction: rising
Level: 0
Hold-off: 0

Figure 5-10: Signal Viewing Subsystem Selected in External Signals & Triggering Dialog Box

Note that if `theSink` were not declared as a Signal Viewing Subsystem, its Gain, Constant, and Sum blocks would run as subsystem code on the target system. The Sine Wave signal would be uploaded to Simulink after being processed by these blocks, and viewed on `sink_examp/theSink/Scope2`. Processing demands on the target system would be increased by the additional signal processing, and by the downloading of block parameter changes from the host.

Overview of External Mode Communications

In external mode, Simulink does not simulate the system represented by the block diagram. When external mode is enabled, Simulink downloads current values of all parameters to the target system. After the initial download, Simulink remains in a waiting mode until you change parameters in the block diagram or until Simulink receives data from the target.

The Download Mechanism

When you change a parameter in the block diagram, Simulink calls the external interface MEX-file, passing new parameter values (along with other information) as arguments.

The external interface MEX-file contains code that implements one side of the interprocess communication (IPC) channel. This channel connects the Simulink process (where the MEX-file executes) to the process that is executing the external program. The MEX-file transfers the new parameter values via this channel to the external program.

The other side of the communication channel is implemented within the external program. This side writes the new parameter values into target's parameter structure (`rtP`).

The Simulink side initiates the parameter download operation by sending a message containing parameter information to the external program. In the terminology of client/server computing, the Simulink side is the client and the external program is the server. The two processes can be remote, or they can be local. Where the client and server are remote, a protocol such as TCP/IP is used to transfer data. Where the client and server are local, shared memory can be used to transfer data.

The following diagram illustrates this relationship.

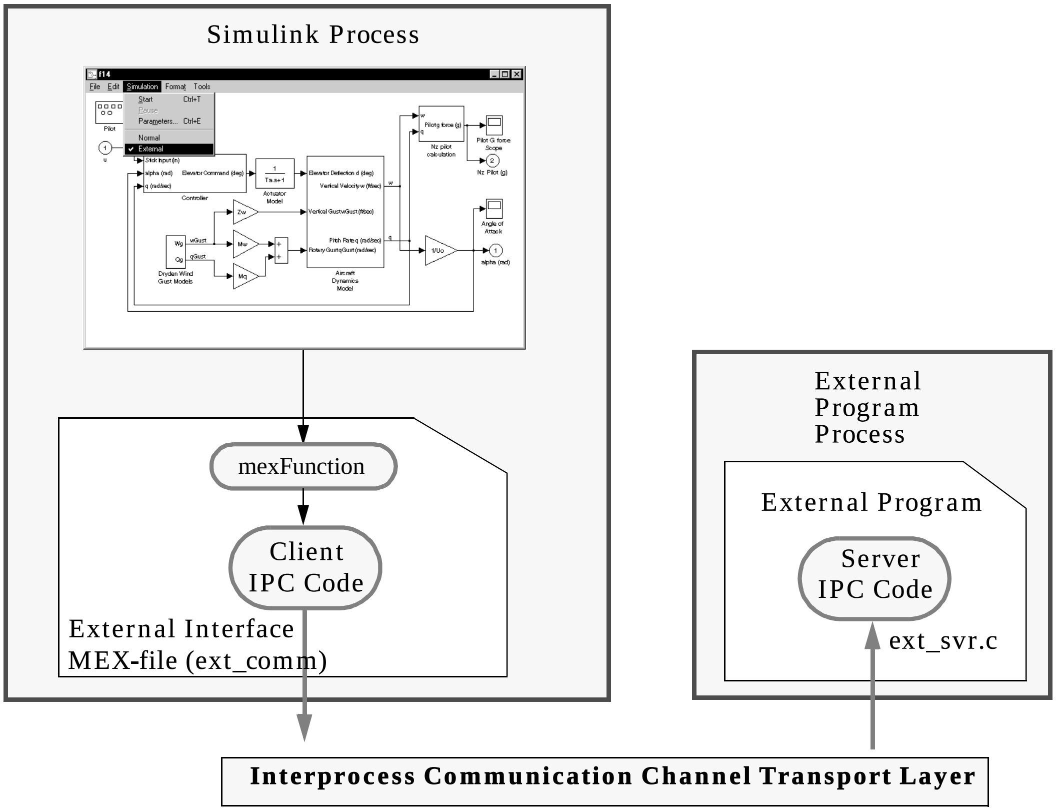


Figure 5-11: External Mode Architecture

Simulink calls the external interface MEX-file whenever you change parameters in the block diagram. The MEX-file then downloads the parameters to the external program via the communication channel.

The TCP/IP Implementation

Overview

The Real-Time Workshop provides code to implement both the client and server side based on TCP/IP. You can use the socket-based external mode implementation provided by the Real-Time Workshop with the generated code, provided that your target system supports TCP/IP.

A low-level *transport layer* handles physical transmission of messages. Both Simulink and the model code are independent of this layer. Both the transport layer and code directly interfacing to the transport layer are isolated in separate modules that format, transmit, and receive messages and data packets.

This design makes it possible for different targets to use different transport layers. For example, the `grt`, `grt_malloc`, and Tornado targets support host/target communication via TCP/IP, whereas the xPC target supports both RS232 (serial) and TCP/IP communication.

Using the TCP/IP Implementation

This section discusses how to use the TCP/IP-based client/server implementation of external mode with real-time programs on a UNIX or PC system. Chapter 12, “Targeting Tornado for Real-Time Applications” illustrates the use of external mode in the Tornado environment.

In order to use Simulink external mode, you must:

- Specify the name of the external interface MEX-file in the **External Target Interface** dialog box. By default, this is `ext_comm`.
- Configure the template makefile so that it links the proper source files for the TCP/IP server code and defines the necessary compiler flags when building the generated code.
- Build the external program.
- Run the external program.
- Set Simulink to external mode and connect to the target.

This figure shows the structure of the TCP/IP-based implementation.

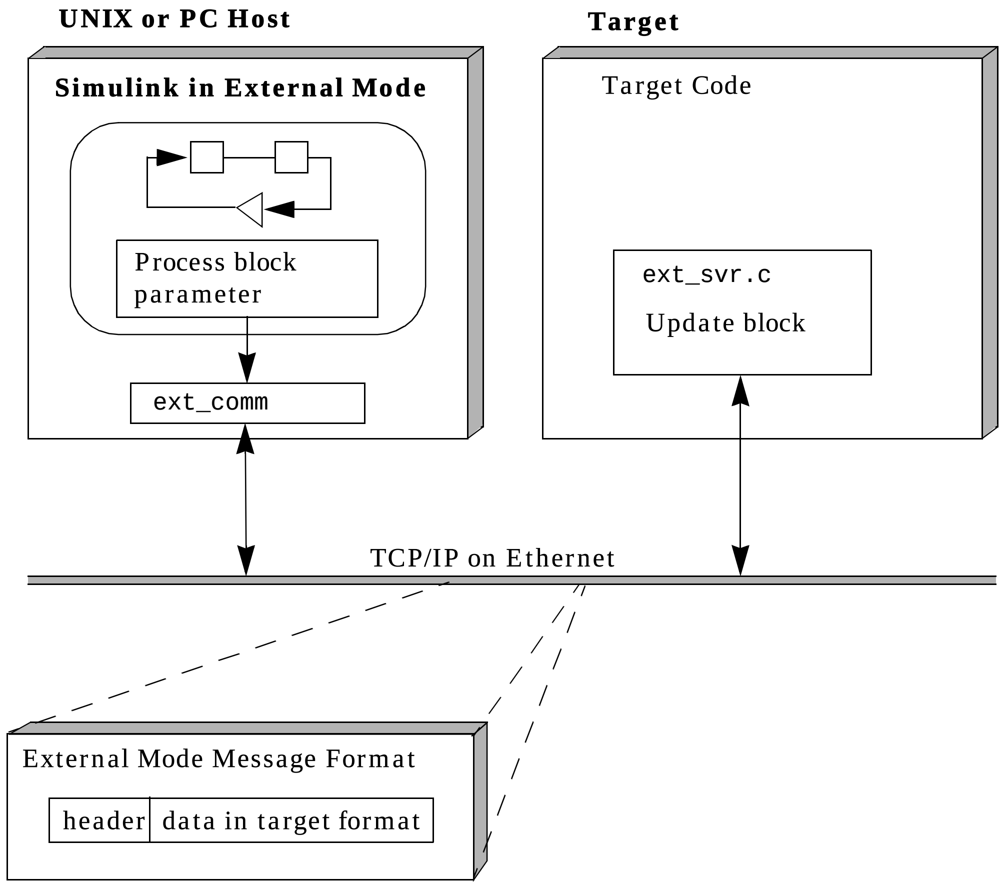
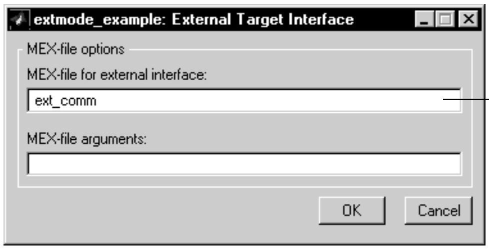


Figure 5-12: TCP/IP-Based Client/Server Implementation for External Mode

The following sections discuss the details of how to use the external mode of Simulink.

The External Interface MEX-File

You must specify the name of the external interface MEX-file in the **External Target Interface** dialog box.



extmode_example: External Target Interface
MEX-file options
MEX-file for external interface:
ext_comm
MEX-file arguments:
OK
Cancel

Enter the name of the external interface MEX-file in the box (you do not need to enter the `.mex` extension). This file must be in the current directory or in a directory that is on your MATLAB path.

The default external interface MEX-file is `ext_comm`. `ext_comm` implements TCP/IP-based communications. `ext_comm` has three optional arguments, discussed in the next section.

MEX-File Optional Arguments

In the **External Target Interface** dialog box, you can specify optional arguments that are passed to the MEX-file. These are:

- Target network name: the network name of the computer running the external program. By default, this is the computer on which Simulink is running. The name can be:
  - a string delimited by single quotes, such as `'myPuter'`
  - an IP address delimited by single quotes, such as `'148.27.151.12'`
- Verbosity level: controls the level of detail of the information displayed during the data transfer. The value is either `0` or `1` and has the following meaning:

  `0` — no information

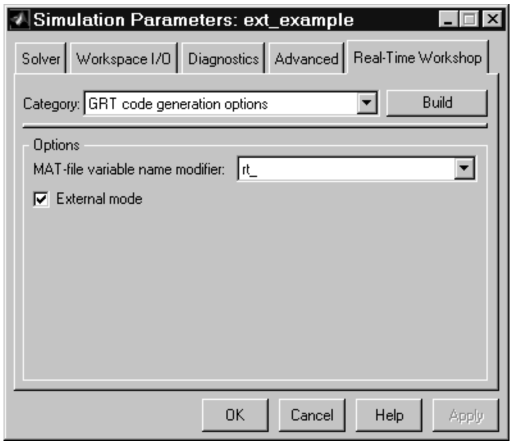
  `1` — detailed information
- TCP/IP server port number: The default value is `17725`. You can change the port number to a value between `256` and `65535` to avoid a port conflict if necessary.

You must specify these options in order. For example, if you want to specify the verbosity level (the second argument), then you must also specify the target host name (the first argument).

Note that you can specify command line options to the external program. See "Running the External Program" on page 5-41 for more information.

External Mode Compatible Targets

The GRT and Tornado targets support external mode. To enable external mode code generation, check **External mode** in the target-specific code generation options section of the Real-Time Workshop page. The following illustration shows the **GRT code generation options** with external mode enabled.



Category: GRT code generation options

MAT-file variable name modifier: rt_

☑ External mode

Running the External Program

The external program must be running before you can use Simulink in external mode. To run the external program, you type a command of the form

```
model -opt1 ... -optN
```

where *model* is the name of the external program and -*opt1 ... -optN* are options. (See "Command Line Options for the External Program" on page 5–42). In the examples in this section, we assume the name of the external program to be `ext_example`.

Running the External Program Under Windows

In the Windows environment, you can run the external programs in either of the following ways:

- Open an MS-DOS command prompt. At the command prompt, type the name of the target executable, followed by any options, as in the following example.

  ```
  ext_example -tf inf -w
  ```

- Alternatively, you can launch the target executable from the MATLAB command prompt. In this case the command must be preceded by an exclamation point (`!`) and followed by an ampersand (`&`) , as in the following example.

  ```
  !ext_example -tf inf -w &
  ```

Running the External Program Under UNIX

In the UNIX environment, you can run the external programs in either of the following ways:

- Open an an `Xterm` window. At the command prompt, type the name of the target executable, followed by any options, as in the following example.

  ```
  ext_example -tf inf -w
  ```

- Alternatively, you can launch the target executable from the MATLAB command prompt. In the UNIX environment, if you start the external program from MATLAB, you must run it in the background so that you can still access Simulink. The command must be preceded by an exclamation point (`!`) and followed by an ampersand (`&`) , as in the following example.

  ```
  !ext_example -tf inf -w &
  ```

  runs the executable from MATLAB by spawning another process to run it.

Command Line Options for the External Program

External mode target executables generated by the Real-Time Workshop support the following command line options:

- `-tf n` option

  The `-tf` option overrides the stop time set for the model in Simulink. The argument `n` specifies the number of seconds the program will run. The value `inf` value directs the model to run indefinitely. In this case, the model code will run until the target program receives a stop message from Simulink.

The following example sets the stop time to 10 seconds.

```
ext_example -tf 10
```

Note The `-tf` option is supported by the GRT and Tornado targets. If you are implementing a custom target and want to support the `-tf` option, you must implement the option yourself. See "Creating an External Mode Communication Channel" on page 17–73 for further information.

- `-w` option

  The `-w` option instructs the target program to enter a wait state until it receives a message from the host. At this point, the target is running, but not executing the model code. The start message is sent when you select **Start real-time code** from the **Simulation** menu or click the **Start real-time code** button in the **External Mode Control Panel**.

  Use the `-w` option if you want to view data from time step 0 of the target program execution, or if you want to modify parameters before the target program begins execution of model code.
- `-port n` option

  the `-port` option specifies the TCP/IP port number, `n`, for the target program. The port number of the target program must match that of the host. The default port number is `17725`. The port number must be a value between `256` and `65535`.

Note The `-w` and `-port` options are supported by the TCP/IP transport layer modules shipped with the Real-Time Workshop. By default, these modules are linked into external mode target executables. If you are implementing a custom external mode transport layer and want to support these options, you must implement them in your code. See "Creating an External Mode Communication Channel" on page 17–73 for further information. See *matlabroot*`/rtw/c/src/ext_transport.c` for example code.

Error Conditions

If the Simulink block diagram does not match the external program, Simulink displays an error box informing you that the checksums do not match (i.e., the model has changed since you generated code). This means you must rebuild the program from the new block diagram (or reload the correct one) in order to use external mode.

If the external program is not running, Simulink displays an error informing you that it cannot connect to the external program.

Implementing an External Mode Protocol Layer

If you want to implement your own transport layer for external mode communication, you must modify certain code modules provided by the Real-Time Workshop, and rebuild `ext_comm`, the external interface MEX-file. This advanced topic is described in detail in “Creating an External Mode Communication Channel” on page 17–73.

Limitations of External Mode

In general, you cannot change a parameter if doing so results in a change in the structure of the model. For example, you cannot change:

- The number of states, inputs, or outputs of any block
- The sample time or the number of sample times
- The integration algorithm for continuous systems
- The name of the model or of any block
- The parameters to the Fcn block

If you cause any of these changes to the block diagram, then you must rebuild the program with newly generated code.

However, parameters in transfer function and state space representation blocks *can* be changed in specific ways:

- The parameters (numerator and denominator polynomials) for the Transfer Fcn (continuous and discrete) and Discrete Filter blocks can be changed (as long as the number of states does not change).
- Zero entries in the State Space and Zero Pole (both continuous and discrete) blocks in the user-specified or computed parameters (i.e., the A, B, C, and D matrices obtained by a zero-pole to state-space transformation) cannot be changed once external simulation is started.
- In the State Space blocks, if you specify the matrices in the controllable canonical realization, then all changes to the A, B, C, D matrices that preserve this realization and the dimensions of the matrices are allowed.

Program Architecture

Introduction

The Real-Time Workshop generates two styles of code. One code style is suitable for rapid prototyping (and simulation via code generation). The other style is suitable for embedded applications. This chapter discusses the program architecture, that is, the structure of the Real-Time Workshop generated code, associated with these two styles of code. The table below classifies the targets shipped with the Real-Time Workshop.

Table 6-1: Code Styles Listed By Target

Target	Code Style (using C unless noted)
Real-Time Workshop Embedded Coder target	Embedded — useful as a starting point when using the generated C code in an embedded application.
Generic real-time (GRT) target	Rapid prototyping — nonreal-time simulation on your workstation. Useful as a starting point for creating a rapid prototyping real-time target that does not use real-time operating system tasking primitives. Also useful for validating the generated code on your workstation.
Real-time `malloc` target	Rapid prototyping — very similar to the generic real-time (GRT) target except that this target allocates all model working memory dynamically rather than statically declaring it in advance.
Rapid simulation target	Rapid prototyping — nonreal-time simulation of your model on your workstation. Useful as a high-speed or batch simulation tool.
S-function target	Rapid prototyping — creates a C-MEX S-function for simulation of your model within another Simulink model.

Table 6-1: Code Styles Listed By Target (Continued)

Target	Code Style (using C unless noted)
Tornado (VxWorks) real-time target	Rapid prototyping — runs model in real time using the VxWorks real-time operating system tasking primitives. Also useful as a starting point for targeting a real-time operating system.
Real-Time Windows target	Rapid prototyping — runs model in real-time at interrupt level while your PC is running Microsoft Windows in the background.
Ada simulation target	Embedded — nonreal-time simulation on your workstation using Ada. Useful for validating the generated code on your workstation.
Ada multitasking real-time target	Embedded — uses Ada tasking primitives to run your model in real time. Useful as a starting point when using the generated Ada code in an embedded application.
xPC target	Rapid prototyping — runs model in real time on target PC running xPC kernel.
DOS real-time target	Rapid prototyping — runs model in real time at interrupt level under DOS.

Third-party vendors supply additional targets for the Real-Time Workshop. Generally, these can be classified as rapid prototyping targets. For more information about third-party products, see the MATLAB Connections Web page: `http://www.mathworks.com/products/connections`.

You can identify the rapid prototyping style of generated code by its use of the `SimStruct` data structure (i.e., `#include "simstruc.h"`). In contrast, the embedded code style does not have a `SimStruct`.

This chapter is divided into three sections. The first section discusses model execution; the second section discusses the rapid prototyping style of code; and the third section discusses the embedded style of code.

Model Execution

Before looking at the two styles of generated code, you need to have a high-level understanding of how the generated model code is executed. The Real-Time Workshop generates algorithmic code as defined by your model. You may include your own code into your model via S-functions. S-functions can range from high-level signal manipulation algorithms to low-level device drivers.

The Real-Time Workshop also provides a run-time interface that executes the generated model code. The run-time interface and model code are compiled together to create the model executable. The diagram below shows a high-level object-oriented view of the executable.

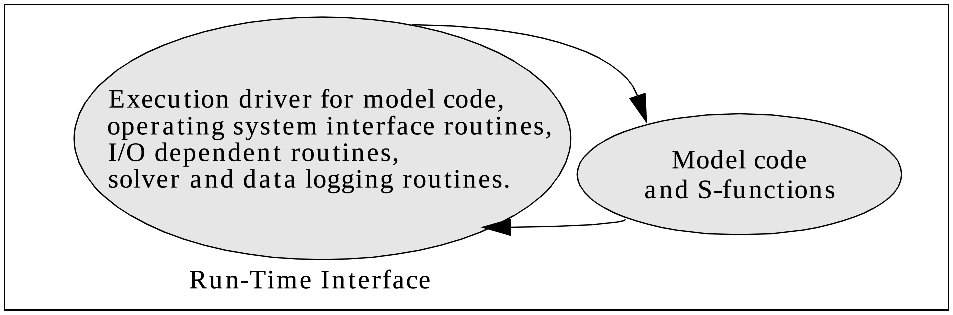

Execution driver for model code,
operating system interface routines,
I/O dependent routines,
solver and data logging routines.
Model code and S-functions
Run-Time Interface

Figure 6-1: The Object-Oriented View of a Real-Time Program

In general, the conceptual design of the model execution driver does not change between the rapid prototyping and embedded style of generated code. The following sections describe model execution for singletasking and multitasking environments both for simulation (nonreal-time) and for real-time. For most models, the multitasking environment will provide the most efficient model execution (i.e., fastest sample rate).

The following concepts are useful in describing how models execute:

- `Initialization` — Initializing the run-time interface code and the model code.
- `ModelOutputs` — Calling all blocks in your model that have a time hit at the current point in time and having them produce their output. `ModelOutputs` can be done in major or minor time steps. In major time steps, the output is

a given simulation time step. In minor time steps, the run-time interface integrates the derivatives to update the continuous states.

- `ModelUpdate` — Calling all blocks in your model that have a sample hit at the current point in time and having them update their discrete states or similar type objects.
- `ModelDerivatives` — Calling all blocks in your model that have continuous states and having them update their derivatives. `ModelDerivatives` is only called in minor time steps.

The pseudocode below shows the execution of a model for a singletasking simulation (nonreal-time).

```
main()
{
  Initialization
  While (time < final time)
    ModelOutputs     -- Major time step.
    LogTXY           -- Log time, states and root outports.
    ModelUpdate      -- Major time step.
    Integrate:       -- Integration in minor time step for models
                     -- with continuous states.
      ModelDerivatives
      Do 0 or more:
        ModelOutputs
        ModelDerivatives
      EndDo (Number of iterations depends upon the solver.)
      Integrate derivatives to update continuous states.
    EndIntegrate
  EndWhile
  Shutdown
}
```

The initialization phase begins first. This consists of initializing model states and setting up the execution engine. The model then executes, one step at a time. First `ModelOutputs` executes at time *t*, then the workspace I/O data is logged, and then `ModelUpdate` updates the discrete states. Next, if your model has any continuous states, `ModelDerivatives` integrates the continuous states' derivatives to generate the states for time $t_{new} = t + h$, where *h* is the step size. Time then moves forward to t_{new} and the process repeats.

During the `ModelOutputs` and `ModelUpdate` phases of model execution, only blocks that have hit the current point in time execute. They determine if they have hit by using a macro (`ssIsSampleHit`, or `ssIsSpecialSampleHit`) that checks for a sample hit.

The pseudocode below shows the execution of a model for a multitasking simulation (nonreal-time).

```
main()
{
  Initialization
  While (time < final time)
    ModelOutputs(tid=0)   -- Major time step.
    LogTXY                -- Log time, states, and root outports.
    ModelUpdate(tid=1)    -- Major time step.
    For i=1:NumTids
      ModelOutputs(tid=i) -- Major time step.
      ModelUpdate(tid=i)  -- Major time step.
    EndFor
    Integrate       -- Integration in minor time step for models
                    -- with continuous states.
      ModelDerivatives
      Do 0 or more:
        ModelOutputs(tid=0)
        ModelDerivatives
      EndDo (Number of iterations depends upon the solver.)
      Integrate derivatives to update continuous states.
    EndIntegrate
  EndWhile
  Shutdown
  }
```

The multitasking operation is more complex when compared with the singletasking execution because the output and update functions are subdivided by the *task identifier* (`tid`) that is passed into these functions. This allows for multiple invocations of these functions with different task identifiers using overlapped interrupts, or for multiple tasks when using a real-time operating system. In simulation, multiple tasks are emulated by executing the code in the order that would occur if there were no preemption in a real-time system.

Note that the multitasking execution assumes that all tasks are multiples of the base rate. Simulink enforces this when you have created a fixed-step multitasking model.

The multitasking execution loop is very similar to that of singletasking, except for the use of the task identifier (`tid`) argument to `ModelOutputs` and `ModelUpdate`. The `ssIsSampleHit` or `ssIsSpecialSampleHit` macros use the `tid` to determine when blocks have a hit. For example, `ModelOutputs (tid=5)` will execute only the blocks that have a sample time corresponding to task identifier 5.

The pseudocode below shows the execution of a model in a real-time singletasking system where the model is run at interrupt level.

```
rtOneStep()
{
  Check for interrupt overflow
  Enable "rtOneStep" interrupt
  ModelOutputs    -- Major time step.
  LogTXY          -- Log time, states and root outports.
  ModelUpdate     -- Major time step.
  Integrate       -- Integration in minor time step for models
                  -- with continuous states.
     ModelDerivatives
     Do 0 or more
       ModelOutputs
       ModelDerivatives
     EndDo (Number of iterations depends upon the solver.)
     Integrate derivatives to update continuous states.
  EndIntegrate
}

main()
{
  Initialization (including installation of rtOneStep as an
    interrupt service routine, ISR, for a real-time clock).
  While(time < final time)
    Background task.
  EndWhile
  Mask interrupts (Disable rtOneStep from executing.)
  Complete any background tasks.
```

```
    Shutdown
  }
```

Real-time singletasking execution is very similar to the nonreal-time single tasking execution, except that the execution of the model code is done at interrupt level.

At the interval specified by the program's base sample rate, the interrupt service routine (ISR) preempts the background task to execute the model code. The base sample rate is the fastest rate in the model. If the model has continuous blocks, then the integration step size determines the base sample rate.

For example, if the model code is a controller operating at 100 Hz, then every 0.01 seconds the background task is interrupted. During this interrupt, the controller reads its inputs from the analog-to-digital converter (ADC), calculates its outputs, writes these outputs to the digital-to-analog converter (DAC), and updates its states. Program control then returns to the background task. All of these steps must occur before the next interrupt.

The following pseudocode shows how a model executes in a real-time multitasking system (where the model is run at interrupt level).

```
  rtOneStep()
  {
    Check for interrupt overflow
    Enable "rtOneStep" interrupt
    ModelOutputs(tid=0)     -- Major time step.
    LogTXY                  -- Log time, states and root outports.
    ModelUpdate(tid=0)      -- Major time step.
    Integrate               -- Integration in minor time step for
                            -- models with continuous states.
       ModelDerivatives
       Do 0 or more:
         ModelOutputs(tid=0)
         ModelDerivatives
       EndDo (Number of iterations depends upon the solver.)
       Integrate derivatives and update continuous states.
    EndIntegrate
    For i=1:NumTasks
      If (hit in task i)
        ModelOutputs(tid=i)
        ModelUpdate(tid=i)
      EndIf
    EndFor
  }

  main()
  {
    Initialization (including installation of rtOneStep as an
      interrupt service routine, ISR, for a real-time clock).
    While(time < final time)
      Background task.
    EndWhile
    Mask interrupts (Disable rtOneStep from executing.)
    Complete any background tasks.
    Shutdown
  }
```

Running models at interrupt level in real-time multitasking environment is very similar to the previous singletasking environment, except that overlapped interrupts are employed for concurrent execution of the tasks.

The execution of a model in a singletasking or multitasking environment when using real-time operating system tasking primitives is very similar to the interrupt-level examples discussed above. The pseudocode below is for a singletasking model using real-time tasking primitives.

```
tSingleRate()
{
  MainLoop:
    If clockSem already "given", then error out due to overflow.
    Wait on clockSem
    ModelOutputs            -- Major time step.
    LogTXY                  -- Log time, states and root outports
    ModelUpdate             -- Major time step
    Integrate               -- Integration in minor time step for
                            -- models with continuous states.
      ModelDeriviatives
      Do 0 or more:
        ModelOutputs
        ModelDerivatives
      EndDo (Number of iterations depends upon the solver.)
      Integrate derivatives to update continuous states.
    EndIntegrate
  EndMainLoop
}

main()
{
  Initialization
  Start/spawn task "tSingleRate".
  Start clock that does a "semGive" on a clockSem semaphore.
  Wait on "model-running" semaphore.
  Shutdown
}
```

In this singletasking environment, the model is executed using real-time operating system tasking primitives. In this environment, we create a single task (`tSingleRate`) to run the model code. This task is invoked when a clock tick occurs. The clock tick gives a `clockSem` (clock semaphore) to the model task (`tSingleRate`). The model task will wait for the semaphore before executing. The clock ticks are configured to occur at the fundamental step size (base rate) for your model.

The pseudocode below is for a multitasking model using real-time tasking primitives.

```
tSubRate(subTaskSem,i)
{
  Loop:
    Wait on semaphore subTaskSem.
    ModelOutputs(tid=i)
    ModelUpdate(tid=i)
  EndLoop
}

tBaseRate()
{
  MainLoop:
    If clockSem already "given", then error out due to overflow.
    Wait on clockSem
    For i=1:NumTasks
      If (hit in task i)
```

```
            If task i is currently executing, then error out due to
              overflow.
            Do a "semGive" on subTaskSem for task i.
          EndIf
        EndFor
        ModelOutputs(tid=0)    -- major time step.
        LogTXY                 -- Log time, states and root outports.
        ModelUpdate(tid=0)     -- major time step.
        Loop:                  -- Integration in minor time step for
                               -- models with continuous states.
          ModelDeriviatives
          Do 0 or more:
            ModelOutputs(tid=0)
            ModelDerivatives
          EndDo (number of iterations depends upon the solver).
          Integrate derivatives to update continuous states.
        EndLoop
      EndMainLoop
    }

    main()
    {
      Initialization
      Start/spawn task "tSingleRate".
      Start clock that does a "semGive" on a clockSem semaphore.
      Wait on "model-running" semaphore.
      Shutdown
    }
```

In this multitasking environment, the model is executed using real-time operating system tasking primitives. In this environment, it is necessary to create several model tasks (tBaseRate and several tSubRate tasks) to run the model code. The base rate task (tBaseRate) has a higher priority than the subrate tasks. The subrate task for tid=1 has a higher priority than the subrate task for tid=2, and so on. The base rate task is invoked when a clock tick occurs. The clock tick gives a clockSem to tBaseRate. The first thing tBaseRate does is give semaphores to the subtasks that have a hit at the current point in time. Since the base rate task has a higher priority, it continues to execute. Next it executes the fastest task (tid=0) consisting of blocks in your model that have the fastest sample time. After this execution, it resumes waiting for the clock semaphore. The clock ticks are configured to occur at executing at the fundamental step size for your model.

Program Timing

Real-time programs require careful timing of the task invocations (either via an interrupt or a real-time operating system tasking primitive) to ensure that the model code executes to completion before another task invocation occurs. This includes time to read and write data to and from external hardware.

The following diagram illustrates interrupt timing.

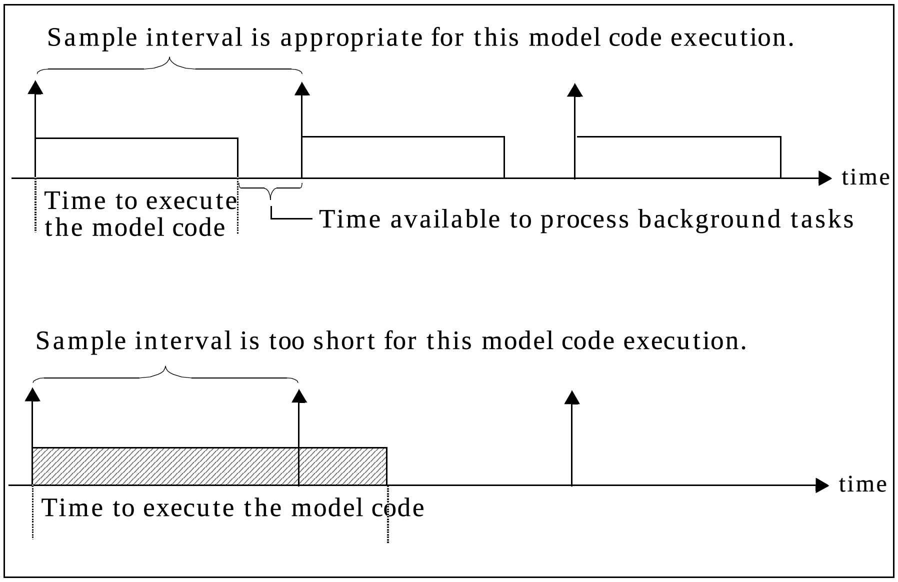

Sample interval is appropriate for this model code execution.
time
Time to execute the model code
Time available to process background tasks
Sample interval is too short for this model code execution.
time
Time to execute the model code

Figure 6-2: Task Timing

The sample interval must be long enough to allow model code execution between task invocations.

In the figure above, the time between two adjacent vertical arrows is the sample interval. The empty boxes in the upper diagram show an example of a program that can complete one step within the interval and still allow time for the background task. The gray box in the lower diagram indicates what happens if the sample interval is too short. Another task invocation occurs before the task is complete. Such timing results in an execution error.

Note also that, if the real-time program is designed to run forever (i.e., the final time is 0 or infinite so the while loop never exits), then the shutdown code never executes.

Program Execution

As the previous section indicates, a real-time program may not require 100% of the CPU's time. This provides an opportunity to run background tasks during the free time.

Background tasks include operations like writing data to a buffer or file, allowing access to program data by third-party data monitoring tools, or using Simulink external mode to update program parameters.

It is important, however, that the program be able to preempt the background task at the appropriate time to ensure real-time execution of the model code.

The way the program manages tasks depends on capabilities of the environment in which it operates.

External Mode Communication

External mode allows communication between the Simulink block diagram and the stand-alone program that is built from the generated code. In this mode, the real-time program functions as an interprocess communication server, responding to requests from Simulink.

Data Logging In Single- and Multitasking Model Execution

The Real-Time Workshop data-logging features, described in "Workspace I/O Options and Data Logging" in Chapter 3, enable you to save system states, outputs, and time to a MAT-file at the completion of the model execution. The `LogTXY` function, which performs data logging, operates differently in singletasking and multitasking environments.

If you examine how `LogTXY` is called in the singletasking and multitasking environments, you will notice that for singletasking `LogTXY` is called after `ModelOutputs`. During this `ModelOutputs` call, all blocks that have a hit at time *t* are executed, whereas in multitasking, `LogTXY` is called after

`ModelOutputs(tid=0)` that executes only the blocks that have a hit at time *t* and that have a task identifier of 0. This results in differences in the logged values between singletasking and multitasking logging. Specifically, consider a model with two sample times, the faster sample time having a period of 1.0 second and the slower sample time having a period of 10.0 seconds. At time t = k*10, k=0,1,2... both the fast (`tid=0`) and slow (`tid=1`) blocks have a hit. When executing in multitasking mode, when `LogTXY` is called, the slow blocks will have a hit, but the previous value will be logged, whereas in singletasking the current value will be logged.

Another difference occurs when logging data in an enabled subsystem. Consider an enabled subsystem that has a slow signal driving the enable port and fast blocks within the enabled subsystem. In this case, the evaluation of the enable signal occurs in a slow task and the fast blocks will see a delay of one sample period, thus the logged values will show these differences.

To summarize differences in logged data between singletasking and multitasking, differences will be seen when:

- Any root outport block has a sample time that is slower than the fastest sample time
- Any block with states has a sample time that is slower than the fastest sample time
- Any block in an enabled subsystem where the signal driving the enable port is slower than the rate of the blocks in the enabled subsystem

For the first two cases, even though the logged values are different between singletasking and multitasking, the model results are not different. The only real difference is where (at what point in time) the logging is done. The third (enabled subsystem) case results in a delay that can be seen in a real-time environment.

Rapid Prototyping and Embedded Model Execution Differences

The rapid prototyping program framework provides a common application programming interface (API) that does not change between model definitions.

The Real-Time Workshop Embedded Coder provides a different framework that we will refer to as the embedded program framework. The embedded program framework provides a optimized API that is tailored to your model. It is intended that when you use the embedded style of generated code, you are modeling how you would like your code to execute in your embedded system. Therefore, the definitions defined in your model should be specific to your embedded targets. Items such as the model name, parameter, and signal storage class are included as part of the API for the embedded style of code.

The single largest difference between the rapid prototyping and embedded style of generated code is that the embedded code does not contain the `SimStruct` data structure. The `SimStruct` defines a common API that the rapid prototyping style of generated code relies heavily on. However, the `SimStruct` data structure supports many options and therefore consumes memory that is not needed in an embedded application.

Another major difference between the rapid prototyping and embedded style of generated code is that the latter contains fewer entry-point functions. The embedded style of code can be configured to have only one run-time function *model*`_step`. You can define a single run-time function because the embedded target:

- Can only be used with models that do not have continuous sample time (and therefore no continuous states)
- Requires that all S-functions must be inlined with the Target Language Compiler, which means that they do not access the `SimStruct` data structure

Thus, when looking at the model execution pseudocode presented earlier in this chapter, you can eliminate the `Loop...EndLoop` statements, and group the `ModelOutputs`, `LogTXY`, and `ModelUpdate` into a single statement, *model*`_step`.

For a detailed discussion of how generated embedded code executes, see “Program Execution” on page 9-9.

Rapid Prototyping Model Functions

The rapid prototyping code defines the following functions that interface with the run-time interface:

- `Model()` — The model registration function. This function for initializes the work areas (e.g., allocating and setting pointers to various data structures) needed by the model. The model registration function calls the `MdlInitializeSizes` and `MdlInitializeSampleTimes` functions. These two functions are very similar to the S-function `mdlInitializeSizes` and `mdlInitializeSampleTimes` methods.

- `MdlStart(void)` — After the model registration functions, `MdlInitializeSizes` and `MdlInitializeSampleTimes` execute, the run-time interface starts execution by calling `MdlStart`. This routine is called once at startup.

  The function `MdlStart` has four basic sections:

  - Code to initialize the states for each block in the root model that has states. A subroutine call is made to the "initialize states" routine of conditionally executed subsystems.
  - Code generated by the one-time initialization (start) function for each block in the model.
  - Code to enable the blocks in the root model that have enable methods, and the blocks inside triggered or function-call subsystems residing in the root model. Simulink blocks can have enable and disable methods. An enable method is called just before a block starts executing, and the disable method is called just after the block stops executing.
  - Code for each block in the model that has a constant sample time.
- `MdlOutputs(int_T tid)` — `MdlOutputs` updates the output of blocks at appropriate times. The `tid` (task identifier) parameter identifies the task that in turn maps when to execute blocks based upon their sample time. This routine is invoked by the run-time interface during major and minor time steps. The major time steps are when the run-time interface is taking an actual time step (i.e., it is time to execute a specific task). If your model contains continuous states, the minor time steps will be taken. The minor time steps are when the solver is generating integration stages, which are points between major outputs. These integration stages are used to compute the derivatives used in advancing the continuous states.
- `MdlUpdate(int_T tid)` — `MdlUpdate` updates the discrete states and work vector state information (i.e., states that are neither continuous nor discrete) saved in work vectors. The `tid` (task identifier) parameter identifies the task that in turn indicates which sample times are active allowing you to conditionally update states of only active blocks. This routine is invoked by the run-time interface after the major `MdlOutputs` has been executed.
- `MdlDerivatives(void)` — `MdlDerivatives` returns the block derivatives. This routine is called in minor steps by the solver during its integration stages. All blocks that have continuous states have an identical number of

derivatives. These blocks are required to compute the derivatives so that the solvers can integrate the states.

• `MdlTerminate(void)` — `MdlTerminate` contains any block shutdown code. `MdlTerminate` is called by the run-time interface, as part of the termination of the real-time program.

The contents of the above functions are directly related to the blocks in your model. A Simulink block can be generalized to the following set of equations.

$$y = f_0(t, x_c, x_d, u)$$

Output, y, is a function of continuous state, x_c, discrete state, x_d, and input, u. Each block writes its specific equation in the appropriate section of `MdlOutput`.

$$x_{d+1} = f_u(t, x_d, u)$$

The discrete states, x_d, are a function of the current state and input. Each block that has a discrete state updates its state in `MdlUpdate`.

$$\dot{x} = f_d(t, x_c, u)$$

The derivatives, x, are a function of the current input. Each block that has continuous states provides its derivatives to the solver (e.g., `ode5`) in `MdlDerivatives`. The derivatives are used by the solver to integrate the continuous state to produce the next value.

The output, y, is generally written to the block I/O structure. Root-level Outport blocks write to the external outputs structure. The continuous and discrete states are stored in the states structure. The input, u, can originate from another block's output, which is located in the block I/O structure, an external input (located in the external inputs structure), or a state. These structures are defined in the *model*`.h` file that the Real-Time Workshop generates.

Figure 6-3 shows the general content of the rapid prototyping style of C code.

```
/*
 * Version, Model options, TLC options,
 * and code generation information are placed here.
*/
<includes>
void MdlStart(void)
{
  /*
   * State initialization code.
   * Model start-up code - one time initialization code.
   * Execute any block enable methods.
   * Initialize output of any blocks with constant sample times.
   */
}

void MdlOutputs(int_T tid)
{
  /* Compute: y = f0(t,xc,xd,u) for each block as needed. */
}

void MdlUpdate(int_T tid)
{
  /* Compute: xd+1 = fu(t,xd,u) for each block as needed. */
}

void MdlDerivatives(void)
{
  /* Compute: dxc = fd(t,xc,u) for each block as needed. */
}

void MdlTerminate(void)
{
  /* Perform shutdown code for any blocks that
     have a termination action */
}
```

Figure 6-3: Content of model.c for the Rapid Prototyping Code Style

Figure 6-4 shows a flow chart describing the execution of the rapid prototyping generated code.

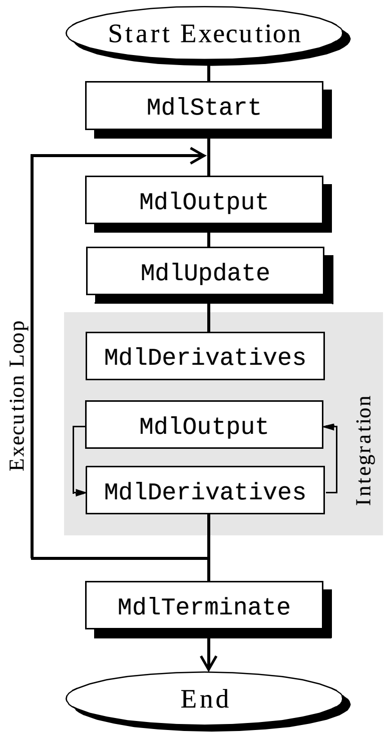


Figure 6-4: Rapid Prototyping Execution Flow Chart

Each block places code into specific `Mdl` routines according to the algorithm that it is implementing. Blocks have input, output, parameters, and states, as well as other general items. For example, in general, block inputs and outputs are written to a block I/O structure (`rtB`). Block inputs can also come from the external input structure (`rtU`) or the state structure when connected to a state port of an integrator (`rtX`), or ground (`rtGround`) if unconnected or grounded.

Block outputs can also go to the external output structure (`rtY`). The following figure shows the general mapping between these items.

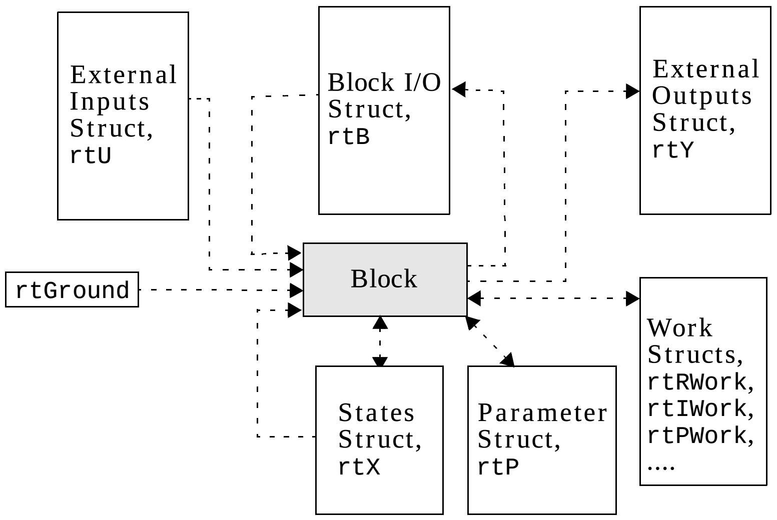

External Inputs Struct, rtU
Block I/O Struct, rtB
External Outputs Struct, rtY
rtGround
Block
Work Structs, rtRWork, rtIWork, rtPWork, ...
States Struct, rtX
Parameter Struct, rtP

Figure 6-5: Data View of the Generated Code

Structure definitions:

- Block I/O Structure (`rtB`) — This structure consists of all block output signals. The number of block output signals is the sum of the widths of the data output ports of all nonvirtual blocks in your model. If you activate block I/O optimizations, Simulink and the Real-Time Workshop reduce the size of the `rtB` structure by:
  - Reusing the entries in the `rtB` structure
  - Making other entries local variables

  See "Signals: Storage, Optimization, and Interfacing" on page 3–65 for further information on these optimizations.

  Structure field names are determined by either the block's output signal name (when present) or by the block name and port number when the output signal is left unlabeled.
- Block States Structure (`rtX`) — The states structure contains the continuous and discrete state information for any blocks in your model that have states.

The states structure has two sections: the first is for the continuous states; the second is for the discrete states.

- Block Parameters Structure (`rtP`) — The parameters structure contains all block parameters that can be changed during execution (e.g., the parameter of a Gain block).
- External Inputs Structure (`rtU`) —The external inputs structure consists of all root-level Inport block signals. Field names are determined by either the block's output signal name, when present, or by the Inport block's name when the output signal is left unlabeled.
- External Outputs Structure (`rtY`) —The external outputs structure consists of all root-level Outport blocks. Field names are determined by the root-level Outport block names in your model.
- Real Work, Integer Work, and Pointer Work Structures (`rtRWork`, `rtIWork`, `rtPWork`) — Blocks may have a need for real, integer, or pointer work areas. For example, the Memory block uses a real work element for each signal. These areas are used to save internal states or similar information.

Embedded Model Functions

The Real-Time Workshop Embedded Coder and Ada Coder targets generate the following functions:

- *model*`_intialize` — Performs all model initialization and should be called once before you start executing your model.
- If the **Single output/update function** code generation option is selected, then you will see:
  - *model*`_step(int_T tid)` — Contains the output and update code for all blocks in your model.

  Otherwise you will see:
  - *model*`_output(int_T tid)` — Contains the output code for all blocks in your model.
  - *model*`_update(int_T tid)` — This contains the update code for all blocks in your model.

- If the **Terminate function required** code generation option is selected, then you will see:
  - *model*_terminate — This contains all model shutdown code and should be called as part of system shutdown.

See “Code Modules” on page 9–5 and “Program Execution” on page 9–9 for complete descriptions of these functions in the context of the Real-Time Workshop Embedded Coder.

Rapid Prototyping Program Framework

The code modules generated from a a Simulink model — *model*.c, *model*.h, and other files — implement the model's system equations, contain block parameters, and perform initialization.

The Real-Time Workshop's program framework provides the additional source code necessary to build the model code into a complete, stand-alone program. The program framework consists of *application modules* (files containing source code to implement required functions) designed for a number of different programming environments.

The automatic program builder ensures the program is created with the proper modules once you have configured your template makefile.

The application modules and the code generated for a Simulink model are implemented using a common API. This API defines a data structure (called a `SimStruct`) that encapsulates all data for your model. Note that the Real-Time Workshop Embedded Coder target does not have a `SimStruct`, but does have a common calling syntax for model execution.

This API is similar to that of S-functions, with one major exception: the API assumes that there is only one instance of the model, whereas S-functions can have multiple instances. The function prototypes also differ from S-functions.

Rapid Prototyping Program Architecture

The structure of a real-time program consists of three components. Each component has a dependency on a different part of the environment in which the program executes. The following diagram illustrates this structure.

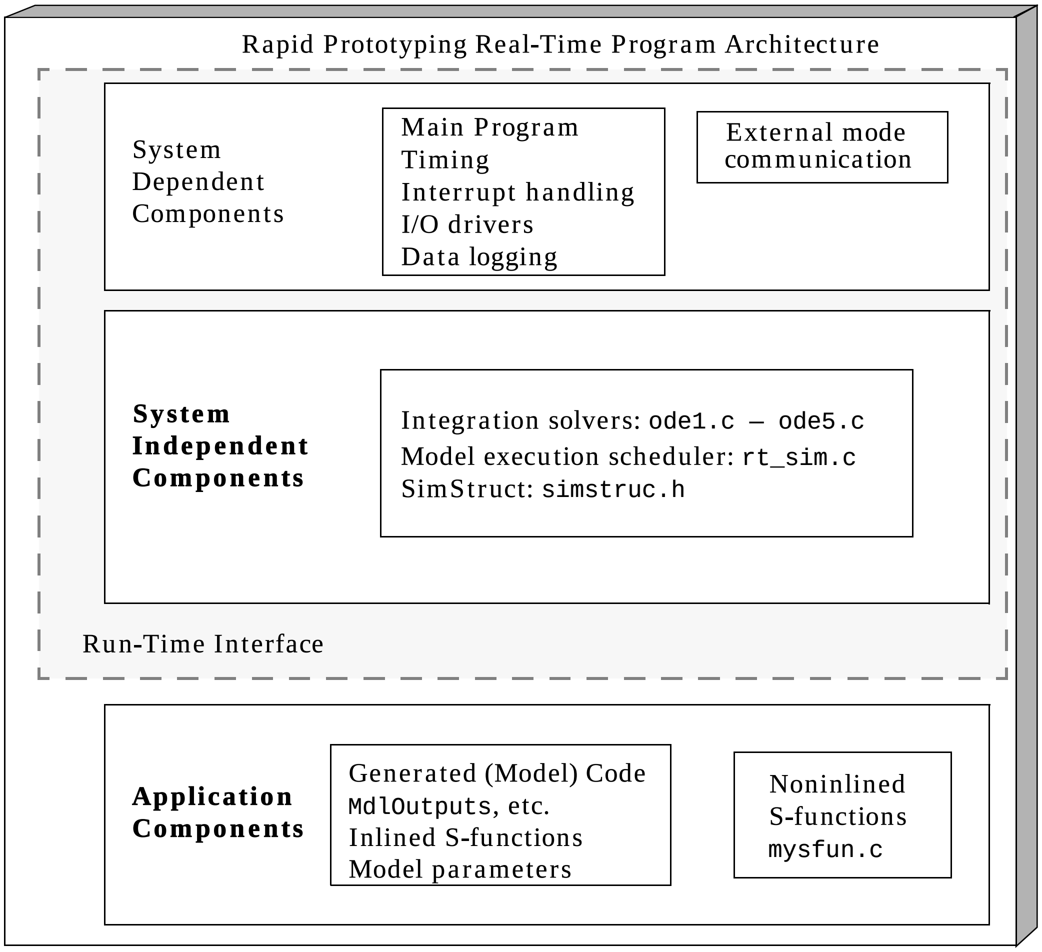


Figure 6-6: The Rapid Prototyping Program Architecture

The Real-Time Workshop architecture consists of three parts. The first two components, system dependent and independent, together form the *run-time interface.*

This architecture readily adapts to a wide variety of environments by isolating the dependencies of each program component. The following sections discuss each component in more detail and include descriptions of the application modules that implement the functions carried out by the system dependent, system independent, and application components.

Rapid Prototyping System Dependent Components

These components contain the program's main function, which controls program timing, creates tasks, installs interrupt handlers, enables data logging, and performs error checking.

The way in which application modules implement these operations depends on the type of computer. This means that, for example, the components used for a DOS-based program perform the same operations, but differ in method of implementation from components designed to run under Tornado on a VME target.

The main Function

The `main` function in a C program is the point where execution begins. In Real-Time Workshop application programs, the `main` function must perform certain operations. These operations can be grouped into three categories: initialization, model execution, and program termination.

Initialization

- Initialize special numeric parameters: `rtInf`, `rtMinusInf`, and `rtNaN`. These are variables that the model code can use.
- Call the model registration function to get a pointer to the `SimStruct`. The model registration function has the same name as your model. It is responsible for initializing `SimStruct` fields and any S-functions in your model.
- Initialize the model size information in the `SimStruct`. This is done by calling `MdlInitializeSizes`.
- Initialize a vector of sample times and offsets (for systems with multiple sample rates). This is done by calling `MdlInitializeSampleTimes`.

- Get the model ready for execution by calling `MdlStart`, which initializes states and similar items.
- Set up the timer to control execution of the model.
- Define background tasks and enable data logging, if selected.

Model Execution

- Execute a background task, for example, communicate with the host during external mode simulation or introduce a wait state until the next sample interval.
- Execute model (initiated by interrupt).
- Log data to buffer (if data logging is used).
- Return from interrupt.

Program Termination

- Call a function to terminate the program if it is designed to run for a finite time — destroy the `SimStruct`, deallocate memory, and write data to a file.

Rapid Prototyping Application Modules for System Dependent Components

The application modules contained in the system dependent components generally include a main module such as `rt_main.c` containing the main entry point for C. There may also be additional application modules for such things as I/O support and timer handling.

Rapid Prototyping System Independent Components

These components are collectively called system independent because all environments use the same application modules to implement these operations. This section steps through the model code (and if the model has continuous states, calls one of the numerical integration routines). This section also includes the code that defines, creates, and destroys the Simulink data structure (`SimStruct`). The model code and all S-functions included in the program define their own `SimStruct`.

The model code execution driver calls the functions in the model code to compute the model outputs, update the discrete states, integrate the continuous states (if applicable), and update time. These functions then write their calculated data to the `SimStruct`.

Model Execution

At each sample interval, the main program passes control to the model execution function, which executes one step though the model. This step reads inputs from the external hardware, calculates the model outputs, writes outputs to the external hardware, and then updates the states.

The following diagram illustrates these steps.

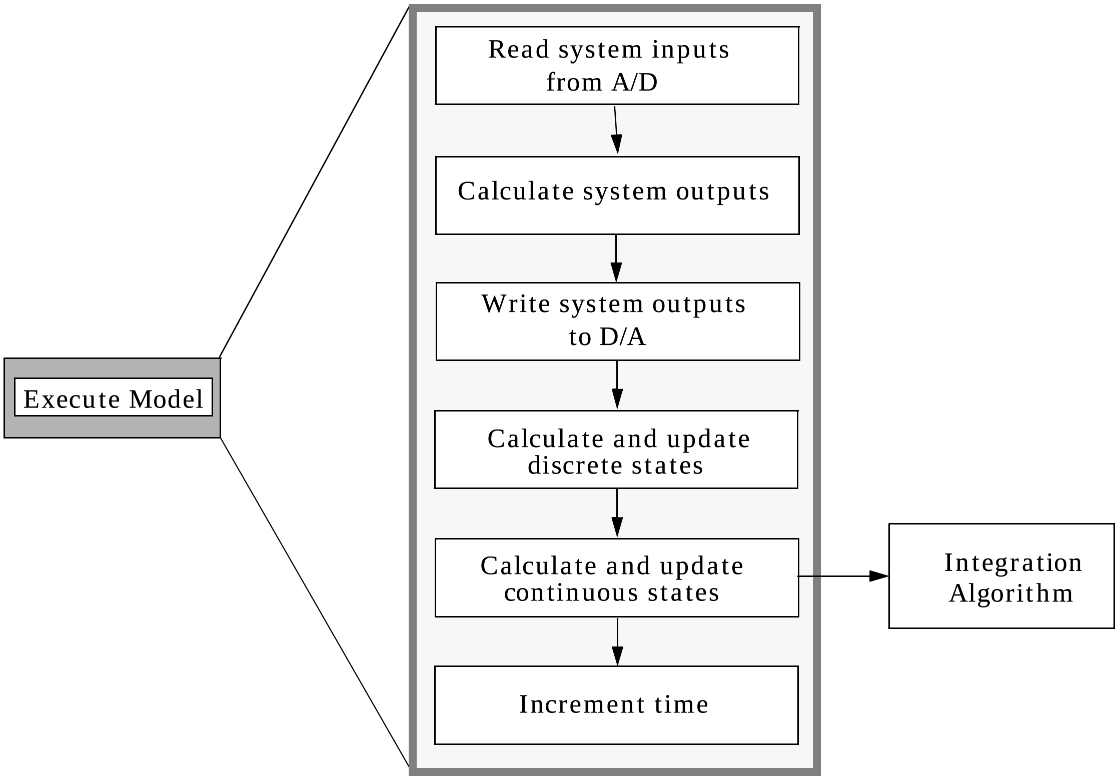

Execute Model
Read system inputs from A/D
Calculate system outputs
Write system outputs to D/A
Calculate and update discrete states
Calculate and update continuous states
Increment time
Integration Algorithm

Figure 6-7: Executing the Model

Note that this scheme writes the system outputs to the hardware before the states are updated. Separating the state update from the output calculation minimizes the time between the input and output operations.

Integration of Continuous States

The real-time program calculates the next values for the continuous states based on the derivative vector, dx/dt, for the current values of the inputs and the state vector.

These derivatives are then used to calculate the next value of the states using a state-update equation. This is the state-update equation for the first order Euler method (`ode1`)

$$x = x + \frac{dx}{dt}h$$

where h is the step size of the simulation, x represents the state vector, and dx/dt is the vector of derivatives. Other algorithms may make several calls to the output and derivative routines to produce more accurate estimates.

Note, however, that real-time programs use a fixed-step size since it is necessary to guarantee the completion of all tasks within a given amount of time. This means that, while you should use higher order integration methods for models with widely varying dynamics, the higher order methods require additional computation time. In turn, the additional computation time may force you to use a larger step size, which can diminish the accuracy increase initially sought from the higher order integration method.

Generally, the stiffer the equations, (i.e., the more dynamics in the system with widely varying time constants), the higher the order of the method that you must use.

In practice, the simulation of very stiff equations is impractical for real-time purposes except at very low sample rates. You should test fixed-step size integration in Simulink to check stability and accuracy before implementing the model for use in real-time programs.

For linear systems, it is more practical to convert the model that you are simulating to a discrete time version, for instance, using the `c2d` function in the Control System Toolbox.

Application Modules for System Independent Components

The system independent components include these modules:

- `ode1.c`, `ode2.c`, `ode3.c`, `ode4.c`, `ode5.c` — These modules implement the integration algorithms supported for real-time applications. See the Simulink documentation for more information about these fixed-step solvers.
- `rt_sim.c` — Performs the activities necessary for one time step of the model. It calls the model function to calculate system outputs and then updates the discrete and continuous states.
- `simstruc.h` — Contains actual definition of the Simulink data structure and the definition of the `SimStruct` access macros.
- `simstruc_types.h` — Contains definitions of various events, including subsystem enable/disable and zero crossings. It also defines data logging variables.

The system independent components also include code that defines, creates, and destroys the Simulink data structure (`SimStruct`). The model code and all S-functions included in the program define their own `SimStruct`.

The `SimStruct` data structure encapsulates all the data relating to the model or S-function, including block parameters and outputs. See *Writing S-Functions* for more information about the `SimStruct`.

Rapid Prototyping Application Components

The application components contain the generated code for the Simulink model, including the code for any S-functions in the model. This code is referred to as the model code because these functions implement the Simulink model.

However, the generated code contains more than just functions to execute the model (as described in the previous section). There are also functions to perform initialization, facilitate data access, and complete tasks before program termination. To perform these operations, the generated code must define functions that:

- Create the `SimStruct`.
- Initialize model size information in the `SimStruct`.
- Initialize a vector of sample times and sample time offsets and store this vector in the `SimStruct`.

- Store the values of the block initial conditions and program parameters in the `SimStruct`.
- Compute the block and system outputs.
- Update the discrete state vector.
- Compute derivatives for continuous models.
- Perform an orderly termination at the end of the program (when the current time equals the final time, if a final time is specified).
- Collect block and scope data for data logging (either with the Real-Time Workshop or third-party tools).

The SimStruct Data Structure

The generated code includes the file `simstruc.h`, which contains the definition of the `SimStruct` data structure. Each instance of a model (or an S-function) in the program creates its own `SimStruct`, which it uses for reading and writing data.

All functions in the generated code are public. For this reason, there can be only one instance of a model in a real-time program. This function, which always has the same name as the model, is called during program initialization to return a pointer to the `SimStruct` and initialize any S-functions.

Rapid Prototyping Model Code Functions

The functions defined by the model code are called at various stages of program execution (i.e., initialization, model execution, or program termination).

The following diagram illustrates the functions defined in the generated code and shows what part of the program executes each function.

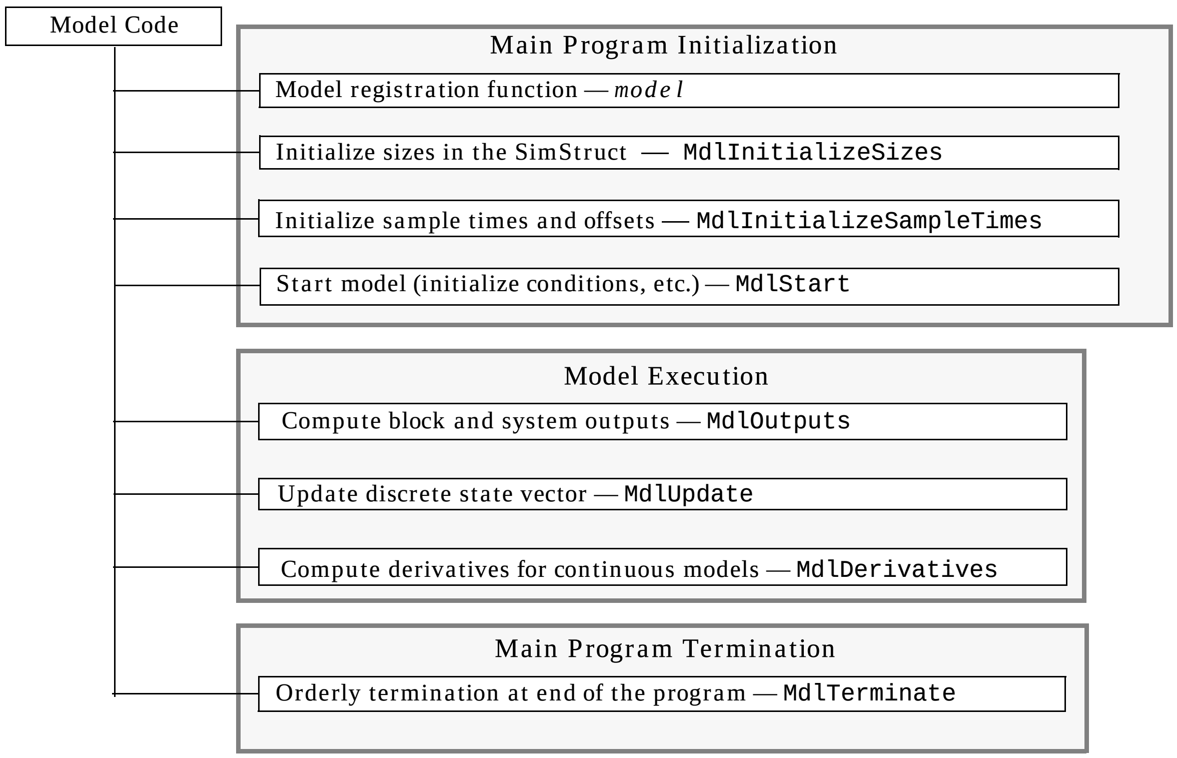

Model Code
Main Program Initialization
Model registration function — *model*
Initialize sizes in the SimStruct — MdlInitializeSizes
Initialize sample times and offsets — MdlInitializeSampleTimes
Start model (initialize conditions, etc.) — MdlStart
Model Execution
Compute block and system outputs — MdlOutputs
Update discrete state vector — MdlUpdate
Compute derivatives for continuous models — MdlDerivatives
Main Program Termination
Orderly termination at end of the program — MdlTerminate

Figure 6-8: Execution of the Model Code

The Model Registration Function

The model registration function has the same name as the Simulink model from which it is generated. It is called directly by the main program during initialization. Its purpose is to initialize and return a pointer to the `SimStruct`.

Models Containing S-Functions

A *noninlined S-function* is any C MEX S-function that is not implemented using a customized TLC file. If you create a C MEX S-function as part of a Simulink model, it is by default noninlined unless you write your own TLC file that inlines it within the body of the *model*`.c` code. The Real-Time Workshop automatically incorporates your non-inlined C code S-functions into the program if they adhere to the S-function API described in the Simulink documentation.

This format defines functions and a `SimStruct` that are local to the S-function. This allows you to have multiple instances of the S-function in the model. The model's `SimStruct` contains a pointer to each S-function's `SimStruct`.

Code Generation and S-Functions

If a model contains S-functions, the source code for the S-function must be on the search path the make utility uses to find other source files. The directories that are searched are specified in the template makefile that is used to build the program.

S-functions are implemented in a way that is directly analogous to the model code. They contain their own public registration function (which is called by the top-level model code) that initializes static function pointers in its `SimStruct`. When the top-level model needs to execute the S-function, it does so via the function pointers in the S-function's `SimStruct`. The S-functions use the same `SimStruct` data structure as the generated code; however, there can be more than one S-function with the same name in your model. This is accomplished by having function pointers to static functions.

Inlining S-Functions

You can incorporate C MEX S-functions, along with the generated code, into the program executable. You can also write a target file for your C MEX S-function to inline the S-function, thus improving performance by eliminating function calls to the S-function itself. For more information on inlining S-functions, see the *Target Language Compiler Reference Guide*.

Application Modules for Application Components

When the Real-Time Workshop generates code, it produces the following files:

- *model*`.c` — The C code generated from the Simulink block diagram. This code implements the block diagram's system equations as well as performing initialization and updating outputs.
- *model*`.h` — Header file containing the block diagram's simulation parameters, I/O structures, work structures, and other declarations.
- *model*`_export.h` — Header file containing declarations of exported signals and parameters.

These files are named for the Simulink model from which they are generated.

If you have created custom blocks using C MEX S-functions, you need the source code for these S-functions available during the build process.

Embedded Program Framework

The Real-Time Workshop Embedded Coder provides a framework for embedded programs. Its architecture is outlined by the following figure.

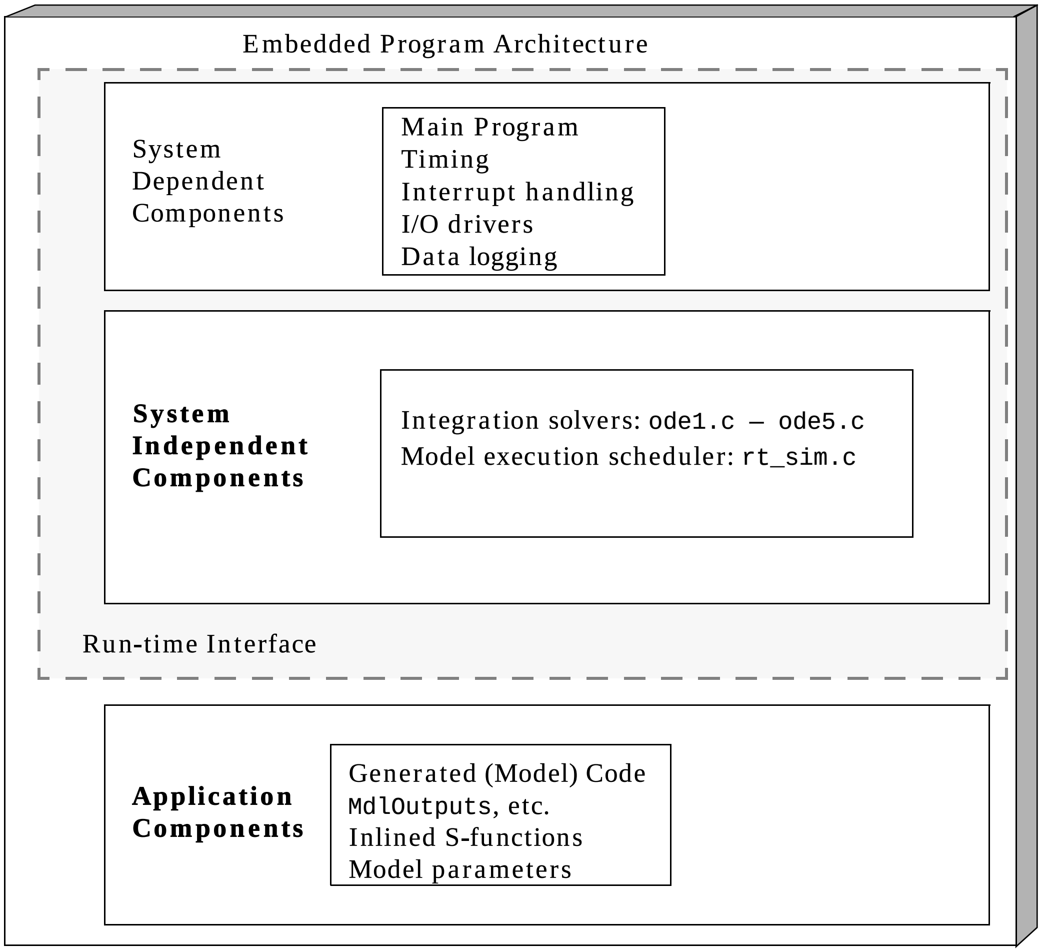


Figure 6-9: Embedded Program Architecture

Note the similarity between this architecture and the rapid prototyping architecture on page 6-25. The main difference is the lack of the `SimStruct` data structure and the removal of the noninlined S-functions.

Using this figure, you can compare the embedded style of generated code, used in the Real-Time Workshop Embedded Coder, with the rapid prototyping style of generated code of the previous section. Most of the rapid prototyping explanations in the previous section hold for the Real-Time Workshop Embedded Coder target. The Real-Time Workshop Embedded Coder target simplifies the process of using the generated code in your custom-embedded applications by providing a model- specific API and eliminating the `SimStruct`. This target contains the same conceptual layering as the rapid prototyping target, but each layer has been simplified.

For a discussion of the structure of embedded real-time code, see Chapter 9, “Real-Time Workshop Embedded Coder.”

Models with Multiple Sample Rates

Introduction

Every Simulink block can be classified according to its sample time as constant, continuous-time, discrete-time, inherited, or variable. Examples of each type include:

- Constant — Constant block, Width
- Continuous-time — Integrator, Derivative, Transfer Function
- Discrete-time — Unit Delay, Digital Filter
- Inherited — Gain, Sum, Lookup Table
- Variable — These are S-Function blocks that set their time of next hit based upon current information. These blocks work only with variable step solvers.

Blocks in the inherited category assume the sample time of the blocks that are driving them. Every Simulink block therefore has a sample time, whether it is explicit, as in the case of continuous or discrete blocks (continuous blocks have a sample time of zero), or implicit, as in the case of inherited blocks.

Simulink allows you to create models without any restrictions on connections between blocks with different sample times. It is therefore possible to have blocks with differing sample times in a model (a mixed-rate system). A possible advantage of employing multiple sample times is improved efficiency when executing in a multitasking real-time environment.

Simulink provides considerable flexibility in building these mixed-rate systems. However, the same flexibility also allows you to construct models for which the code generator cannot generate correct real-time code for execution in a multitasking environment. But to make these models operate correctly in real time (i.e., give the right answers), you must modify your model. In general, the modifications involve placing Unit Delay and Zero Order Hold blocks between blocks that have unequal sample rates. The sections that follow discuss the issues you must address to use a mixed-rate model successfully in a multitasking environment.

Single- Versus Multitasking Environments

There are two basic ways in which you can execute a fixed-step Simulink model: singletasking and multitasking. You use the **Solver options** pull-down menu on the Solver page of the **Simulation Parameters** dialog box to specify how to execute your model. The default is `auto`, which specifies that your model will use multitasking if your model contains two or more different rates. Otherwise, it will use singletasking.

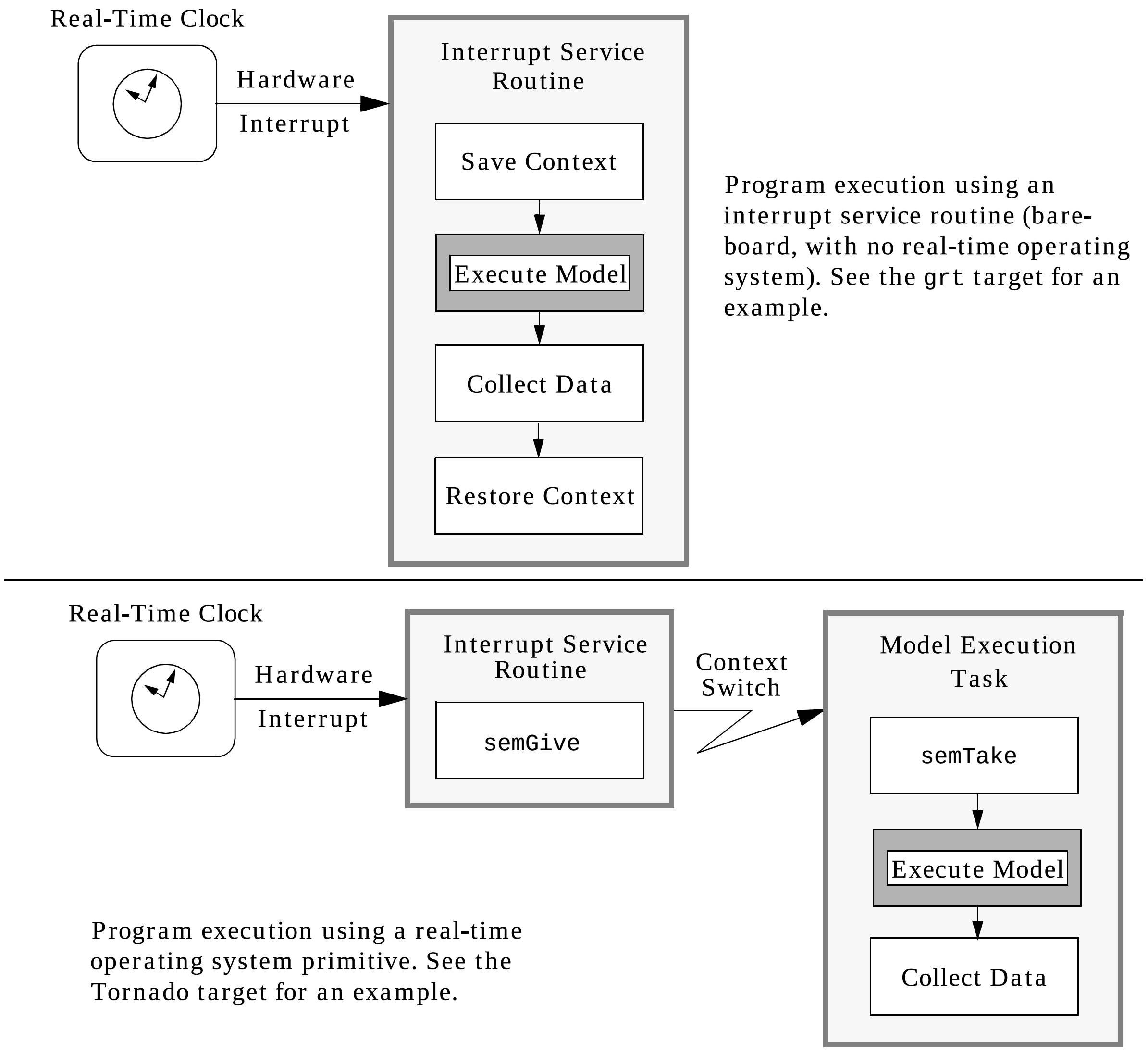
Execution of models in a real-time system can be done with the aid of a real-time operating system, or it can be done on a *bare-board* target, where the model runs in the context of an interrupt service routine (ISR).

Note that the fact that a system (such as UNIX or Microsoft Windows) is multitasking does not guarantee that the program can execute in real time. This is because it is not guaranteed that the program can preempt other processes when required.

In DOS, where only one process can exist at any given time, an interrupt service routine (ISR) must perform the steps of saving the processor context, executing the model code, collecting data, and restoring the processor context.

Tornado, on the other hand, provides automatic context switching and task scheduling. This simplifies the operations performed by the ISR. In this case, the ISR simply enables the model execution task, which is normally blocked.

Figure 7-1 illustrates this difference.



Program execution using an interrupt service routine (bare-board, with no real-time operating system). See the `grt` target for an example.

Program execution using a real-time operating system primitive. See the Tornado target for an example.

Figure 7-1: Real-Time Program Execution

This chapter focuses on when and how the run-time interface executes your model. See “Program Execution” on page 6-14 for a description of what happens during model execution.

Executing Multitasking Models

In cases where the continuous part of a model executes at a rate that is different from the discrete part, or a model has blocks with different sample rates, the code assigns each block a *task identifier* (`tid`) to associate it with the task that executes at its sample rate.

Certain restrictions apply to the sample rates that you can use:

- The sample rate of any block must be an integer multiple of the base (i.e., the fastest) sample rate. The base sample rate is determined by the fixed step size specified on the Solver page of the **Simulation parameters** dialog box (if a model has continuous blocks) or by the fastest sample time specified in the model (if the model is purely discrete). Continuous blocks always execute via an integration algorithm that runs at the base sample rate.
- The continuous and discrete parts of the model can execute at different rates only if the discrete part is executed at the same or a slower rate than the continuous part (and is an integer multiple of the base sample rate).

Multitasking and Pseudomultitasking

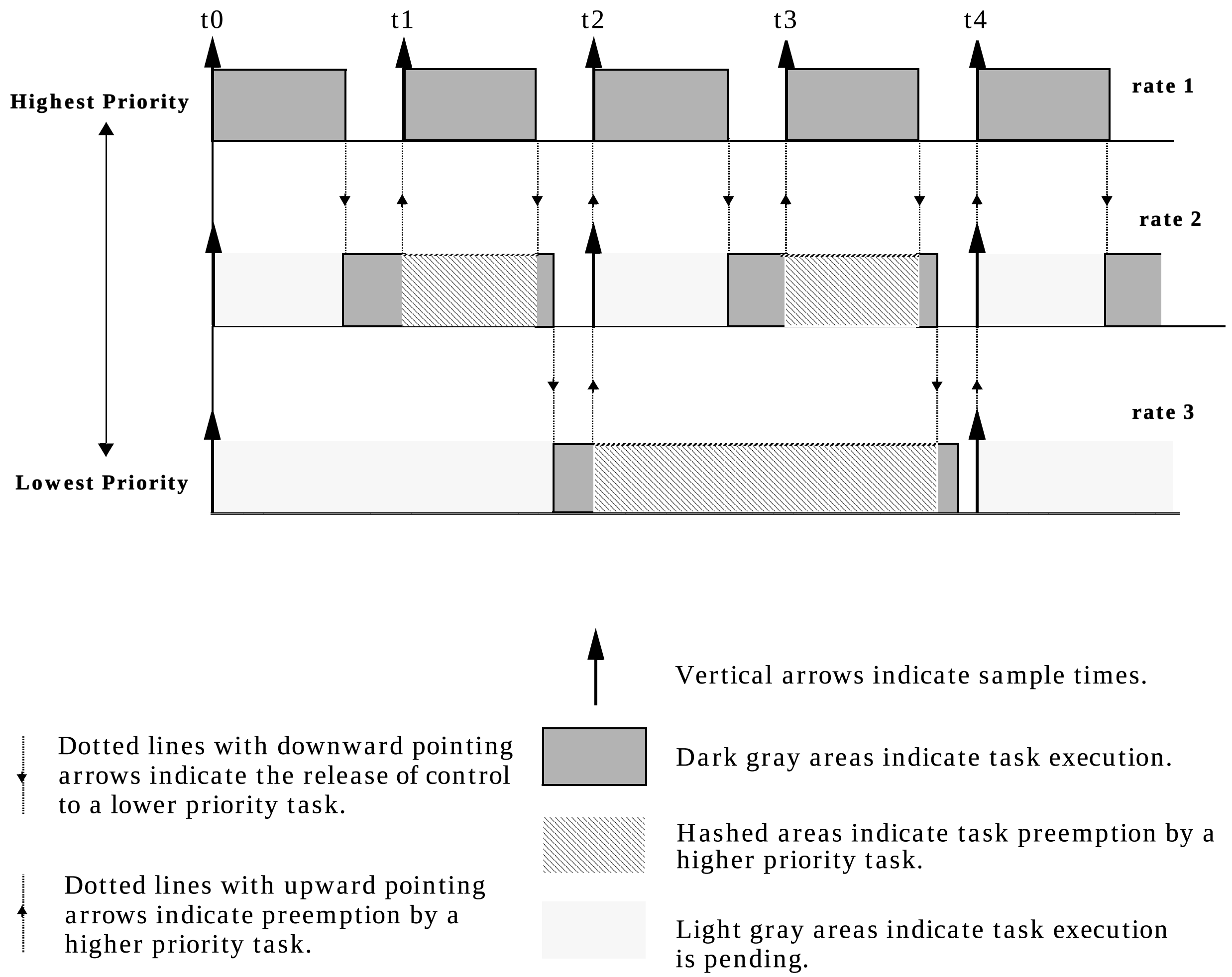
In a multitasking environment, the blocks with the fastest sample rates are executed by the task with the highest priority, the next slowest blocks are executed by a task with the next lower priority, and so on. Time available in between the processing of high priority tasks is used for processing lower priority tasks. This results in efficient program execution.

See “Multitasking System Execution” on page 7-7 for a graphical representation of task timing.

In multitasking environments (i.e., a real-time operating system), you can define separate tasks and assign them priorities. In a bare-board target (i.e., no real-time operating system present), you cannot create separate tasks. However, the Real-Time Workshop application modules implement what is effectively a multitasking execution scheme using overlapped interrupts, accompanied by manual context switching.

This means an interrupt can occur while another interrupt is currently in progress. When this happens, the current interrupt is preempted, the floating-point unit (FPU) context is saved, and the higher priority interrupt executes its higher priority (i.e., faster sample rate) code. Once complete, control is returned to the preempted ISR.

The following diagrams illustrate how mixed-rate systems are handled by the Real-Time Workshop in these two environments.



Vertical arrows indicate sample times.

Dotted lines with downward pointing arrows indicate the release of control to a lower priority task.

Dark gray areas indicate task execution.

Hashed areas indicate task preemption by a higher priority task.

Dotted lines with upward pointing arrows indicate preemption by a higher priority task.

Light gray areas indicate task execution is pending.

Figure 7-2: Multitasking System Execution

Figure 7-3 illustrates how overlapped interrupts are used to implement pseudomultitasking. Note that in this case, Interrupt 0 does not return until after Interrupts 1, 2, and 3.

.

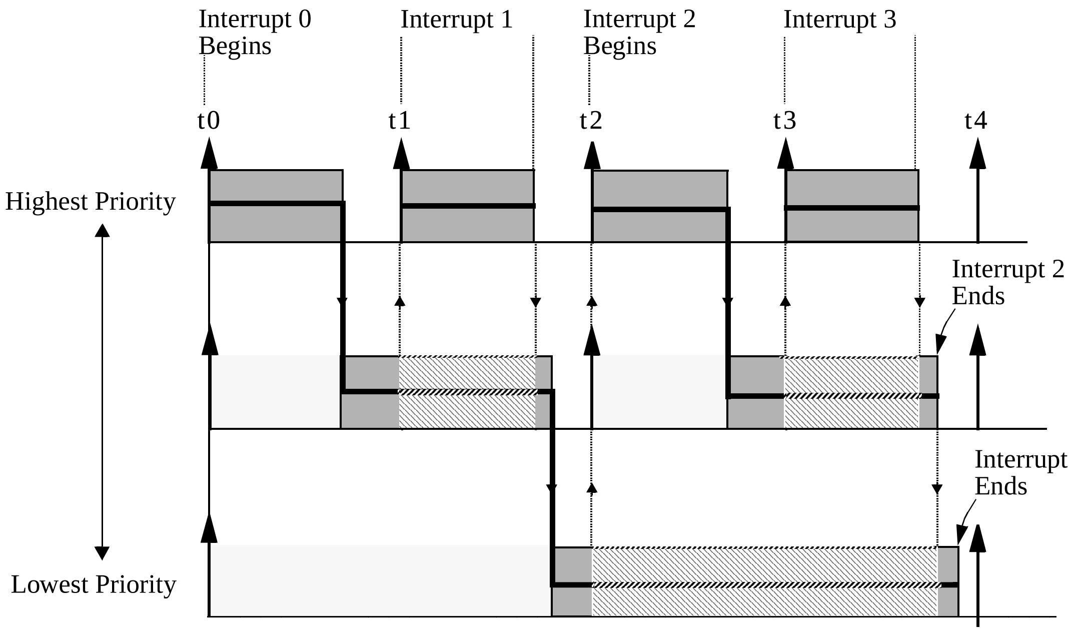


Figure 7-3: Pseudomultitasking Using Overlapped Interrupts

Building the Program for Multitasking Execution

To use multitasking execution, select `auto` (the default) or multitasking as the mode on the Solver page of the **Simulation Parameters** dialog box. The **Mode** menu is only active if you have selected fixed-step as the Solver options type. `auto` solver mode will result in a multitasking environment if your model has more than two sample times or it has two different sample times. In particular, a model with a continuous and a discrete sample time will run in singletasking mode if the fixed-step size is equal to the discrete sample time.

Singletasking

It is possible to execute the model code in a strictly singletasking manner. While this method is less efficient with regard to execution speed, in certain situations it may allow you to simplify your model.

In a singletasking environment, the base sample rate must define a time interval that is long enough to allow the execution of all blocks within that interval.

The following diagram illustrates the inefficiency inherent in singletasking execution.

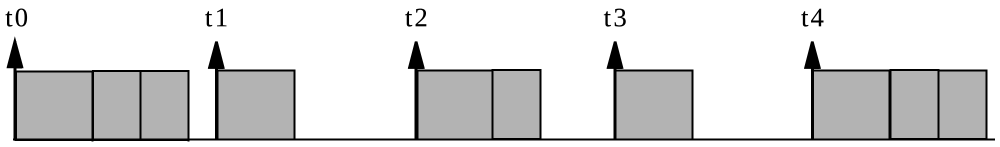


Figure 7-4: Singletasking System Execution

Singletasking system execution requires a sample interval that is long enough to execute one step through the entire model.

Building the Program for Singletasking Execution

To use singletasking execution, select the singletasking mode on the Solver page of the **Simulation Parameters** dialog box. If the solver mode is `auto`, singletasking is used in the following cases:

- If your model contains one sample time
- If your model contains a continuous and a discrete sample time and the fixed step size is equal to the discrete sample time

Model Execution

To generate code that executes correctly in real time, you may need to modify sample rate transitions within the model before generating code. To understand this process, first consider how Simulink simulations differ from real-time programs.

Simulating Models with Simulink

Before Simulink simulates a model, it orders all of the blocks based upon their topological dependencies. This includes expanding subsystems into the individual blocks they contain and flattening the entire model into a single list. Once this step is complete, each block is executed in order.

The key to this process is the proper ordering of blocks. Any block whose output is directly dependent on its input (i.e., any block with direct feedthrough) cannot execute until the block driving its input has executed.

Some blocks set their outputs based on values acquired in a previous time step or from initial conditions specified as a block parameter. The output of such a block is determined by a value stored in memory, which can be updated independently of its input. During simulation, all necessary computations are performed prior to advancing the variable corresponding to time. In essence, this results in all computations occurring instantaneously (i.e., no computational delay).

Executing Models in Real Time

A real-time program differs from a Simulink simulation in that the program must execute the model code synchronously with real time. Every calculation results in some computational delay. This means the sample intervals cannot be shortened or lengthened (as they can be in Simulink), which leads to less efficient execution.

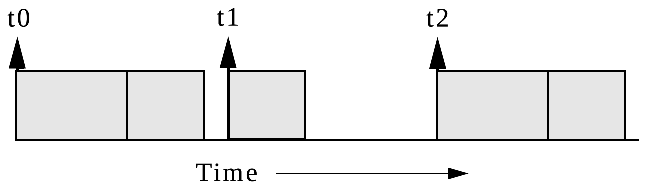


Figure 7-5: Unused Time in Sample Interval

Sample interval t1 cannot be compressed to increase execution speed because by definition, sample times are clocked in real time.

Real-Time Workshop application programs are designed to circumvent this potential inefficiency by using a multitasking scheme. This technique defines tasks with different priorities to execute parts of the model code that have different sample rates.

See “Multitasking and Pseudomultitasking” on page 7–5 for a description of how this works. It is important to understand that section before proceeding here.

Multitasking Operation

The use of multitasking can improve the efficiency of your program if the model is large and has many blocks executing at each rate. It can also degrade performance if your model is dominated by a single rate, and only a few blocks execute at a slower rate. In this situation, the overhead incurred in task switching can be greater than the time required to execute the slower blocks. It is more efficient to execute all blocks at the dominant rate.

If you have a model that can benefit from multitasking execution, you may need to modify your Simulink model for this scheme to generate correct results.

Singletasking Operation

Alternatively, you can run your real-time program in singletasking mode. Singletasking programs require longer sample intervals due to the inherent inefficiency of that mode of execution.

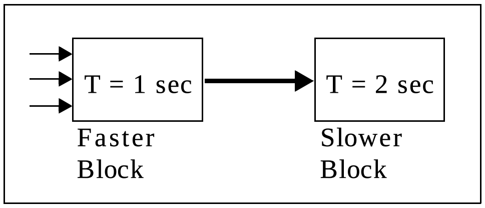
Sample Rate Transitions

There are two possible sample rate transitions that can exist within a model:

- A faster block driving a slower block
- A slower block driving a faster block

In singletasking systems, there are no issues involved with multiple sample rates. In multitasking and pseudomultitasking systems, however, differing sample rates can cause problems. To prevent possible errors in calculated data, you must control model execution at these transitions. In transitioning from faster to slower blocks, you must add Zero-Order Hold blocks between fast to slow transitions and set the sample rate of the Zero-Order Hold block to that of the slower block.

This diagram



becomes

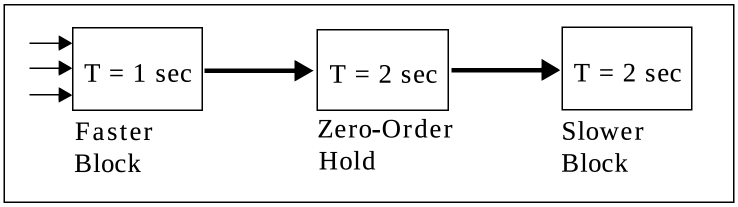
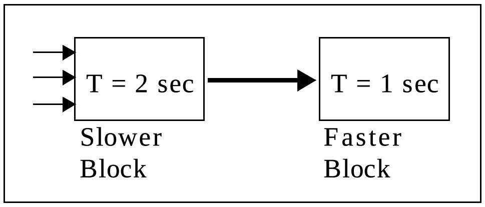


Figure 7-6: Transitioning from Faster to Slower Blocks (T = sample period)

In transitioning from slower to faster blocks, you must add Unit Delay blocks between slow to fast transitions and set the sample rate of the Unit Delay block to that of the slower block.

This diagram



becomes

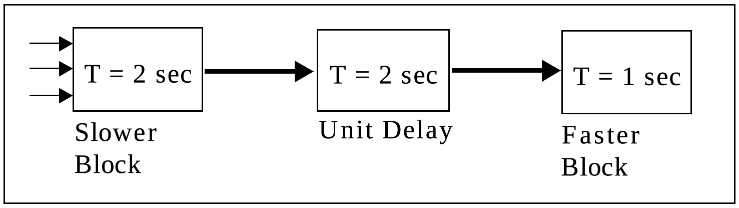


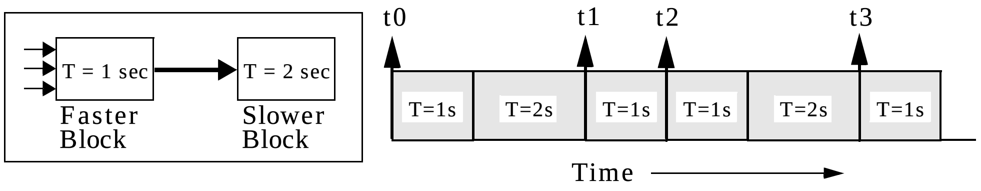
Figure 7-7: Transitioning from Slower to Faster Blocks (T = Sample Period)

The next four sections describe the theory and reasons why Unit Delay and Zero-Order Hold blocks are necessary for sample time transitions.

Faster to Slower Transitions in Simulink

In a model where a faster block drives a slower block having direct feedthrough, the outputs of the faster block are always computed first. In simulation intervals where the slower block does not execute, the simulation progresses more rapidly because there are fewer blocks to execute.

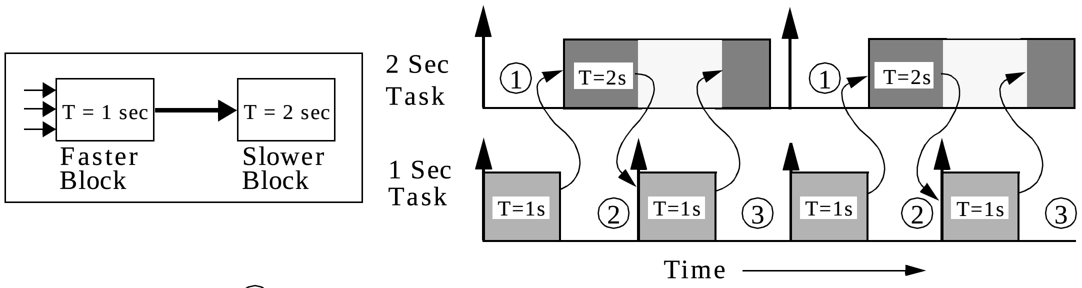
The following diagram illustrates this situation.



Simulink does not execute in real time, which means that it is not bound by real-time constraints. Simulink waits for, or moves ahead to, whatever tasks are necessary to complete simulation flow. The actual time interval between sample time steps can vary.

Faster to Slower Transitions in Real Time

In models where a faster block drives a slower block, you must compensate for the fact that execution of the slower block may span more than one execution period of the faster block. This means that the outputs of the faster block may change before the slower block has finished computing its outputs. The following diagram illustrates a situation where this problem arises. The hashed area indicates times when tasks are preempted by higher priority before completion.



① The faster task (T=1s) completes.

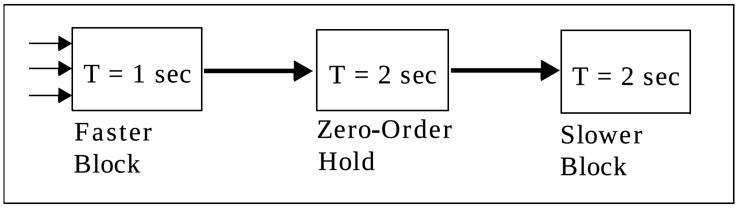
② Higher priority preemption occurs.

③ The slower task (T=2s) resumes and its inputs have changed. This leads to unpredictable results.

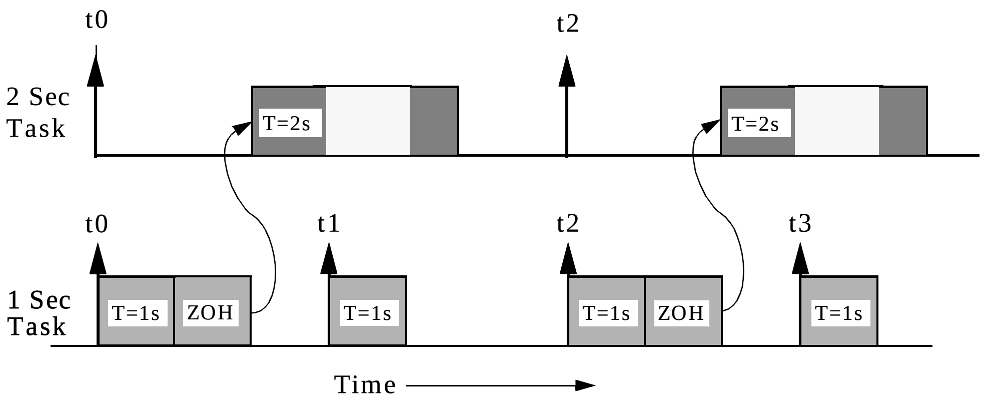
Figure 7-8: Time Overlaps in Faster to Slower Transitions (T=Sample Time)

In Figure 7-8, the faster block executes a second time before the slower block has completed execution. This can cause unpredictable results because the input data to the slow task is changing.

To avoid this situation, you must hold the outputs of the 1 second (faster) block until the 2 second (slower) block finishes executing. The way to accomplish this is by inserting a Zero-Order Hold block between the 1 second and 2 second blocks. The sample time of the Zero Order Hold block must be set to 2 seconds (i.e., the sample time of the slower block).



The Zero Order Hold block executes at the sample rate of the slower block, but with the priority of the faster block.

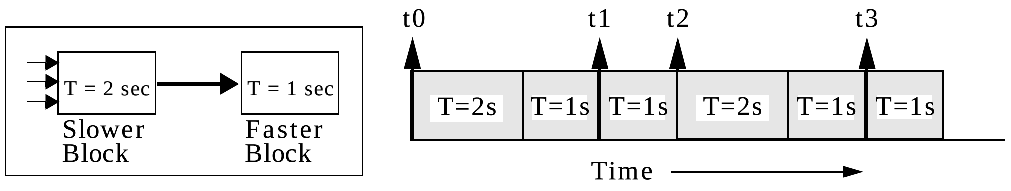


This ensures that the Zero Order Hold block executes before the 1 second block (its priority is higher) and that its output value is held constant while the 2 second block executes (it executes at the slower sample rate).

Slower to Faster Transitions in Simulink

In a model where a slower block drives a faster block, Simulink again computes the output of the driving block first. During sample intervals where only the faster block executes, the simulation progresses more rapidly.

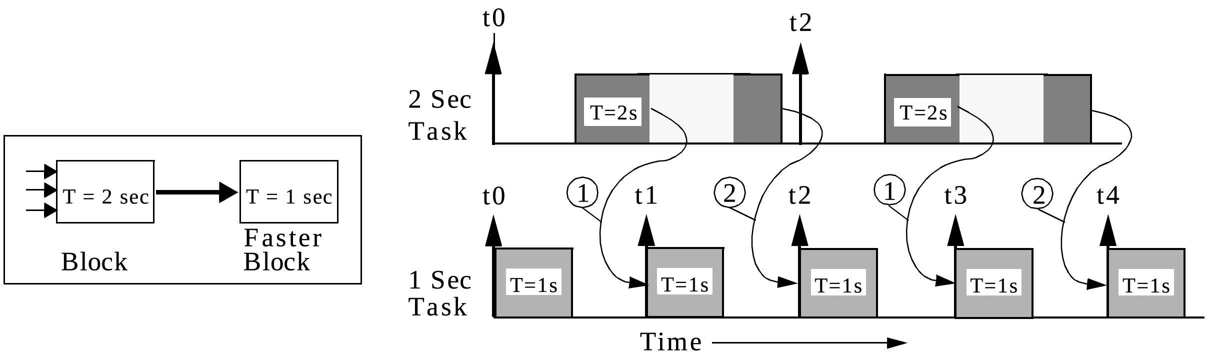
The following diagram illustrates the execution sequence.



As you can see from the preceding diagrams, Simulink can simulate models with multiple sample rates in an efficient manner. However, Simulink does not operate in real time.

Slower to Faster Transitions in Real Time

In models where a slower block drives a faster block, the generated code assigns the faster block a higher priority than the slower block. This means the faster block is executed before the slower block, which requires special care to avoid incorrect results.



① The faster block executes a second time prior to the completion of the slower block.

② The faster block executes before the slower block.

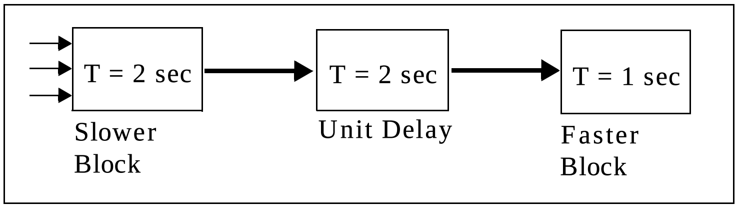
Figure 7-9: Time Overlaps in Slower to Faster Transitions

This timing diagram illustrates two problems:

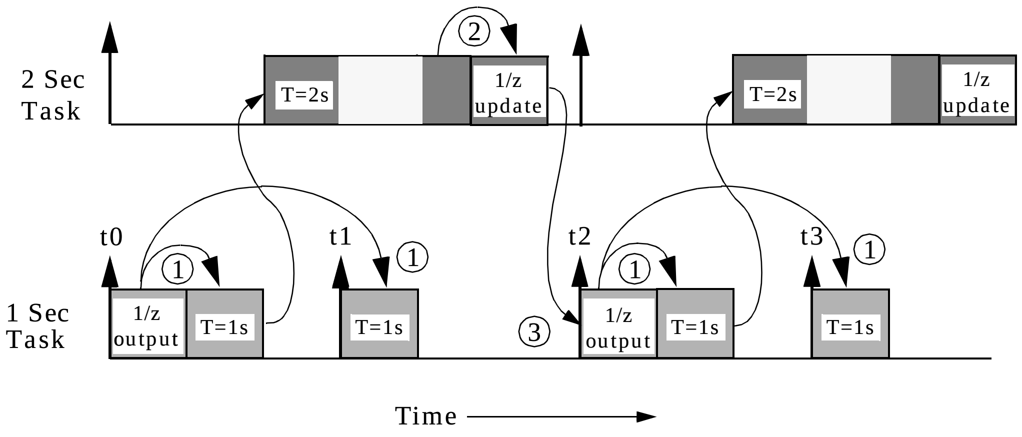
1. Execution of the slower block is split over more than one faster block interval. In this case the faster task executes a second time before the slower task has completed execution. This means the inputs to the slower task can change, causing unpredictable results.

2. The faster block executes before the slower block (which is backwards from the way Simulink operates). In this case, the 1 second block executes first; but the inputs to the faster task have not been computed. This can cause unpredictable results.

To eliminate these problems, you must insert a Unit Delay block between the slower and faster blocks. The sample rate for a Unit Delay block must be set to that of the block that is driving it (i.e., the slower block).



The picture below shows the timing sequence that results with the added Unit Delay block.



Three key points about this diagram:

1 Unit delay output runs in 1 second task, but only at its rate (2 seconds). The output of the unit delay block feeds the 1 second task blocks.

2 The unit delay update uses the output of the 2 second task in its update of its internal state.

3 The unit delay update uses the state of the unit delay in the 1 second task.

The output portion of a Unit Delay block is executed at the sample rate of the slower block, but with the priority of the faster block. Since a Unit Delay block drives the faster block and has effectively the same priority, it is executed before the faster block. This solves the first problem.

The second problem is alleviated because the Unit Delay block executes at a slower rate and its output does not change during the computation of the faster block it is driving.

Note Inserting a Unit Delay block changes the model. The output of the slower block is now delayed by one time step compared to the output without a Unit Delay block.

Optimizing the Model for Code Generation

Overview

There are a number of ways that you can optimize code generated by the Real-Time Workshop from your model, with respect to both memory usage and performance.

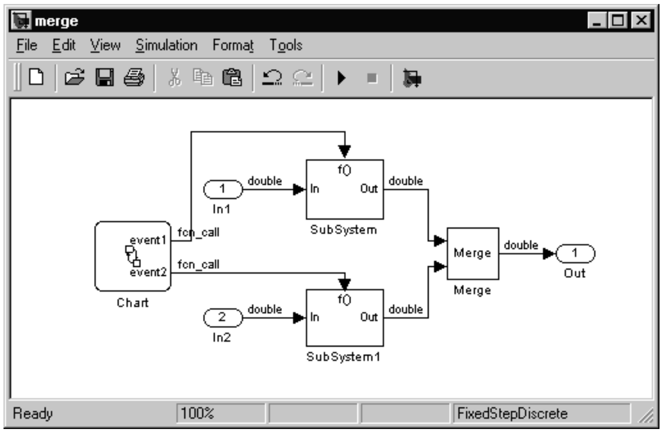
This chapter discusses optimization techniques that are common to all target configurations and code formats. For optimizations specific to a particular target configuration, see the chapter relevant to that target.

General Modeling Techniques

The following are techniques that you can use with any code format:

- The `slupdate` command automatically converts older models to use current features. Run `slupdate` on old models.
- Directly inline C code S-functions into the generated code by writing a TLC file for the S-function. See the *Target Language Compiler Reference Guide* for more information on inlining S-functions. Also see "Creating Device Drivers" on page 17-34 for information on inlining device driver S-functions.
- Use a Simulink data type other than double when possible. The available data types are Boolean, signed and unsigned 8-, 16-, and 32-bit integers, and 32- and 64-bit floats. A double is a 64-bit float. See *Using Simulink* for more information on data types.
- Remove repeated values in lookup table data.
- Use the Merge block to merge the output of function-call subsystems. This block is particularly helpful when controlling the execution of function-call subsystems with Stateflow.

  This diagram is an example of how to use the Merge block.



Block Diagram Performance Tuning

Certain block constructs in Simulink will run faster, or require less code or data memory, than other seemingly equivalent constructs. Knowing the trade-offs between similar blocks and block parameter options will enable you to create Simulink models that have intuitive diagrams, and to produce the tight code that you want from Real-Time Workshop. Many of the options and constructs discussed in this section will improve the simulation speed of the model itself, even without code generation.

Look-Up Tables and Polynomials

Simulink provides several blocks that allow approximation of functions. These include blocks that perform direct, interpolated and cubic spline lookup table operations, and a polynomial evaluation block.

There are currently six different blocks in Simulink that perform lookup table operations:

- Look-Up Table
- Look-Up Table (2-D)
- Look-Up Table (n-D)
- Direct Look-Up Table (n-D)
- PreLook-Up Index Search
- Interpolation (n-D) Using PreLook-Up Index Search

In addition, the Repeating Sequence block uses a lookup table operation, the output of which is a function of the real-time (or simulation-time) clock.

To get the most out of the following discussion, you should familiarize yourself with the features of these blocks, as documented in the *Using Simulink* manual.

Each type of lookup table block has its own set of options and associated trade-offs. The examples in this section show how to use lookup tables effectively. The techniques demonstrated here will help you achieve maximal performance with minimal code and data sizes.

Multi-Channel Nonlinear Signal Conditioning

Figure 8-1 shows a Simulink model that reads input from two 8-channel, high-speed 8-bit analog/digital converters (ADCs). The ADCs are connected to Type K thermocouples through a gain circuit with an amplification of 250. Since the popular Type K thermocouples are highly nonlinear, there is an international standard for converting their voltages to temperature. In the range of 0 to 500 degrees Celsius, this conversion is a tenth-order polynomial. One way to perform the conversion from ADC readings (0-255) into temperature (in degrees Celsius) is to evaluate this polynomial. In the best case, the polynomial evaluation requires 9 multiplications and 10 additions per channel.

A polynomial evaluation is not the fastest way to convert these 8-bit ADC readings into measured temperature. Instead, the model uses a Direct Look-Up (n-D) Table block (named `TypeK_TC`) to map 8-bit values to temperature values. This block performs one array reference per channel.

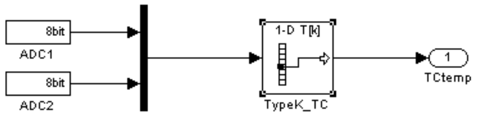
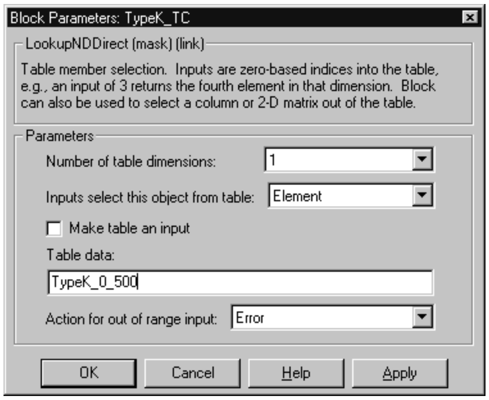


Figure 8-1: Direct Look-Up Table (n-D) Block Conditions ADC Input

The block's table parameter is populated with 256 values that correspond to the temperature at an ADC reading of 0, 1, 2, ... up to 255. The table data, calculated in MATLAB, is stored in the workspace variable `TypeK_0_500`. The block's **Table data** parameter field references this variable, as shown in Figure 8-2.


Block Parameters: TypeK_TC

LookupNDDirect (mask) (link)

Table member selection. Inputs are zero-based indices into the table, e.g., an input of 3 returns the fourth element in that dimension. Block can also be used to select a column or 2-D matrix out of the table.

Parameters

Number of table dimensions: 1

Inputs select this object from table: Element

☐ Make table an input

Table data: TypeK_0_500

Action for out of range input: Error

Figure 8-2: Parameters of Direct Look-Up Table (n-D) Block

The model uses a Mux block to collect all similar signals (e.g., Type K thermocouple readings) and feed them into a singleDirect Look-Up Table block. This is more efficient than using one Direct Look-Up Table block per device. If multiple blocks share a common parameter (such as the table in this example), the Real-Time Workshop creates only one copy of that parameter in the generated code.

This is the recommended approach for signal conditioning when the size of the table can fit within your memory constraints. In this example, the table stores 256 double (8-byte) values, utilizing 2 KB of memory.

Note that the `TypeK_TC` block processes 16 channels of data sequentially.

The Real-Time Workshop generates the following code for the `TypeK_TC` block shown in Figure 8-1.

```
/* (LookupNDDirect) Block: <Root>/TypeK_TC */
/* 1-dimensional Direct Look-Up Table returning 16 Scalars */
{
  int_T i1;
  const uint8_T *u0 = &rtb_s1_Data_Type_Conversion[0];
  real_T *y0 = &rtb_root_TypeK_TC[0];

  for (i1=0; i1 < 8; i1++) {
    y0[i1] = (rtP.root_TypeK_TC_table[(uint8_T)u0[i1]]);
  }
  u0 = &rtb_s2_Data_Type_Conversion[0];
  y0 = &rtb_root_TypeK_TC[8];

  for (i1=0; i1 < 8; i1++) {
    y0[i1] = (rtP.root_TypeK_TC_table[(uint8_T)u0[i1]]);
  }
}
```

Notice that the core of each loop is one line of code that directly retrieves a table element from the table and places it in the block output variable. There are two loops in the generated code because the two simulated ADCs are not merged into a contiguous memory array in the Mux block. Instead, to avoid a copy operation, the Direct Look-Up Table block performs the lookup on two sets of data using a single table array (`rtP.root_TypeK_TC_table[]`).

If the input accuracy for your application (not to be confused with the number of I/O bits) is 24 bits or less, you can use a single precision table for signal conditioning. Then, cast the lookup table output to double precision for use in the rest of the block diagram. This technique, shown in Figure 8-3, causes no loss of precision.

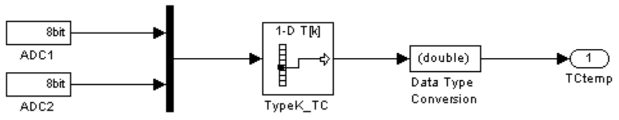
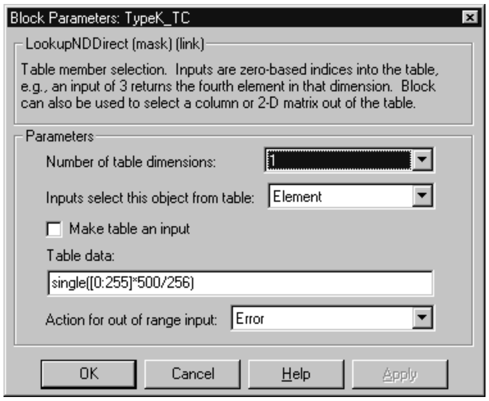


Figure 8-3: Single Precision Lookup Table Output Is Cast to Double Precision

Note that a direct lookup table covering 24 bits of accuracy would require 64 megabytes of memory, which is typically not practical. To create a single precision table, use the MATLAB `single()` cast function in your table calculations. Alternatively, you can perform the type cast directly in the **Table data** parameter, as shown in Figure 8-4.


Block Parameters: TypeK_TC

LookupNDDirect (mask) (link)

Table member selection. Inputs are zero-based indices into the table, e.g., an input of 3 returns the fourth element in that dimension. Block can also be used to select a column or 2-D matrix out of the table.

Parameters

Number of table dimensions: 1

Inputs select this object from table: Element

☐ Make table an input

Table data:

single([0:255]*500/256)

Action for out of range input: Error

Figure 8-4: Type Casting Table Data in a Direct Look-Up Block

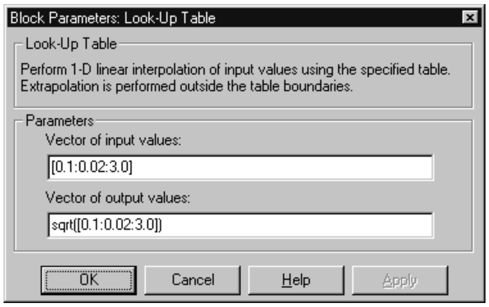
When table size becomes impractical, you must use other nonlinear techniques, such as interpolation or polynomial techniques. The Look-Up Table (n-D) block supports linear interpolation and cubic spline interpolation.The Polynomial block supports evaluation of noncomplex polynomials.

Compute-Intensive Equations

The blocks described in this section are useful for simplifying fixed, complex relationships that are normally too time consuming to compute in real time.

The only practical way to implement some compute-intensive functions or arbitrary nonlinear relationships in real time is to use some form of lookup table. On processors that do not have floating-point instructions, even functions like `sqrt()` can become too expensive to evaluate in real time.

An approximation to the nonlinear relationship in a known range will work in most cases. For example, your application might require a square root calculation that your target processor's instruction set does not support. The illustration below shows how you can use a Look-Up Table block to calculate an approximation of the square root function that covers a given range of the function.


Block Parameters: Look-Up Table

Look-Up Table

Perform 1-D linear interpolation of input values using the specified table. Extrapolation is performed outside the table boundaries.

Parameters

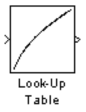
Vector of input values:

```
[0.1:0.02:3.0]
```

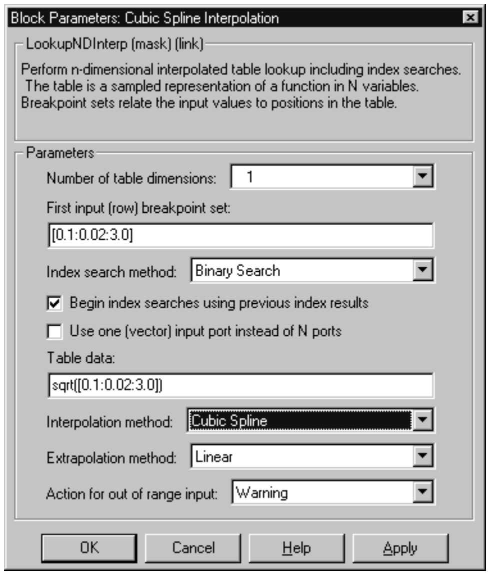
Vector of output values:

```
sqrt([0.1:0.02:3.0])
```

The interpolated values are plotted on the block icon.



For more accuracy on widely spaced points, use a cubic spline interpolation in the Look-Up Table (n-D) block, as shown below.



Block Parameters: Cubic Spline Interpolation

LookupNDInterp (mask) (link)

Perform n-dimensional interpolated table lookup including index searches. The table is a sampled representation of a function in N variables. Breakpoint sets relate the input values to positions in the table.

Parameters

Number of table dimensions: 1

First input (row) breakpoint set: [0.1:0.02:3.0]

Index search method: Binary Search

☑ Begin index searches using previous index results

☐ Use one (vector) input port instead of N ports

Table data: sqrt([0.1:0.02:3.0])

Interpolation method: Cubic Spline

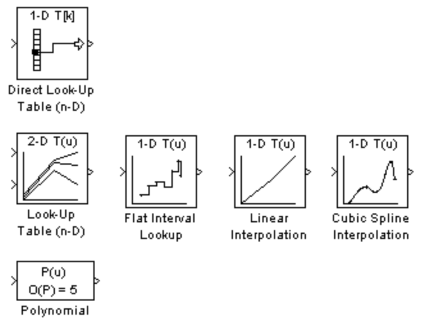
Extrapolation method: Linear

Action for out of range input: Warning

Techniques available in Simulink include n-dimensional support for direct lookup, linear interpolations in a table, cubic spline interpolations in a table, and 1-D real polynomial evaluation.

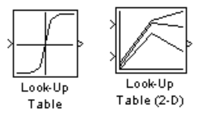
The Look-Up Table (n-D) block supports flat interval lookup, linear interpolation and cubic spline interpolation. Extrapolation for the Look-Up Table (n-D) block can either be disabled (clipping) or enabled for linear or spline extrapolations.

The icons for the Direct Look-Up Table (n-D) and Look-Up Table (n-D) blocks change depending on the type of interpolation selected and the number of dimensions in the table, as illustrated below.



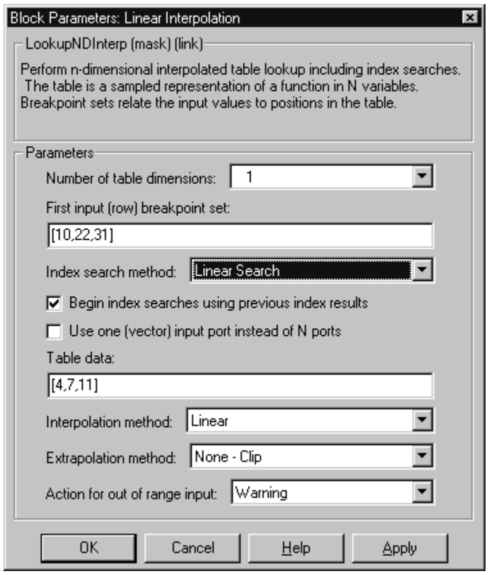
Tables with Repeated Points

The Look-Up Table and Look-Up Table (2-D) blocks, shown below, support linear interpolation with linear extrapolation. In these blocks, the row and column parameters can have repeated points, allowing pure step behavior to be mixed in with the linear interpolations. Note that this capability is not supported by the Look-Up Table (n-D) block.



Slowly vs. Rapidly Changing Look-Up Table Block Inputs

You can optimize lookup table operations using the Look-Up Table (n-D) block for efficiency if you know the input signal's normal rate of change. Figure 8-5 shows the parameters for the Look-Up Table (n-D) block.



Block Parameters: Linear Interpolation

LookupNDInterp (mask) (link)

Perform n-dimensional interpolated table lookup including index searches. The table is a sampled representation of a function in N variables. Breakpoint sets relate the input values to positions in the table.

Parameters

Number of table dimensions: 1

First input (row) breakpoint set: [10,22,31]

Index search method: Linear Search

[✓] Begin index searches using previous index results

[ ] Use one (vector) input port instead of N ports

Table data: [4,7,11]

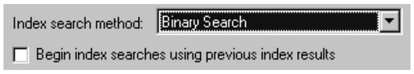
Interpolation method: Linear

Extrapolation method: None - Clip

Action for out of range input: Warning

Figure 8-5: Parameter Dialog for the Look-Up Table (n-D) Block

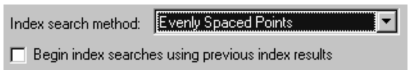
If you do not know the input signal's normal rate of change in advance, it would be better to choose the **Binary Search** option for the index search in the Look-Up Table (n-D) block and the PreLook-Up Index Search block.



Index search method: Binary Search

[ ] Begin index searches using previous index results

Regardless of signal behavior, if the table's breakpoints are evenly spaced, it is best to select the **Evenly Spaced Points** option from the Look-Up Table (n-D) block's parameter dialog.



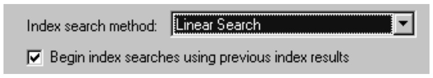
Index search method: Evenly Spaced Points

[ ] Begin index searches using previous index results

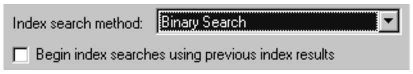
If the breakpoints are not evenly spaced, first decide which of the following best describes the input signal behavior.

- Behavior 1: The signal stays in a given breakpoint interval from one time step to the next. When the signal moves to a new interval, it tends to move to an adjacent interval.
- Behavior 2: The signal has many discontinuities. It jumps around in the table from one time step to the next, often moving three or more intervals per time step.

Given behavior 1, the best optimization for a given lookup table is to use the **Linear search option** and **Begin index searches using previous index results** options, as shown below.

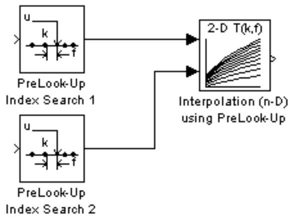

Index search method: Linear Search
[✓] Begin index searches using previous index results

Given behavior 2, the **Begin index searches using previous index results** option does not necessarily improve performance. Choose the **Binary Search** option, as shown below.


Index search method: Binary Search
[ ] Begin index searches using previous index results

The choice of an index search method can be more complicated for lookup table operations of two or more dimensions with linear interpolation. In this case, several signals are input to the table. Some inputs may have evenly spaced points, while others may exhibit behavior 1 or behavior 2.

Here it may be best to use PreLook-Up Index Search blocks with different search methods (evenly spaced, linear search or binary search) chosen according to the input signal characteristics. The outputs of these search blocks are then connected to an Interpolation (n-D) Using PreLook-Up Index Search block, as shown in the block diagram below.



You can configure each PreLook-Up Index Search block independently to use the best search algorithm for the breakpoints and input time variation cases.

Multiple Tables with Common Inputs

The index search can be the most time consuming part of flat or linear interpolation calculations. In large block diagrams, lookup table blocks often have the same input values as other lookup table blocks. If this is the case in your block diagram, you can obtain a large savings in computation time by making the breakpoints common to all tables. This savings is obtained by using one set of PreLook-Up Index Search blocks to perform the searches once for all tables, so that only the interpolation remains to be calculated. Figure 8-6 is an example of a block diagram that can be optimized by this method.

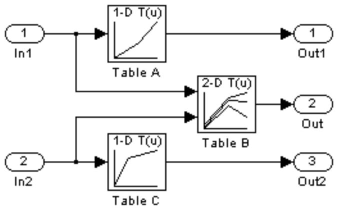


Figure 8-6: Before Optimization

Assume that Table A's breakpoints are the same as Table B's first input breakpoints, and that Table C's breakpoints are the same as Table B's second input breakpoints.

A 50% reduction in index search time is obtained by pulling these common breakpoints out into a pair of PreLook-Up Index Search blocks, and using Interpolation (n-D) Using PreLook-Up Index Search blocks to perform the interpolation. Figure 8-7 shows the optimized block diagram.

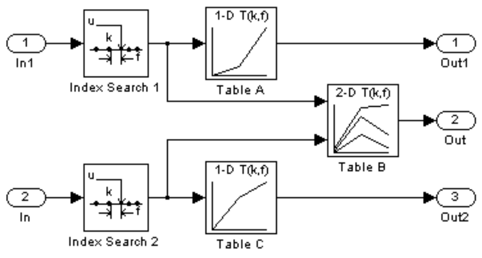


Figure 8-7: After Optimization

In Figure 8-7, the Look-Up Table (n-D) blocks have been replaced with Interpolation (n-D) Using PreLook-Up blocks.The PreLook-Up Index Search blocks have been added to perform the index searches separately from the interpolations, in order to realize the savings in computation time.

In large controllers and simulations, it is not uncommon for hundreds of multidimensional tables to rely on a dozen or so breakpoint sets. Using the optimization technique shown in this example, you can greatly increase the efficiency of your application.

Accumulators

Simulink recognizes the block diagram shown in Figure 8-8 as an accumulator. An accumulator construct — comprising a Constant block, a Sum block, and feedback through a Unit Delay block — is recognized anywhere across a block diagram, or within subsystems at lower levels.

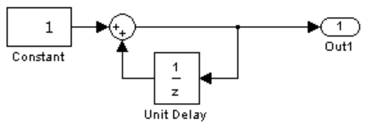
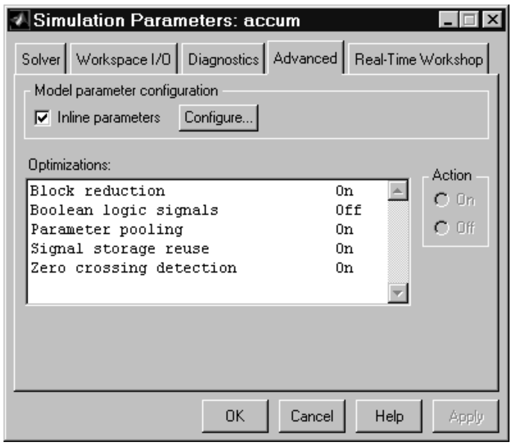


Figure 8-8: An Accumulator Algorithm

By using the **Block reduction** option, you can significantly optimize code generated from an accumulator. Turn this option on in the Advanced page of the Simulink **Simulation parameters** dialog, as shown in Figure 8-9.



Model parameter configuration

☑ Inline parameters

Optimizations:

```
Block reduction             On
Boolean logic signals       Off
Parameter pooling           On
Signal storage reuse        On
Zero crossing detection     On
```

Figure 8-9: Block Reduction Option

With the **Block reduction** option on, Simulink creates a synthesized block, `Sum_sythesized_accumulator`. This synthesized block replaces the block diagram of Figure 8-8, resulting in a simple increment calculation.

```
/* Compute block outputs */
void MdlOutputs(int_T tid)
{
  /* UnadornAccum Block: <Root>/Sum_sythesized_accumulator */
```

```
    rtB.Sum_sythesized_accumulator++;

    /* Outport Block: <Root>/Out1 */
    rtY.Out1 = rtB.Sum_sythesized_accumulator;
  }
```

With **Block reduction** turned off, the generated code reflects the block diagram more literally, but less efficiently.

```
  /* Compute block outputs */
  void MdlOutputs(int_T tid)
  {
    /* local block i/o variables */
    real_T rtb_Unit_Delay;

    /* UnitDelay Block: <Root>/Unit Delay */
    rtb_Unit_Delay = rtDWork.Unit_Delay_DSTATE;

    /* Sum Block: <Root>/Sum */
    rtB.Sum = rtC_Constant + rtb_Unit_Delay;

    /* Outport Block: <Root>/Out1 */
    rtY.Out1 = rtB.Sum;
  }

  /* Perform model update */
  void MdlUpdate(int_T tid)
  {
    /* UnitDelay Block: <Root>/Unit Delay */
    rtDWork.Unit_Delay_DSTATE = rtB.Sum;
  }
```

Use of Data Types

In most processors, the use of integer data types can result in a significant reduction in data storage requirements, as well as a large increase in the speed of operation. You can achieve large performance gains on most processors by identifying those portions of your block diagram that are really integer calculations (such as accumulators), and implementing them with integer data types.

Floating-point DSP targets are an obvious exception to this rule.

The accumulator from the previous example used 64-bit floating-point calculations by default. The block diagram in Figure 8-9 implements the accumulator with 16-bit integer operations.

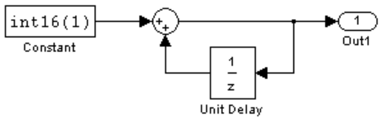


Figure 8-10: Accumulator Implemented with 16-bit Integers

If the **Saturate on integer overflow** option of the Sum block is turned off, the code generated from the integer implementation looks the same as code generated from the floating-point block diagram. However, since `Sum_sythesized_accumulator` is performing integer arithmetic internally, the accumulator executes more efficiently.

Note that, by default, the **Saturate on integer overflow** option is on. This option generates extra error-checking code from the integer implementation, as in the following example.

```
void MdlOutputs(int_T tid)
{
  /* UnadornAccum Block: <Root>/Sum_sythesized_accumulator */
  {
    int16_T tmpVar = rtB.Sum_sythesized_accumulator;
    rtB.Sum_sythesized_accumulator = tmpVar + (1);
    if ((tmpVar >= 0) && ((1) >= 0) &&
       (rtB.Sum_sythesized_accumulator < 0)) {
      rtB.Sum_sythesized_accumulator = MAX_int16_T;
    } else if ((tmpVar < 0) && ((1) < 0) &&
              (rtB.Sum_sythesized_accumulator >= 0)) {
      rtB.Sum_sythesized_accumulator = MIN_int16_T;
    }
  }

  /* Outport Block: <Root>/Out1 */
```

```
    rtY.Out1 = rtB.Sum_sythesized_accumulator;
  }
```

The floating-point implementation would not have generated the saturation error checks, which apply only to integers. When using integer data types, consider whether or not you need to generate saturation checking code.

Figure 8-11 shows an efficient way to add reset capability to the accumulator. When `resetSig` is greater than or equal to the threshold of the Switch block, the Switch block passes the reset value (0) back into the accumulator.

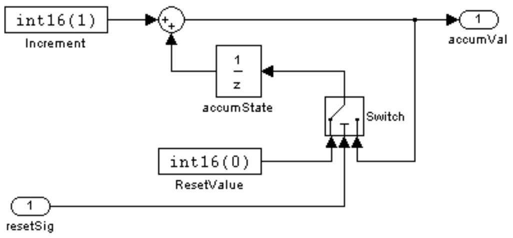


Figure 8-11: Integer Accumulator with Reset via External Input

The size of the resultant code is minimal. The code uses no floating-point operations.

```
  /* Compute block outputs */
  void MdlOutputs(int_T tid)
  {
    /* local block i/o variables */
    int16_T rtb_temp3;

    /* UnitDelay Block: <Root>/accumState */
    rtb_temp3 = rtDWork.accumState_DSTATE;

    /* Sum Block: <Root>/Sum */
    {
      int16_T tmpVar1 = 0;
      int16_T tmpVar2;
```

```
        /* port 0 */
        tmpVar1 = rtC_Increment;
        /* port 1 */
        tmpVar2 = tmpVar1 + rtb_temp3;
        if ((tmpVar1 >= 0) && (rtb_temp3 >= 0) &&
            (tmpVar2 < 0)) {
          tmpVar2 = MAX_int16_T;
        } else if ((tmpVar1 < 0) && (rtb_temp3 < 0) &&
                   (tmpVar2 >= 0)) {
          tmpVar2 = MIN_int16_T;
        }
        rtb_temp3 = tmpVar2;
      }

      /* Outport Block: <Root>/accumVal */
      rtY.accumVal = rtb_temp3;

      /* Switch Block: <Root>/Switch */
      if (rtU.resetSig) {
        rtB.Switch = rtC_ResetValue;
      } else {
        rtB.Switch = rtb_temp3;
      }
    }

    /* Perform model update */
    void MdlUpdate(int_T tid)
    {
      /* UnitDelay Block: <Root>/accumState */
      rtDWork.accumState_DSTATE = rtB.Switch;
    }
```

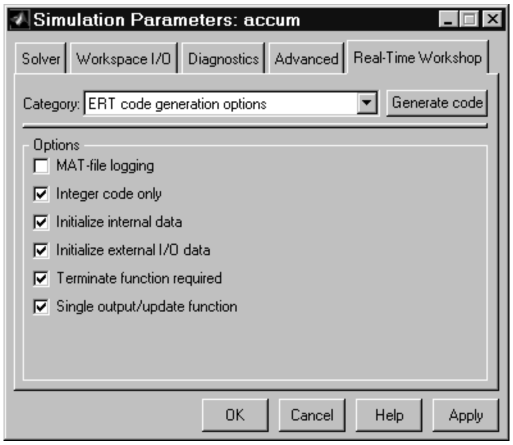
In this example, it would be easy to use an input to the system as the reset value, rather than a constant.

Generating Pure Integer Code

The Real-Time Workshop Embedded Coder target provides the **Integer code only** option to ensure that generated code contains no floating-point data or operations. When this option is selected, an error is raised if any noninteger data or expressions are encountered during compilation of the model. The error message reports the offending blocks and parameters.

If pure integer code generation is important to your design, you should consider using the Real-Time Workshop Embedded Coder target (or a target of your own, based on the Real-Time Workshop Embedded Coder target).

To use the **Integer code only** option, select **ERT code generation options** from the **Category** menu in the Real-Time Workshop page. Then enable the **Integer code only** option, as shown below.



Simulation Parameters: accum

Category: ERT code generation options

Options
- [ ] MAT-file logging
- [✓] Integer code only
- [✓] Initialize internal data
- [✓] Initialize external I/O data
- [✓] Terminate function required
- [✓] Single output/update function

The Real-Time Workshop Embedded Coder target offers many other optimizations. See Chapter 9, "Real-Time Workshop Embedded Coder" for further information.

Data Type Optimizations with Fixed-Point Blockset and Stateflow

The Fixed-Point Blockset (a separate product) is designed to deliver the highest levels of performance for noninteger algorithms on processors lacking floating-point hardware. The Fixed-Point Blockset's code generation in Real-Time Workshop implements calculations using a processor's integer operations. The code generation strategy maps the integer value set to a range of expected real world values to achieve the high efficiency.

Finite-state machine or flowchart constructs can often represent decision logic (or mode logic) efficiently. Stateflow (a separate product) provides these capabilities. Stateflow, which is fully integrated into Simulink, supports integer data-typed code generation.

Stateflow Optimizations

If your model contains Stateflow blocks, select the **Use Strong Data Typing with Simulink I/O** check box (on the **Chart Properties** dialog box) on a chart-by-chart basis.

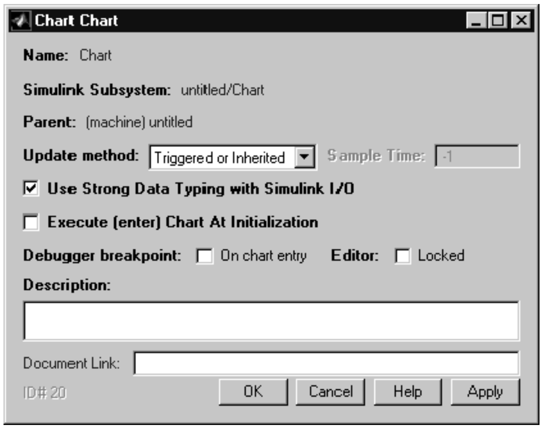


Chart Chart

Name: Chart

Simulink Subsystem: untitled/Chart

Parent: (machine) untitled

Update method: Triggered or Inherited **Sample Time:** -1

[✓] Use Strong Data Typing with Simulink I/O

[ ] Execute (enter) Chart At Initialization

Debugger breakpoint: [ ] On chart entry **Editor:** [ ] Locked

Description:

Document Link:

ID# 20

See the *Stateflow User's Guide* for more information about the **Chart Properties** dialog box.

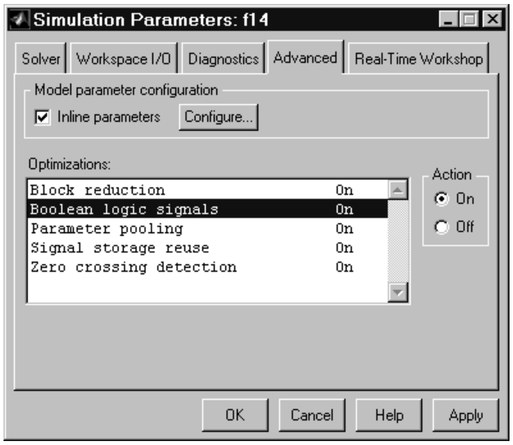
Simulation Parameters

Options on each page of the **Simulation Parameters** dialog box affect the generated code.

Advanced Page

- Turn on the **Signal storage reuse** option. The directs the Real-Time Workshop to store signals in reusable memory locations. It also enables the **Local block outputs** option (see “General Code Generation Options” on page 8-25).

  Disabling **Signal storage reuse** makes all block outputs global and unique, which in many cases significantly increases RAM and ROM usage.



[✓] Inline parameters

```
Block reduction                 On
Boolean logic signals           On
Parameter pooling               On
Signal storage reuse            On
Zero crossing detection         On
```

Action: [✓ On] Off

- Enable strict Boolean type checking by selecting the **Boolean logic signals** option.

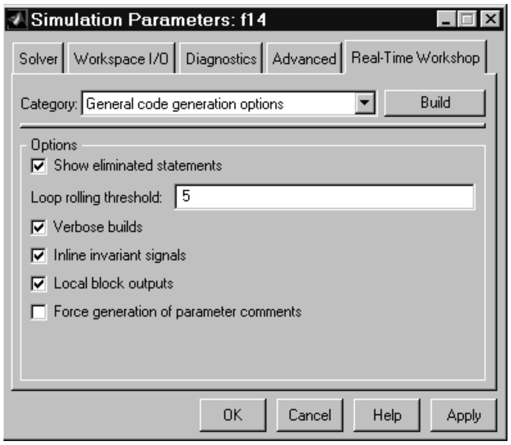
  Selecting this check box is recommended. Generated code will require less memory, because a Boolean signal typically requires one byte of storage while a double signal requires eight bytes of storage.

- Select the **Inline parameters** check box. Inlining parameters reduces global RAM usage, since parameters are not declared in the global parameters structure. Note that you can override the inlining of individual parameters by using the **Model Parameter Configuration** dialog box.

- Consider using the **Parameter pooling** option if you have multiple block parameters referring to workspace locations that are separately defined but structurally identical. See “Parameter Pooling Option” on page 3-24 for further information.

General Code Generation Options

To access these options, select **General code generation options** from the **Category** menu on the Real-Time Workshop page.



Category: General code generation options

☑ Show eliminated statements

Loop rolling threshold: 5

☑ Verbose builds

☑ Inline invariant signals

☑ Local block outputs

☐ Force generation of parameter comments

- Set an appropriate **Loop rolling threshold**. The loop rolling threshold determines when a wide signal should be wrapped into a `for` loop and when it should be generated as a separate statement for each element of the signal See “Loop Rolling Threshold Field” on page 3-10 details on loop rolling.
- Select the **Inline invariant signals** option. The Real-Time Workshop will not generate code for blocks with a constant (invariant) sample time.
- Select the **Local block outputs** option. Block signals will be declared locally in functions instead of being declared globally (when possible). You must turn on the **Signal storage reuse** option in the Advanced page to enable the **Local block outputs** check box.

Compiler Options

- If you do not require double precision for your application, define `real_T` as `float` in your template make file, or you can simply specify `-DREAL_T=float` after `make_rtw` in the **Make command** field.
- Turn on the optimizations for the compiler (e.g., `-O2` for `gcc`, `-Ot` for Microsoft Visual C).

Real-Time Workshop Embedded Coder

Introduction

The Real-Time Workshop Embedded Coder is a separate, add-on product for use with the Real-Time Workshop.

The Real-Time Workshop Embedded Coder provides a framework for the production of code that is optimized for speed, memory usage, and simplicity. The Real-Time Workshop Embedded Coder is intended for use in embedded systems.

The Real-Time Workshop Embedded Coder generates code in the `Embedded-C` format. Optimizations inherent in the `Embedded-C` code format include:

- Use of model-specific *real-time object* data structure rather than generic `SimStruct` significantly reduces code size and memory usage.
- Simplified calling interface reduces overhead. Model output and update functions are combined into a single routine.
- In-lined S-functions (required) reduce calling overhead and code size.
- Static memory allocation reduces overhead and promotes deterministic performance.

The Real-Time Workshop Embedded Coder supports the following key features:

- Integer only code generation
- Floating-point code generation
- Supports asynchronous interrupt-driven execution of models with either single or multiple sample rates
- Automatic generation of S-function wrappers, allowing you to validate the generated code in Simulink (Software-in-the-loop)
- Web-viewable code generation report describes code modules and helps to identify code generation optimizations relevant to your program

This chapter describes the components of the Real-Time Workshop Embedded Coder provided with Real-Time Workshop. It also describes options for optimizing your generated code, and for automatically generating an S-function wrapper that calls your Real-Time Workshop Embedded Coder generated code from Simulink. In addition, certain restrictions that apply to the use of the Real-Time Workshop Embedded Coder are discussed.

We assume you have read Chapter 6, “Program Architecture” and Chapter 7, “Models with Multiple Sample Rates” in this manual. Those chapters give a general overview of the architecture and execution of programs generated by Real-Time Workshop.

Data Structures and Code Modules

This section describes the main data structures of the Real-Time Workshop Embedded Coder. It also summarizes the code modules and header files that make up a Real-Time Workshop Embedded Coder program, and describes where to find them.

Real-Time Object

Unlike other Real-Time Workshop code formats, Real-Time Workshop Embedded Coder generated code does not employ the `SimStruct`. Instead, the Real-Time Workshop Embedded Coder uses a data structure called the *real-time object*. The real-time object, like the `SimStruct`, contains essential timing and scheduling data, as well as model information. The real-time object is much more compact than the `SimStruct`, achieving a significant reduction in code size. For example, the real-time object for a single-rate model typically requires 4 bytes.

Note that the real-time object is a model-specific data structure. The real-time object for a particular model is defined in *model*`_export.h`.

Your code should not reference fields of the real-time object directly. The Real-Time Workshop provides accessor macros for the real-time object. These macros are defined in *matlabroot*`/rtw/c/ert/ertformat.h`. The macros are syntactically and functionally identical to the `SimStruct` macros used with other code formats. If you are interfacing your code to a single model, you should refer to its real-time object generically as `RT_OBJ`, and use the macros to access `RT_OBJ`, as in the following code fragment.

```
#include "ertformat.h"
const char_T *errStatus = ssGetErrorStatus(RT_OBJ);
```

For an example of how to interface your code to the real-time objects of more than one model, see “How to Call the Entry Points Directly” on page 9-16.

The *logging object* is a subobject of the real-time object. This data structure is used in generated code if the **MAT-file logging** code generation option is enabled.

Code Modules

This section summarizes the code modules and header files of the Real-Time Workshop Embedded Coder, and describes where to find them.

Generated Code Modules

The Real-Time Workshop creates a build directory in your working directory to store generated source code. The build directory also contains object (`.obj`) files, a makefile, and other files created during the code generation process.

The default name of the build directory is *model*`_ert_rtw`. The build directory contains the following generated source code modules:

- **`model.c`**

  model`.c` defines all entry points to the generated code. These are:

  - *model*`_step` implements all computations required for one time step of your model.
  - *model*`_initialize` initializes the real-time object. If logging is enabled, it also initializes the logging object.
  - *model*`_terminate` performs any cleanup operations required after your program's main loop has stopped executing. Real-Time Workshop generates *model*`_terminate` if you select the **Terminate function required** code generation option (by default, this option is selected).

  A standard way to call these entry points is via the macros `MODEL_STEP`, `MODEL_INITIALIZE`, and `MODEL_TERMINATE`. These macros are defined in *matlabroot*`/rtw/c/ert/ertformat.h`. The following code fragment illustrates their use.

  ```
  #include "ertformat.h"
  MODEL_INITIALIZE(1);
  ```

- **`model_export.h`**

  model`_export.h` defines the real-time object and provides a public interface to the entry points of the model code. See "How to Call the Entry Points Directly" on page 9-16 for an example of how to use this interface.

- **`model.h`**

  This header defines parameters and data structures private to *model*`.c`.

- **`autobuild.h`** is generated only for use in test and simulation. This file is not required in production environments. You can replace

```
#include "autobuild.h"
```

with

```
#include "model_export.h"
```

Note that the Real-Time Workshop Embedded Coder code generation report contains further information about code modules generated during a build. (See "Generating a Code Generation Report" on page 9-23).

Main Program Module

Real-Time Workshop provides the module *matlabroot*/`rtw/c/ert/ert_main.c` as a template example for developing embedded applications. `ert_main.c` is not part of the generated code; it is provided as a basis for your custom modifications, and for use in simulation.

We recommend that you copy `ert_main.c` to your working directory and rename it to *model*`_ert_main.c` before making modifications. The Build process will create *model*`_ert_main.obj` in the build directory.

`ert_main.c` contains:

- `rt_OneStep`, a timer interrupt service routine (ISR). `rt_OneStep` calls `MODEL_STEP` to execute processing for one clock period of the model.
- A skeletal `main` function. As provided, `main` is useful in simulation only. You must modify `main` for real-time interrupt-driven execution.

"Program Execution" on page 9-9 contains a detailed discussion of the main module, as well as guidelines for modifying it to meet your requirements.

Utility Header Files

The following support header files are provided in the directory *matlabroot*/`rtw/c/ert`:

- **`ertformat.h`** defines accessor macros for the real-time object, as well as the Real-Time Workshop Embedded Coder entry-point macros.
- **`tmwtypes.h`** and **`simstruc_types.h`** define Real-Time Workshop data types. These headers are included by `ertformat.h`.
- **`log_object.h`** defines the logging object.
- **`log_macros.h`** defines macros that perform logging functions.

You can include these header files generically for all models. Inclusion of the header files adds no overhead to the generated code.

Table 9-1 summarizes the Real-Time Workshop Embedded Coder header files.

Table 9-1: Real-Time Workshop Embedded Coder Header Files

File	Directory
`autobuild.h`	Build directory
`ertformat.h`	*matlabroot*`/rtw/c/ert`
`log_object.h,log_macros.h` (Used only if MAT-file logging is enabled)	*matlabroot*`/rtw/c/ert`
model`.h`	Build directory
model`_export.h`	Build directory

User-Written Code Modules

Code that you write to interface with generated model code will include a customized main module, and may also include interrupt handlers, device driver blocks and other S-functions, and other supervisory or supporting code.

We recommend that you establish a working directory for your own code modules. Your working directory should be on the MATLAB path. You must also modify the Real-Time Workshop Embedded Coder template makefile and system target file so that the build process can find your source and object files. See Chapter 17, "Targeting Real-Time Systems" for information.

Program Execution

The Real-Time Workshop Embedded Coder generates self-sufficient, standalone programs that do not require an external real-time executive or operating system. The architecture of Real-Time Workshop Embedded Coder generated programs supports execution of models with either single or multiple sample rates.

If your application requires interfacing Real-Time Workshop Embedded Coder generated code to a real-time operating system, we suggest that you study the interface between Real-Time Workshop code and the VxWorks operating system, discussed in Chapter 12, “Targeting Tornado for Real-Time Applications.” OSEK example code is also available for study in *matlabroot*/`rtw/c/osek_leo`.

This section describes how Real-Time Workshop Embedded Coder programs execute, from the top level down to timer interrupt level.

Overview

The core of a Real-Time Workshop Embedded Coder program is typically the main loop. On each iteration, the main loop executes a background or null task and checks for a termination condition.

The main loop is periodically interrupted by a timer. The Real-Time Workshop function `rt_OneStep` is either installed as a timer ISR, or called from a timer ISR at each clock step.

The execution driver, `rt_OneStep`, sequences calls to the *model*`_step` function. The operation of `rt_OneStep` differs depending on whether the generating model is single-rate or multirate. In a single-rate model, `rt_OneStep` simply calls the *model*`_step` function. In a multirate model, `rt_OneStep` prioritizes and schedules execution of blocks according to the rates at which they run.

If your model includes device driver blocks, the *model*`_step` function will incorporate your in-lined driver code to perform I/O functions such as reading inputs from an analog-digital converter (ADC) or writing computed outputs to a digital-analog converter (DAC).

Main Program

Overview of Operation

The following pseudocode shows the execution of a Real-Time Workshop Embedded Coder main program.

```
main()
{
  Initialization (including installation of rt_OneStep as an
    interrupt service routine for a real-time clock).
  Initialize and start timer hardware
  Enable interupts
  While(not Error) and (time < final time)
    Background task.
  EndWhile
  Disable interrupts (Disable rt_OneStep from executing.)
  Complete any background tasks.
  Shutdown
}
```

The pseudocode is a design for a harness program to drive your model. The `ert_main.c` program, as shipped, only partially implements this design. You must modify it according to your specifications.

Guidelines for Modifying the Main Program

This section describes the minimal modifications you should make in your production version of `ert_main.c` to implement your harness program.

- After calling *model*`_initialize`:
  - Initialize target-specific data structures and hardware such as ADCs or DACs.
  - Install `rt_OneStep` as a timer ISR.
  - Initialize timer hardware.
  - Enable timer interrupts and start the timer.

Note The real-time object is not in a valid state until *model*_initialize has been called. Servicing of timer interrupts should not begin until *model*_initialize has been called.

- Replace the rt_OneStep call in the main loop with a background task call or null statement.
- On termination of main loop (if applicable):
  - Disable timer interrupts.
  - Perform target-specific cleanup such as zeroing DACs.
  - Detect and handle errors. Note that even if your program is designed to run indefinitely, you may need to handle severe error conditions such as timer interrupt overruns.

    You can use the macros ssGetErrorStatus, ssSetErrorStatus, and ssSetStopRequested to detect and signal errors, or to stop execution. These macros are documented in the *Writing S-Functions* manual.

rt_OneStep

Overview of Operation

The operation of rt_OneStep depends upon whether your model is single-rate or multirate. Code compilation is controlled by the symbol NUMST, which represents the number of sample times (i.e., rates) in the model. NUMST is defined to be 1 for a single-rate model; otherwise NUMST is greater than 1. NUMST is defined in the generated makefile *model*.mk.

Single-Rate Operation. The following pseudocode shows the design of rt_OneStep in a single-rate program.

```
rt_OneStep()
{
  Check for interrupt overflow or other error
  Enable "rt_OneStep" (timer) interrupt
  ModelStep-- Time step combines output,logging,update.
}
```

Single-rate `rt_OneStep` is designed to execute *model*`_step` within a single clock period. To enforce this timing constraint, `rt_OneStep` maintains and checks a timer overrun flag. On entry, timer interrupts are disabled until the overrun flag and other error conditions have been checked. If the overrun flag is clear, `rt_OneStep` sets the flag, and proceeds with timer interrupts enabled.

The overrun flag is cleared only upon successful return from *model*`_step`. Therefore, if `rt_OneStep` is reinterrupted before completing *model*`_step`, the reinterruption will be detected through the overrun flag.

Reinterruption of `rt_OneStep` by the timer is an error condition. If this condition is detected `rt_OneStep` signals an error and returns immediately. (Note that you can change this behavior if you want to handle the condition differently.)

Note that the design of `rt_OneStep` assumes that interrupts are disabled before `rt_OneStep` is called. `rt_OneStep` should be non-interruptible until the interrupt overflow flag has been checked.

Multirate Operation. The following pseudocode shows the design of `rt_OneStep` in a multirate program.

```
rt_OneStep()
{
  Check for base-rate interrupt overflow
  Enable "rt_OneStep" interrupt

  ModelStep(tid=0)     --base-rate time step.

  For i=1:NumTasks    -- iterate over sub-rate tasks
    Check for sub-rate interrupt overflow
    If (sub-rate task i is scheduled)
      ModelStep(tid=i)    --Sub-rate time step.
    EndIf
  EndFor
}
```

In a multirate system, `rt_OneStep` uses a prioritized, preemptive multitasking scheme to execute the different sample rates in your model.

The execution of blocks having different sample rates is broken into tasks. Each block that executes at a given sample rate is assigned a *task identifier* (`tid`), which associates it with a task that executes at that rate. Where there are `NUMST` tasks in the system, the range of task identifiers is 0..`NUMST`-1.

`rt_OneStep` prioritizes tasks, in descending order, by rate. The *base-rate task* is the task that runs at the fastest rate in the system (the hardware clock rate). The base-rate task has highest priority (`tid` 0) . The next fastest task (`tid` 1) has the next highest priority, and so on down to the slowest, lowest priority task (`tid` `NUMST`-1).

The slower tasks, running at submultiples of the base rate, are called *sub-rate* tasks.

On each invocation, `rt_OneStep` makes one or more calls to *model*`_step`, passing in the appropriate `tid`. The `tid` informs *model*`_step` that all blocks having that `tid` should execute. `rt_OneStep` always calls *model*`_step` `(tid = 0)` because the base-rate task must execute on every clock step.

On each clock tick, `rt_OneStep` also maintains scheduling counters and *event flags* for each sub-rate task. Both the counters and the event flags are implemented as arrays, indexed on `tid`.

The counters are, in effect, clock rate dividers that count up the sample period associated with each sub-rate task. The event flags indicate whether or not a given task is scheduled for execution. When a counter indicates that a task's sample period has elapsed, `rt_OneStep` sets the event flag for that task.

After updating its scheduling data structures and stepping the base-rate task, `rt_OneStep` iterates over the scheduling flags in `tid` order, calling *model*`_step(tid)` for any task whose flag is set. This ensures that tasks are executed in order of priority.

The event flag array and loop variables used by `rt_OneStep` are stored as local (stack) variables. This ensures that `rt_OneStep` is reentrant. If `rt_OneStep` is reinterrupted, higher priority tasks will preempt lower priority tasks. Upon return from interrupt, lower priority tasks will resume in the previously scheduled order.

Multirate `rt_OneStep` also maintains an array of timer overrun flags. `rt_OneStep` detects timer overrun, per task, by the same logic as single-rate `rt_OneStep`.

Note that the design of `rt_OneStep` assumes that interrupts are disabled before `rt_OneStep` is called. `rt_OneStep` should be non-interruptible until the base-rate interrupt overflow flag has been checked (see pseudo-code above).

Guidelines for Modifying rt_OneStep

`rt_OneStep` does not require extensive modification. The only required modification is to re-enable interrupts after the overrun flag(s) and error conditions have been checked. Comments in `rt_OneStep` indicate the appropriate place to add your code.

In multirate `rt_OneStep`, you can improve performance by unrolling `for` and `while` loops.

You may also want to replace the `MODEL_STEP` macro call(s) with model-specific call(s). If so, see "How to Call the Entry Points Directly" on page 9-16.

In addition, you may choose to modify the overrun behavior to continue execution after error recovery is complete.

You should not modify the way in which the counters are set in `rt_OneStep`. The `rt_OneStep` timing data structures (including the real-time object) and logic are critical to correct operation of any Real-Time Workshop Embedded Coder program.

Model Entry Points

model_step

Calling Interface. The `MODEL_STEP` macro is the standard way to call your model's generated step function.

In a single-rate model, the macro expands to a function call with the prototype

```
void model_step(void);
```

In a multirate model, the macro expands to a function call with the prototype

```
void model_step(int_T tid);
```

where `tid` is a task identifier. The `tid` is determined by logic within `rt_OneStep`. (See "rt_OneStep" on page 9-11.)

Operation. *model*`_step` combines the model output and update functions into a single routine. *model*`_step` is designed to be called at interrupt level from `rt_OneStep`, which is assumed to be invoked as a timer ISR.

Single-Rate Operation. In a single-rate model, *model*`_step` computes the current value of all blocks. If logging is enabled, *model*`_step` updates logging variables. If the model's stop time is finite, *model*`_step` signals the end of execution when the current time equals the stop time.

Multirate Operation. In a multirate model, *model*`_step` execution is almost identical to single-rate execution, except for the use of the task identifier (`tid`) argument.

The caller (`rt_OneStep`) assigns each block a `tid`. (See "rt_OneStep" on page 9-11.) *model*`_step` uses the `tid` argument to determine which blocks have a sample hit (and therefore should execute).

Note If the model's stop time is set to `inf`, or if logging is disabled, *model*`_step` does not check the current time against the stop time. Therefore, the program runs indefinitely.

---

model_initialize

Calling Interface. The `MODEL_INITIALIZE` macro is the standard way to call your model's generated initialization code. The macro expands to a function call with the prototype

```
void model_initialize(boolean_T firstTime);
```

Operation. If `firstTime` equals 1 (`TRUE`), *model*`_initialize` initializes the real-time object and other data structures private to the model. If `firstTime` equals 0 (`FALSE`), *model*`_initialize` resets the model's states.

The generated code calls *model*`_initialize` once, passing in `firstTime` as `1`(`TRUE`).

model_terminate

Calling Interface. The `MODEL_TERMINATE` macro is a standard way to call your model's generated termination code. The macro expands to a function call with the prototype

```
void model_terminate(void);
```

Operation. When *model*_terminate is called, blocks that have a terminate function execute their terminate code. If logging is enabled, *model*_terminate ends data logging. *model*_terminate should only be called once. If your application runs indefinitely, you do not need the *model*_terminate function.

If you do not require a terminate function, see "Basic Code Generation Options" on page 9–20 for information on using the **Terminate function required** option.

How to Call the Entry Points Directly

You can replace the generated macro calls with direct calls to the entry points. This is necessary when interfacing your code with code generated from more than one model. In such cases, the macro calls are ambiguous. Include *model*_export.h to make the entry points visible to your code, as in the following code fragment.

```
#include "modelA_Export.h" /* Make model A entry points visible */
#include "modelB_Export.h" /* Make model B entry points visible */

void myHandWrittenFunction(void)
{
  const char_T *errStatus;

  modelA_initialize(1); /* Call model A initializer */
  modelB_initialize(1); /* Call model B initializer */
  /* Refer to model A's real-time Object */
  errStatus = ssGetErrorStatus(modelA_rtO);
  /* Refer to model B's real-time Object */
  errStatus = ssGetErrorStatus(modelB_rtO);
}
```

Note If you are modifying *model*_step calls in multirate rt_OneStep, take care to pass in the correct tid argument. The first call, which steps the model for the base sample time, always passes in tid 0. The calls made in the sub-rate loop always pass in the loop variable i.

Automatic S-Function Wrapper Generation

An S-function wrapper is an S-function that calls your C code from within Simulink. S-function wrappers provide a standard interface between Simulink and externally written code, allowing you to integrate your code into a model with minimal modification. For a complete description of wrapper S-functions, see the *Writing S-Functions* manual.

Using the Real-Time Workshop Embedded Coder **Create Simulink (S-Function) block** option, you can build, in one automated step:

- A noninlined C MEX S-function wrapper that calls Real-Time Workshop Embedded Coder generated code.
- A model containing the generated S-function block, ready for use with other blocks or models.

This is useful for code validation and simulation acceleration purposes.

When **Create Simulink (S-Function) block** option is on, the Real-Time Workshop generates an additional source code file, *model*`_sf.c`, in the build directory. This module contains the S-function that calls the Real-Time Workshop Embedded Coder code that you deploy. This S-function can be used within Simulink.

The build process then compiles and links *model*`_sf.c` with *model*`.c` and the other Real-Time Workshop Embedded Coder generated code modules, building a MEX-file.The MEX-file is named *model*`_sf.`*mexext*. (*mexext* is the file extension for MEX-files on your platform, as given by the MATLAB `mexext` command.) The MEX-file is stored in your working directory. Finally, the Real-Time Workshop creates and opens an untitled model containing the generated S-Function block.

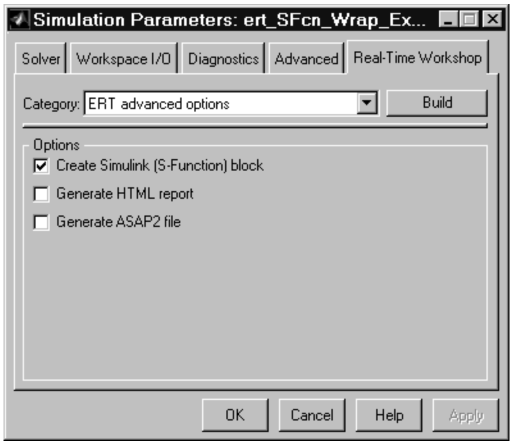
Limitations

It is not possible to create multiple instances of a Real-Time Workshop Embedded Coder generated S-Function block within a model, because the code uses static memory allocation.

To generate an S-function wrapper for your Real-Time Workshop Embedded Coder code:

1 Select the Real-Time Workshop tab of the **Simulation Parameters** dialog box. Then select **ERT advanced options** from the **Category** menu.

2 Enable the **Create Simulink (S-Function) block** option, as shown.



Category: ERT advanced options

Options

☑ Create Simulink (S-Function) block

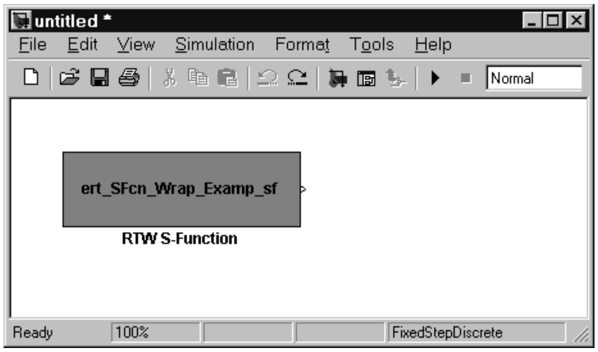
☐ Generate HTML report

☐ Generate ASAP2 file

3 Configure the other code generation options as required.

4 Click the **Build** button.

5 When the build process completes, an untitled model window opens. This model contains the generated S-Function block.



6 Save the new model.

7 The generated S-Function block is now ready to use with other blocks or models in Simulink.

Optimizing the Generated Code

The Real-Time Workshop Embedded Coder features a number of code generation options that can help you further optimize the generated code. The Real-Time Workshop Embedded Coder can also produce a code generation report in HTML format. This report documents code modules and helps you to identify optimizations that are relevant to your model.

"Basic Code Generation Options" on page 9-20 documents code generation options you can use to improve performance and reduce code size.

"Generating a Code Generation Report" on page 9-23 describes how to generate and use a code generation report.

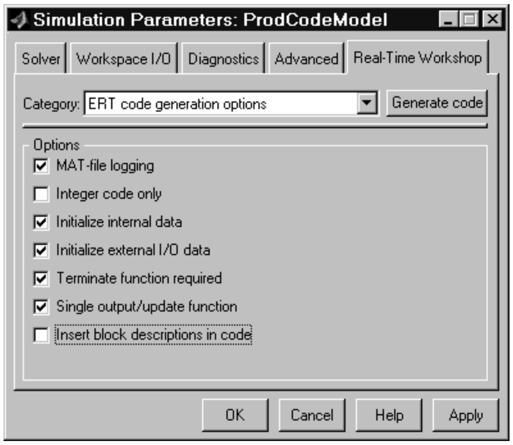
"Controlling Stack Space Allocation" on page 9-25 discusses options related to the storage of signals.

Please see Chapter 8, "Optimizing the Model for Code Generation" for information about code optimization techniques common to all code formats.

Basic Code Generation Options

To access the basic code generation options, select the Real-Time Workshop tab of the **Simulation Parameters** dialog box. Then select **ERT code generation options** from the **Category** menu.

Figure 9-1 displays the basic code generation options for the Real-Time Workshop Embedded Coder.



Category: ERT code generation options

Options
- [✓] MAT-file logging
- [ ] Integer code only
- [✓] Initialize internal data
- [✓] Initialize external I/O data
- [✓] Terminate function required
- [✓] Single output/update function
- [ ] Insert block descriptions in code

Figure 9-1: Basic Code Generation Options

Note `ert_main.c` contains certain compile-time error checks on code generation options, intended for use during simulation only. You should remove these error checks, as instructed below, from your production version of `ert_main.c`

Setting the code generation options as follows will result in more highly optimized code:

- Deselect the **MAT-file logging** check box.

  Also, remove or comment out the `#if MAT_FILE...` error check in your production version of `ert_main.c`.

  Note that disabling logging causes the program to run indefinitely, regardless of the setting of the model's stop time.

- Deselect the **Initialize internal data** and **Initialize external I/O data** check boxes.

  Initializing the internal and external data is a precaution and may not be necessary for your application. Many embedded application environments initialize all RAM to zero at startup. Therefore, reinitializing RAM to zero is redundant.

Note that nonzero initialization of your program's data structures is still performed when **Initialize internal data** and **Initialize external I/O data** are selected.

- Deselect the **Terminate function required** check box if you do not require a terminate function for your model.

  Also, remove or comment out the following in your production version of `ert_main.c`:

  - The `#if TERMFCN...` error check
  - The call to `MODEL_TERMINATE`
- Select the **Single output/update function** check box. Combining the output and update functions is the default. This option generates the *model*`_step` call, which reduces overhead and allows the Real-Time Workshop to use more local variables in the step function of the model.

  If you do *not* want to combine output and update functions, make the following changes in your production version of `ert_main.c`:

  - Replace calls to `MODEL_STEP` with calls to *model*`_output` and *model*`_update`.
  - Remove the `#if ONESTEPFCN...` error check.
- If your application uses only integer arithmetic, select the **Integer code only** option to ensure that generated code contains no floating-point data or operations. When this option is selected, an error is raised if any noninteger data or expressions are encountered during code generation. The error message reports the offending blocks and parameters.

Generating Code from Subsystems

Note that when generating code from a subsystem, it is recommended that you set the sample times of all subsystem inports explicitly.

Generating Block Comments

When the **Insert block descriptions in code** option is selected, comments are inserted into the code generated for any blocks that have text in their **Description** fields.

To generate block comments:

1 Right-click on the block you want to comment. Select **Block Properties** from the context menu. The **Block Properties** dialog box opens.

2 Type the comment into the **Description** field.

3 Select the **Insert block descriptions in code** option in the **ERT code generation options** category of the Real-Time Workshop page.

Note For virtual blocks or blocks that have been removed due to block reduction optimizations, no comments are generated.

Generating a Code Generation Report

The Real-Time Workshop Embedded Coder code generation report is an HTML file consisting of several sections:

• The Summary section lists version and date information, TLC options used in code generation, and Simulink model settings.

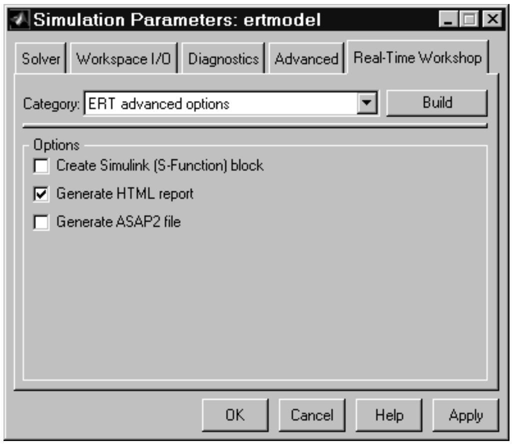
• The Optimizations section lists the optimizations used during the build, and also those that are available. If options were chosen that generated non-optimal code, they are marked in red. This section can help you select options that will better optimize your code.

• The Generated Source Files section contains a table of source code files generated from subsystems (if any) in your model. Each row contains a hyperlink to the relevant subsystem. You can click on the hyperlink to view the subsystem in a Simulink model window.

To generate a code generation report:

1 Select the Real-Time Workshop tab of the **Simulation Parameters** dialog box. Then select **Advanced code generation options** from the **Category** menu.

2 Select **Generate HTML report**, as shown in this picture.



Simulation Parameters: ertmodel

Solver | Workspace I/O | Diagnostics | Advanced | Real-Time Workshop

Category: ERT advanced options

Build

Options

- [ ] Create Simulink (S-Function) block
- [x] Generate HTML report
- [ ] Generate ASAP2 file

OK | Cancel | Help | Apply

3 Click the **Build** button.

4 The Real-Time Workshop writes the code generation report file in the build directory. The file is named *model*`_codegen_rpt.html`.

5 The Real-Time Workshop automatically opens the MATLAB Help Browser and displays the code generation report.

Alternatively, you can view the code generation report in your Web browser.

Controlling Stack Space Allocation

The Real-Time Workshop offers a number of options that let you control how signals in your model are stored and represented in the generated code. This section discusses options that:

- Let you control whether signal storage is declared in global memory space, or locally in functions (i.e., in stack variables).
- Control the allocation of stack space when using local storage.

For a complete discussion of signal storage options, see “Signals: Storage, Optimization, and Interfacing” on page 3-65.

If you want to store signals in stack space, you must turn the **Local block outputs** option on. To do this:

1. Select the Advanced tab of the **Simulation Parameters** dialog box. Make sure that the **Signal storage reuse** is on. If **Signal storage reuse** is off, the **Local block outputs** option is not available.
2. Click **Apply** if necessary.
3. Select the Real-Time Workshop tab of the **Simulation Parameters** dialog box.
4. From the **Category** menu, select **General code generation options**.
5. Check the **Local block outputs** option. Click **Apply** if necessary.

Your embedded application may be constrained by limited stack space. When the **Local block outputs** option is on, you can limit the use of stack space by using the following Target Language Compiler variables:

- `MaxStackSize`: the total allocation size of local variables that are declared by all functions in the entire model may not exceed `MaxStackSize` (in bytes). `MaxStackSize` can be any positive integer. If the total size of local variables exceeds this maximum, the Target Language Compiler will allocate the remaining variables in global, rather than local, memory.
- `MaxStackVariableSize`: limits the size of any local variable declared in a function to `N` bytes, where `N>0`. A variable whose size exceeds

`MaxStackVariableSize` will be allocated in global, rather than local, memory.

To set either of these variables, use `assign` statements in the system target file (`ert.tlc`), as in the following example.

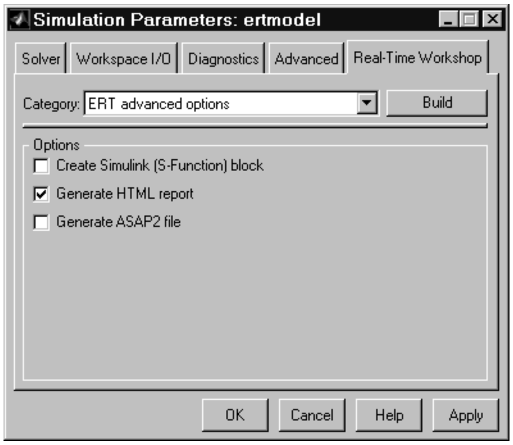
```
%assign MaxStackSize = 4096
```

It is recommended that you write your `%assign` statements in the `Configure RTW code generation settings` section of the system target file. The `%assign` statement is described in the *Target Language Compiler Reference Guide*.

Advanced Code Generation Options

To access the advanced Real-Time Workshop Embedded Coder code generation options, select the Real-Time Workshop tab of the **Simulation Parameters** dialog box. Then select **Advanced code generation options** from the **Category** menu.

Figure 9-2 displays the advanced code generation options for the Real-Time Workshop Embedded Coder.



Category: ERT advanced options

- [ ] Create Simulink (S-Function) block
- [✓] Generate HTML report
- [ ] Generate ASAP2 file

Figure 9-2: Advanced Code Generation Options

Create Simulink (S-Function) Block

See “Automatic S-Function Wrapper Generation” on page 9-17 for information on this feature.

Generate HTML Report

See “Generating a Code Generation Report” on page 9-23 for information on this feature.

Generate ASAP2 File

The Real-Time Workshop Embedded Coder **Generate ASAP2 File** code generation option lets you export an ASAP2 file containing information about your model during the code generation process.

The ASAP2 file generation process requires information about your model's parameters and signals. Some of this information is contained in the model itself. The rest must be supplied by using Simulink data objects with the necessary properties. Simulink provides two data classes to assist you in providing the necessary information. See “Generating ASAP2 Files” in the Real-Time Workshop online documentation for information on this feature.

Requirements and Restrictions

- By definition, a Real-Time Workshop Embedded Coder program functions in discrete time. Your model must use the following solver options:
  - Solver type: fixed-step
  - Algorithm: discrete (no continuous states)
- You must select the `SingleTasking` or `Auto` solver mode when the model is single-rate. The following table indicates permitted solver modes for single-rate and multirate models.

Table 9-2: Permitted Solver Modes for Real-Time Workshop Embedded Coder-Targeted Models

Mode	Single-rate	Multirate
`SingleTasking`	Allowed	Allowed
`MultiTasking`	Disallowed	Allowed
`Auto`	Allowed (defaults to `SingleTasking`)	Allowed (defaults to `MultiTasking`)

- You cannot have any continuous time blocks in your model. (See “Unsupported Blocks” on page 9-30.).
- If you are designing a program that is intended to run indefinitely, you should not use blocks that have a dependency on absolute time. See Appendix A for a list of blocks that depend on absolute time.
- You must inline all S-functions with a corresponding Target Language Compiler (TLC) file. The reason for this is that Real-Time Workshop Embedded Coder generated code uses the real-time object, rather than the `SimStruct`. Since noninlined S-functions require reference to the `SimStruct`, they cannot be used in Real-Time Workshop Embedded Coder generated programs. See the *Writing S-Functions* manual for information about inlining S-functions.

Unsupported Blocks

The `Embedded-C` format does not support the following built-in blocks:

- Continuous
  - No blocks in this library are supported
- Discrete
  - First-Order Hold
- Functions and Tables
  - MATLAB Fcn
  - The following S-functions: M-file and Fortran S-functions, or noninlined C-MEX S-functions that call into MATLAB.
- Math
  - Algebraic Constraint
  - Matrix Gain
- Nonlinear
  - Rate Limiter
- Sinks
  - XY Graph
  - Display
- Sources
  - Clock
  - Chirp Signal
  - Pulse Generator
  - Ramp
  - Repeating Sequence
  - Signal Generator

System Target File and Template Makefiles

The Real-Time Workshop Embedded Coder system target file is `ert.tlc`.

The Real-Time Workshop provides template makefiles for the Real-Time Workshop Embedded Coder in the following development environments:

- `ert_bc.tmf` — Borland C
- `ert_lcc.tmf` — LCC compiler
- `ert_unix.tmf` — UNIX host
- `ert_vc.tmf` — Visual C
- `ert_watc.tmf` — Watcom C

The S-Function Target

Introduction

Using the S-function target, you can build an S-function component and use it as an S-Function block in another model. The S-function code format used by the S-function target generates code that conforms to the Simulink C MEX S-function application programming interface (API). Applications of this format include:

- Conversion of a model to a component. You can generate an S-Function block for a model, `m1`. Then, you can place the generated S-Function block in another model, `m2`. Regenerating code for `m2` does not require regenerating code for `m1`.
- Conversion of a subsystem to a component. By extracting a subsystem to a separate model, and generating an S-Function block from that model, you can create a reusable component from the subsystem. See "Creating an S-Function Block from a Subsystem" on page 10-4 for an example of this procedure.
- Speeding up simulation. In many cases, an S-function generated from a model performs more efficiently than the original model.
- Code reuse. You can incorporate multiple instances of one model inside another without replicating the code for each instance. Each instance will continue to maintain its own unique data.

The S-function target generates noninlined S-functions. You can generate an executable from a model that contains generated S-functions by using the generic real-time or real-time malloc targets. You cannot use the Real-Time Workshop Embedded Coder target for this purpose, since it requires inlined S-functions.

You can place a generated S-Function block into another model from which you can generate another S-function format. This allows any level of nested S-functions.

Note that sample times propagation for the S-function code format is slightly different from the other code formats. A generated S-Function block will inherit its sample time from the model in which it is placed if no blocks in the original model specify their sample times.

Intellectual Property Protection

In addition to the technical applications of the S-function target listed above, you can use the S-function target to protect your designs and algorithms. By generating an S-function from a proprietary model or algorithm, you can share the model's functionality without providing the source code. You need only provide the binary `.dll` or MEX-file object to users.

Creating an S-Function Block from a Subsystem

This section demonstrates how to extract a subsystem from a model and generate a reusable S-function component from it.

Figure 10-1 illustrates `SourceModel`, a simple model that inputs signals to a subsystem. Figure 10-2 illustrates the subsystem, `SourceSubsys`. The signals, which have different widths and sample times, are:

- A Step block with sample time 1
- A Sine Wave block with sample time 0.5
- A Constant block whose value is the vector [-2 3]

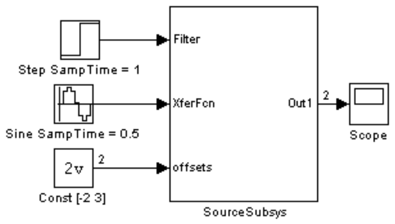


Figure 10-1: SourceModel

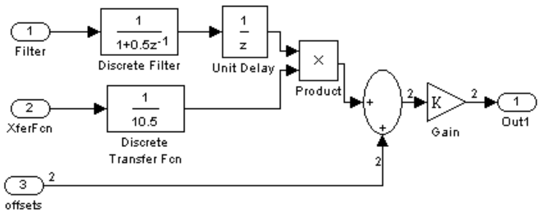


Figure 10-2: SourceSubsys

Our objective is to extract `SourceSubsys` from the model and build an S-Function block from it, using the S-function target. We want the S-Function block to perform identically to the subsystem from which it was generated.

Note that in this model, `SourceSubsys` inherits sample times and signal widths from its input signals. If an S-Function block is built from `SourceSubsys`, without explicitly setting input widths and sample times, the new block will inherit its sample times and signal widths from the model in which it is placed.

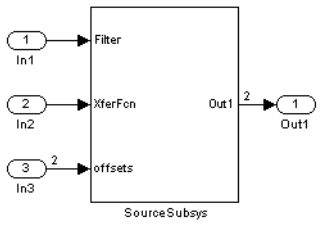
In this example, however, we want the S-Function block to retain the properties of `SourceSubsys` as it exists in `SourceModel`. Before building the subsystem as a separate S-function component, the inport sample times and widths must be set explicitly. In addition, the solver parameters of the S-function component must be the same as those of the original model. This ensures that the generated S-function component will operate identically to the original subsystem.

To build `SourceSubsys` as an S-function component:

1. Create a new model and copy/paste `SourceSubsys` into the empty window.

2. Set the signal widths and sample times of inports inside `SourceSubsys` such that they match those of the signals in the original model. Inport 1, `Filter`, has a width of 1 and a a sample time of 1. Inport 2, `Xferfcn`, has a width of 1 and a sample time of 0.5. Inport 3, `offsets`, has a width of 2 and an inherited sample time of 1.

3 The generated S-Function block should have three inports and one outport. Connect inports and an outport to `SourceSubsys`, as shown below.



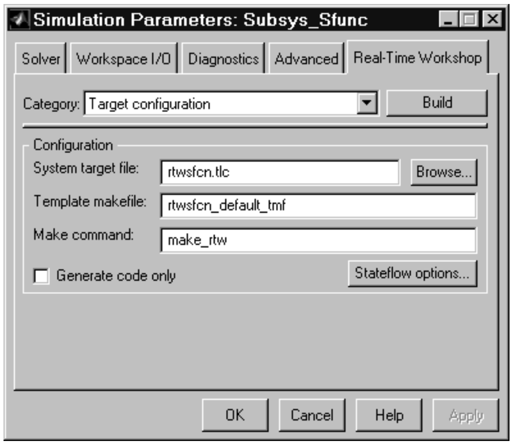
Note that the correct signal widths and sample times propagate to these ports.

4 Set the solver type, mode, and other solver parameters such that they are identical to those of the source model.

5 Save the new model.

6 Open the **Simulation Parameters** dialog and click the Real-Time-Workshop tab. On the Real-Time-Workshop page, select **Target configuration** from the **Category** menu.

7 Click the **Browse** button to open the System Target Browser. Select the S-function target in the System Target Browser, and click **OK**. The Real-Time-Workshop page parameters should appear as below.


Simulation Parameters: Subsys_Sfunc

Solver | Workspace I/O | Diagnostics | Advanced | Real-Time Workshop

Category: Target configuration — Build

Configuration

System target file: rtwsfcn.tlc — Browse...

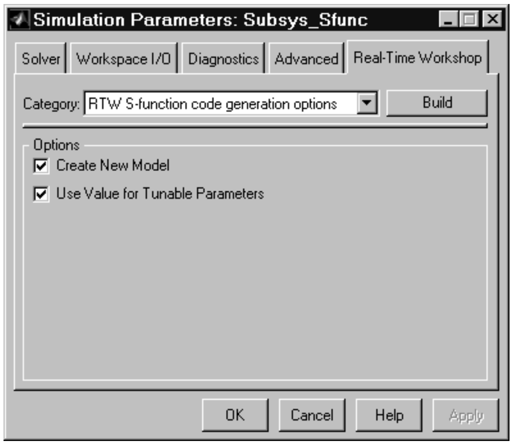
Template makefile: rtwsfcn_default_tmf

Make command: make_rtw

☐ Generate code only — Stateflow options...

OK | Cancel | Help | Apply

8 Select **RTW S-function code generation options** from the **Category** menu. Make sure that **Create New Model** is selected.


Simulation Parameters: Subsys_Sfunc

Solver | Workspace I/O | Diagnostics | Advanced | Real-Time Workshop

Category: RTW S-function code generation options — Build

Options

☑ Create New Model

☑ Use Value for Tunable Parameters

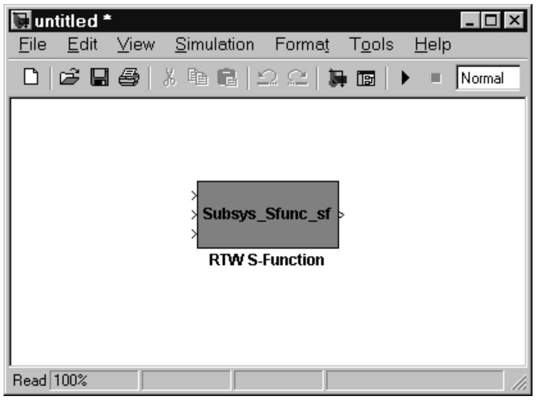
OK | Cancel | Help | Apply

When this option is selected, the build process creates a new model after it builds the S-function component. The new model contains an S-Function block, linked to the S-function component.

9 Click **Apply** if necessary and select **Target configuration** from the **Category** menu.

10 Click **Build**.

11 Real-Time Workshop builds the S-function component in the working directory. After the build, a new model window displays.



12 You can now copy the Real-Time Workshop S-Function block from the new model and use it in other models or in a library. Figure 10-3 shows the S-Function block plugged in to the original model. Given identical input signals, the S-Function block will perform identically to the original subsystem.

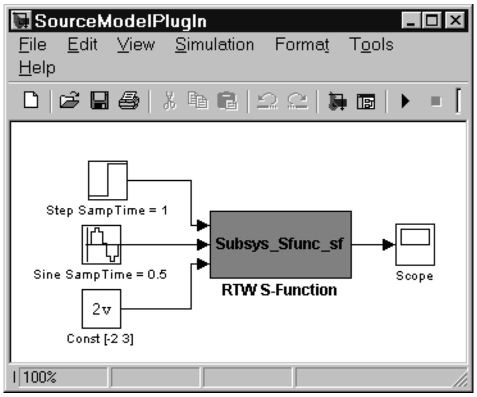


Figure 10-3: Generated S-Function Plugged into SourceModel

Note that the speed at which the S-Function block executes is typically faster than the original model. This difference in speed is more pronounced for larger and more complicated models. By using generated S-functions, you can increase the efficiency of your modeling process.

Tunable Parameters in Generated S-Functions

You can utilize tunable parameters in generated S-functions in two ways:

- Use the **Generate S-function** feature (see “Automated S-Function Generation” on page 10-12).

  or

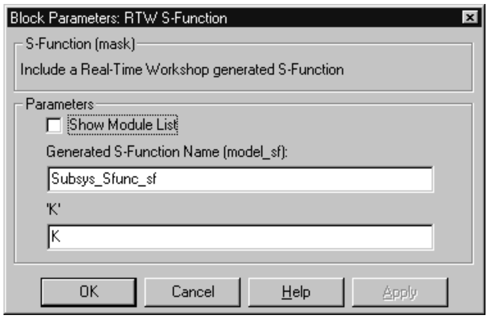
- Use the **Model Parameter Configuration** dialog (see “Parameters: Storage, Interfacing, and Tuning” on page 3-51) to declare desired block parameters tunable.

  Block parameters that are declared tunable with the `auto` storage class in the source model become tunable parameters of the generated S-function.

  Note that these parameters do not become part of a generated `rtP` parameter data structure, as they would in code generated from other targets. Instead, the generated code accesses these parameters via MEX API calls such as `mxGetPr` or `mxGetData`. Your code should access these parameters in the same way.

  For further information on MEX API calls, see *Writing S-Functions* and the *MATLAB Application Program Interface Guide.*

S-Function blocks created via the S-function target are automatically masked. The mask displays each tunable parameter in an edit field. By default, the edit field displays the parameter by variable name, as in the following example.



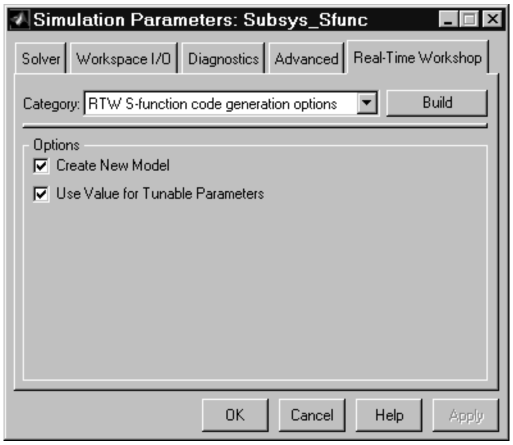
Include a Real-Time Workshop generated S-Function

Show Module List: [ ]

Generated S-Function Name (model_sf): Subsys_Sfunc_sf

'K': K

You can choose to display the value of the parameter rather than its variable name. To do this, select **Use Value for Tunable Parameters** in the **Options** section.



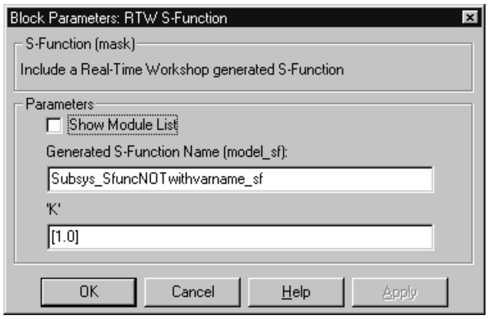
Category: RTW S-function code generation options

Options

☑ Create New Model

☑ Use Value for Tunable Parameters

When this option is chosen, the value of the variable (at code generation time) is displayed in the edit field, as in the following example.



S-Function (mask)

Include a Real-Time Workshop generated S-Function

Parameters

☐ Show Module List

Generated S-Function Name (model_sf):

Subsys_SfuncNOTwithvarname_sf

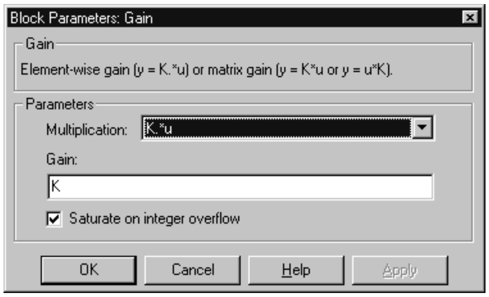
'K'

[1.0]

Automated S-Function Generation

The **Generate S-function** feature automates the process of generating an S-function from a subsystem. In addition, the **Generate S-function** feature presents a display of parameters used within the subsystem, and lets you declare selected parameters tunable.

As an example, consider `SourceSubsys`, the subsystem illustrated in Figure 10-2. Our objective is to automatically extract `SourceSubsys` from the model and build an S-Function block from it, as in the previous example. In addition, we want to set the gain factor of the Gain block within `SourceSubsys` to the workspace variable `K` (as illustrated below) and declare `K` as a tunable parameter.


Block Parameters: Gain

Gain

Element-wise gain (y = K.*u) or matrix gain (y = K*u or y = u*K).

Parameters

Multiplication: K.*u

Gain: K

☑ Saturate on integer overflow

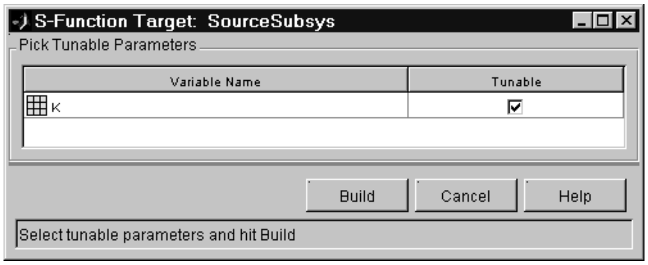
To auto-generate an S-function from `SourceSubsys` with tunable parameter `K`:

1 Click on the subsystem to select it.

2 Select **Generate S-function** from the **Real-Time Workshop** submenu of the **Tools** menu. This menu item is enabled when a subsystem is selected in the current model.

Alternatively, you can choose **Generate S-function** from the **Real-Time Workshop** submenu of the subsystem block's context menu.

3 A window displaying a list of the subsystem parameters opens.



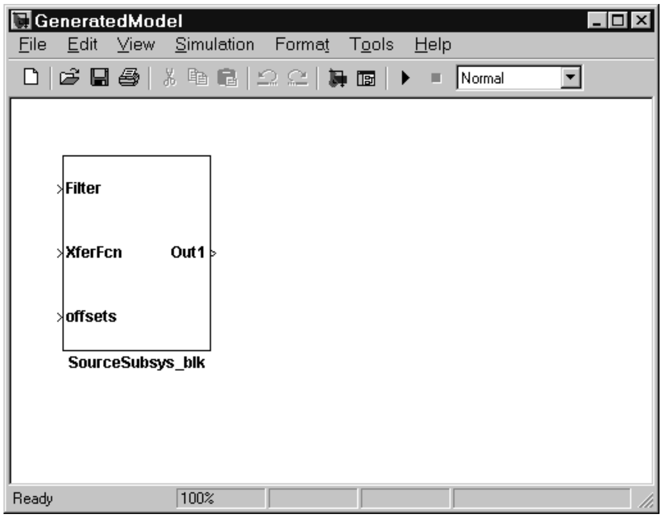
Variable Name	Tunable
K	✓

Select tunable parameters and hit Build

In the illustration above, the parameter K is declared tunable.

4 After selecting tunable parameters, click the **Build** button. This initiates code generation and compilation of the S-function, using the S-function target. The **Create New Model** option is automatically enabled.

5 The build process displays status messages in the MATLAB command window. When the build completes, the tunable parameters window closes, and a new untitled model window opens.



6 The model window contains an S-Function block, *subsys*_blk, where *subsys* is the name of the subsystem from which the block was generated.

   The generated S-function component, *subsys*, is stored in the working directory. The generated source code for the S-function is written to a build directory, *subsys*_sfcn_rtw.

7 Note that the untitled generated model does not persist, unless you save it via the **File** menu.

8 Note that the generated S-Function block has inports and outports whose widths and sample times correspond to those of the original model.

The following code fragment, from the mdlOutputs routine of the generated S-function code (in SourceSubsys_sf.c), illustrates how the tunable variable K is referenced via calls to the MEX API.

```
static void mdlOutputs(SimStruct *S, int_T tid)
...
real_T rtb_temp3[2];
...
/* Gain Block: <S1>/Gain */
rtb_temp3[0] *= ((*(real_T *)(mxGetData(K(S)))));
rtb_temp3[1] *= ((*(real_T *)(mxGetData(K(S)))));
```

Note In automatic S-function generation, the **Use Value for Tunable Parameters** option is always set to its default value (off).

Restrictions

- Hand-written S-functions without corresponding TLC files must contain exception-free code. For more information on exception-free code, refer to “Exception-Free Code” in *Writing S-Functions*.
- If you modify the source model that generated an S-Function block, the Real-Time Workshop does not automatically rebuild models containing the generated S-Function block.

Unsupported Blocks

The S-function format does not support the following built-in blocks:

- MATLAB Fcn Block
- S-Function blocks containing any of the following:
  - M-file S-functions
  - Fortran S-functions
  - C MEX S-functions that call into MATLAB
- Scope block
- To Workspace block

System Target File and Template Makefiles

The following system target file and template makefiles are provided for use with the S-function target.

System Target File

- `rtwsfcn.tlc`

Template Makefiles

- `rtwsfcn_bc.tmf` — Borland C
- `rtwsfcn_lcc.tmf` — LCC compiler
- `rtwsfc_unix.tmf` — UNIX host
- `rtwsfcn_vc.tmf` — Visual C
- `rtwsfcn_watc.tmf` — Watcom C

Real-Time Workshop Rapid Simulation Target

Introduction

The Real-Time Workshop rapid simulation target (`rsim`) consists of a set of target files for nonreal-time execution on your host computer. You can use `rsim` to generate fast, stand-alone simulations that allow batch parameter tuning and loading of new simulation data (signals) from a standard MATLAB MAT-file without needing to recompile your model.

C code generated from Real-Time Workshop is highly optimized to provide fast execution of discrete-time systems or systems that use a fixed-step solver. The speed of the generated code also makes it ideal for batch or Monte Carlo simulation. The run-time interface for the rapid simulation target enables the generated code to read and write data to standard MATLAB MAT-files. Using these support files, `rsim` reads new signals and parameters from MAT-files at the start of the simulation.

After building an `rsim` executable with Real-Time Workshop and an appropriate C compiler for your host computer, you can perform any combination of the following by using command line options. Without recompiling, the rapid simulation target allows you to:

- Specify a new file(s) that provides input signals for From File blocks
- Specify a new file that provides input signals with any Simulink data type (`double`, `float`, `int32`, `uint32`, `int16`, `uint16`, `int8`, `uint8`, and complex data types) by using the From Workspace block
- Replace the entire block diagram parameter vector and run a simulation
- Specify a new stop time for ending the stand-alone simulation
- Specify a new name of the MAT-file used to save model output data
- Specify name(s) of the MAT-files used to save data connected to To File blocks

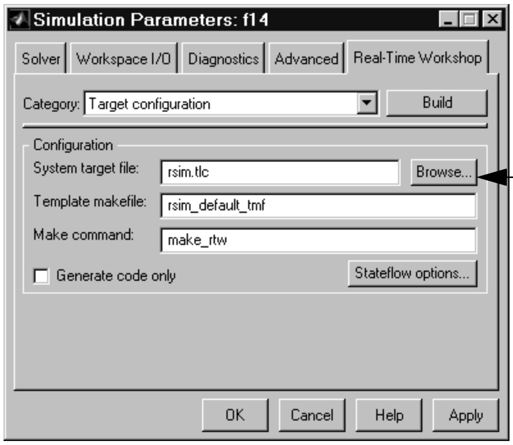
You can run these options:

- Directly from your operating system command line (for example, DOS box or UNIX shell) or
- By using the bang (!) command with a command string at the MATLAB prompt

Therefore, you can easily write simple scripts that will run a set of simulations in sequence while using new data sets. These scripts can be written to provide unique filenames for both input parameters and input signals, as well as output filenames for the entire model or for To File blocks.

Building for the Rapid Simulation Target

To generate and build an `rsim` executable, press the **Browse** button on the Real-Time Workshop page of the **Simulation Parameters** dialog box, and select the rapid simulation target from the **System Target File Browser**. This picture shows the dialog box settings for the rapid simulation target.



Category: Target configuration

Configuration

System target file: rsim.tlc

Template makefile: rsim_default_tmf

Make command: make_rtw

☐ Generate code only

Press the **Browse** button and select the rapid simulation target from the **System Target File Browser**. This automatically selects the correct settings for the system target file, the template makefile, and the `make` command.

Figure 11-1: Specifying Target and Make Files for rsim

After specifying system target and make files as noted above, select any desired Workspace I/O settings, and press **Build**. The Real-Time Workshop will automatically generate C code and build the executable for your host machine using your host machine C compiler. See “Choosing and Configuring Your Compiler” on page 3-99 and “Template Makefiles and Make Options” on page 3-102 for additional information on compilers that are compatible with Simulink and the Real-Time Workshop.

Note `rsim` executables can be transferred and run on computers that do not have MATLAB installed. When running an `rsim` executable on such a machine, it is necessary to have the following dlls in your working directory: `libmx.dll`, `libut.dll`, and `libmat.dll`. These dlls are required for any `rsim` executable that writes or reads data to or from a `.mat` file.

Running a Rapid Simulation

The rapid simulation target lets you run a simulation similar to the generic real-time (GRT) target provided by the Real-Time Workshop. This simulation does not use timer interrupts, and therefore is a nonreal-time simulation environment. The difference between GRT and `rsim` simulations is that `rsim` allows you to change parameter values or input signals at the start of a simulation without the need to generate code or recompile. The GRT target, on the other hand, is a starting point for targeting a new processor.

A single build of your model can be used to study effects from varying parameters or input signals. Command line arguments provide the necessary mechanism to specify new data for your simulation. This table lists all available command line options.

Table 11-1: rsim Command Line Options

Command Line Option	Description
`model -f old.mat=new.mat`	Read From File block input signal data from a replacement MAT-file.
`model -o newlogfile.mat`	Write MAT-file logging data to a file named `newlogfile.mat`.
`model -p filename.mat`	Read a new (replacement) parameter vector from a file named `filename.mat`.
`model -s <stoptime>`	Run the simulation until the time value `<stoptime>` is reached.
`model -t old.mat=new.mat`	The original model specified saving signals to the output file `old.mat`. For this run use the file `new.mat` for saving signal data.
`model -v`	Run in verbose mode.
`model -h`	Display a help message listing options.

Specifying a New Signal Data File for a From File Block

To understand how to specify new signal data for a From File block, create a working directory and connect to that directory. Open the model `rsimtfdemo` by typing

```
rsimtfdemo
```

at the MATLAB prompt. Type

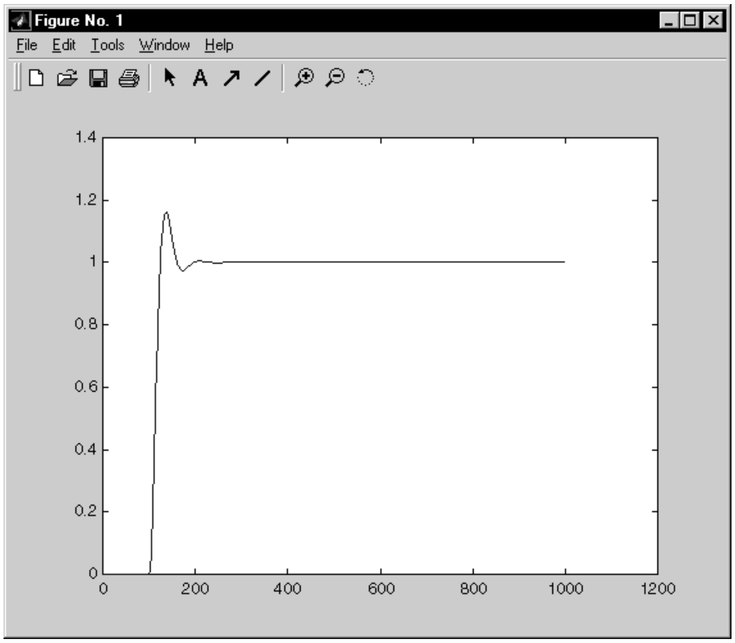
```
w = 100;
zeta = 0.5;
```

to set parameters. `rsimtfdemo` requires a data file, `rsim_tfdata.mat`. Make a local copy of *matlabroot*`/toolbox/rtw/rtwdemos/rsim_tfdata.mat` in your working directory.

Be sure to specify `rsim.tlc` as the system target file and `rsim_default_tmf` as the template makefile. Then press the **Build** button on the Real-Time Workshop page to create the `rsim` executable.

```
!rsimtfdemo
load rsimtfdemo
plot(rt_yout)
```

The resulting plot shows simulation results using the default input data.



Replacing Input Signal Data. New data for a From File block can be placed in a standard MATLAB MAT-file. As in Simulink, the From File block data must be stored in a matrix with the first row containing the time vector while subsequent rows contain u vectors as input signals. After generating and compiling your code, you can type the model name rsimtfdemo at a DOS prompt to run the simulation. In this case, the file rsim_tfdata.mat provides the input data for your simulation.

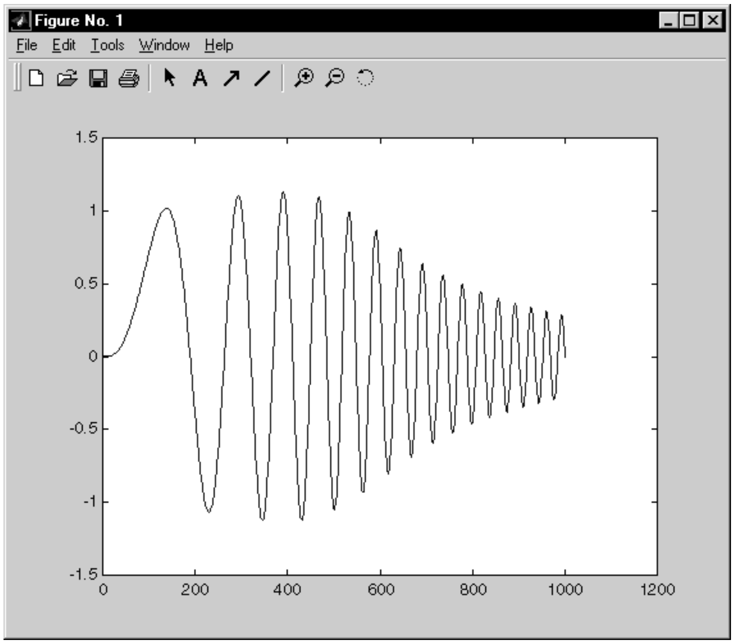
For the next simulation, create a new data file called newfrom.mat and use this to replace the original file (rsim_tfdat.mat) and run an rsim simulation with this new data. This is done by typing

```
t=[0:.001:1];
u=sin(100*t.*t);
tu=[t;u];
save newfrom.mat tu;
!rsimtfdemo -f rsim_tfdata.mat=newfrom.mat
```

at the MATLAB prompt. Now you can load the data and plot the new results by typing

```
load rsimtfdemo
plot(rt_yout)
```

This picture shows the resulting plot.



As a result the new data file is read and the simulation progresses to the stop time specified in the Solver page of the **Simulation Parameters** dialog box. It is possible to have multiple instances of From File blocks in your Simulink model.

Since `rsim` does not place signal data into generated code, it reduces code size and compile time for systems with large numbers of data points that originate in From File blocks. The From File block requires the time vector and signals to be data of type double. If you need to import signal data of a data type other than double, use a From Workspace block with the data specified as a structure.

The workspace data must be in the format

```
variable.time
variable.signals.values
```

If you have more than one signal, the format must be

```
variable.time
variable.signals(1).values
variable.signals(2).values
```

Specifying a New Output Filename for the Simulation

If you have specified **Save to Workspace** options (that is, checked **Time**, **States**, **Outputs**, or **Final States** check boxes on the Workspace I/O page of the **Simulation Parameters** dialog box), the default is to save simulation logging results to the file *model*`.mat`. You can now specify a replacement filename for subsequent simulations. In the case of the model `rsimtfdemo`, by typing

```
!rsimtfdemo
```

at the MATLAB prompt, a simulation runs and data is normally saved to `rsimtfdemo.mat`.

```
!rsimtfdemo
created rsimtfdemo.mat
```

You can specify a new output filename for data logging by typing

```
!rsimtfdemo -o sim1.mat
```

In this case, the set of parameters provided at the time of code generation, including any From File block data, is run. You can combine a variety of `rsim` flags to provide new data, parameters, and output files to your simulation. Note that the MAT-file containing data for the From File blocks is required. This differs from the `grt` operation, which inserts MAT-file data directly into the generated C code that is then compiled and linked as an executable. In contrast, `rsim` allows you to provide new or replacement data sets for each successive simulation. A MAT-file containing From File or From Workspace data must be present, if any From File or From Workspace blocks exist in your model.

Changing Block Parameters for an rsim Simulation

Once you have altered one or more parameter in the Simulink block diagram, you can extract the parameter vector, `rtP`, for the entire model. The `rtP` vector, along with a model checksum, can then be saved to a MATLAB MAT-file. This MAT-file can be read in directly by the stand-alone `rsim` executable, allowing you to replace the entire parameter vector quickly, for running studies of variations of parameter values where you are adjusting model parameters or coefficients or importing new data for use as input signals.

The model checksum provides a safety check to ensure that any parameter changes are only applied to `rsim` models that have the same model structure. If any block is deleted, or a new block added, then when generating a new `rtP` vector, the new checksum will no longer match the original checksum. `rsim` will detect this incompatibility in parameter vectors and exit to avoid returning incorrect simulation results. In this case, where model structure has changed, you must regenerate the code for the model.

The `rsim` target allows you to alter any model parameter, including parameters that include *side-effects* functions. An example of a side-effects function is a simple Gain block that includes the following parameter entry in a dialog box.

```
gain value:   2 * a
```

In general, the Real-Time Workshop evaluates side-effects functions prior to generating code. The generated code for this example retains only one memory location entry, and the dependence on parameter `a` is no longer visible in the generated code. The `rsim` target overcomes the problem of handling side-effects functions by replacing the entire parameter structure, `rtP`. You must create this new structure by using `rsimgetrtp.m`. and then save it in a MAT-file. For the `rsimtfdemo` example, type

```
zeta = .2;
myrtp = rsimgetrtp('modelname');
save myparamfile myrtp;
```

at the MATLAB prompt.

In turn, `rsim` can read the MAT-file and replace the entire `rtP` structure whenever you need to change one or more parameters — without recompiling the entire model.

For example, assume that you have changed one or more parameters in your model, generated the new `rtP` vector, and saved `rtP` to a new MAT-file called myparamfile.mat. In order to run the same rsimtfdemo model and use these new parameter values, execute the model by typing

```
!rsimtfdemo -p myparamfile.mat
load rsimtfdemo
plot(rt_yout)
```

Note that the p is lower-case and represents "Parameter file."

Specifying a New Stop Time for an rsim Simulation

If a new stop time is not provided, the simulation will run until reaching the value specified in the Solver page at the time of code generation. You can specify a new stop time value as follows.

```
!rsimtfdemo -s 6.0
```

In this case, the simulation will run until it reaches 6.0 seconds. At this point it will stop and log the data according to the MAT-file data logging rules as described above.

If your model includes From File blocks that also include a time vector in the first row of the time and signal matrix, the end of the simulation is still regulated by the original setting in the Solver page of the **Simulation Parameters** dialog box or from the -s option as described above. However, if the simulation time exceeds the end points of the time and signal matrix (that is, if the final time is greater than the final time value of the data matrix), then the signal data will be extrapolated out to the final time value as specified above.

Specifying New Output Filenames for To File Blocks

In much the same way as you can specify a new system output filename, you can also provide new output filenames for data saved from one or more To File blocks. This is done by specifying the original filename at the time of code generation with a new name as follows.

```
!mymodel -t original.mat=replacement.mat
```

In this case, assume that the original model wrote data to the output file called original.mat. Specifying a new filename forces rsim to write to the file replacement.mat. This technique allows you to avoid over-writing an existing simulation run.

Simulation Performance

It is not possible to predict accurately the simulation speedup of an `rsim` simulation compared to a standard Simulink simulation. Performance will vary. Larger simulations have achieved speed improvements of up to 10 times faster than standard Simulink simulations. Some models may not show any noticeable improvement in simulation speed. The only way to determine speedup is to time your standard Simulink simulation and then compare its speed with the associated `rsim` simulation.

Batch and Monte Carlo Simulations

The `rsim` target is intended to be used for batch simulations in which parameters and/or input signals are varied for each new simulation. New output filenames allow you run new simulations without over-writing prior simulation results. A simple example of such a set of batch simulations can be run by creating a `.bat` file for use under Microsoft Windows 95 or Windows NT.

This simple file (for Windows 95 or Windows NT) is created with any text editor and executed by typing the filename, for example, `mybatch`, where the name of the text file is `mybatch.bat`.

```
rsimtfdemo -f rsimtfdemo.mat=run1.mat -o results1.mat -s 10.0
rsimtfdemo -f rsimtfdemo.mat=run2.mat -o results2.mat -s 10.0
rsimtfdemo -f rsimtfdemo.mat=run3.mat -o results3.mat -s 10.0
rsimtfdemo -f rsimtfdemo.mat=run4.mat -o results4.mat -s 10.0
```

In this case, batch simulations are run using the four sets of input data in files `run1.mat`, `run2.mat`, and so on. `rsim` saves the data to the corresponding files specified after the `-o` option.

The variable names containing simulation results in each of these files are identical. Therefore, loading consecutive sets of data without renaming the data once it is in the MATLAB workspace will result in over-writing the prior workspace variable with new data. If you want to avoid over-writing, you can copy the result to a new MATLAB variable prior to loading the next set of data.

You can also write M-file scripts to create new signals, and new parameter structures, as well as to save data and perform batch runs using the bang command (`!`).

For additional insight into the rapid simulation target, explore `rsimdemo1` and `rsimdemo2`, located in *matlabroot*`/toolbox/rtw/rtwdemos`. These examples demonstrate how `rsim` can be called repeatedly within an M-file for Monte Carlo simulations.

Targeting Tornado for Real-Time Applications

This chapter contains the following topics:

- **Introduction.** Overview of the Tornado (VxWorks) Real-Time Target and the VxWorks Support library.
- **Run-Time Architecture Overview.** Single- and multitasking architecture and host/target communications.
- **Implementation Overview.** Designing, implementing and running a VxWorks-based real-time program using the Real-Time Workshop.

Introduction

This chapter describes how to create real-time programs for execution under VxWorks, which is part of the Tornado environment.

The VxWorks real-time operating system is available from Wind River Systems, Inc. It provides many UNIX-like features and comes bundled with a complete set of development tools.

Note Tornado is an integrated environment consisting of VxWorks (a high-performance real-time operating system), application building tools (compiler, linker, make, and archiver utilities), and interactive development tools (editor, debugger, configuration tool, command shell, and browser).

This chapter discusses the run-time architecture of VxWorks-based real-time programs generated by the Real-Time Workshop and provides specific information on program implementation. Topics covered include:

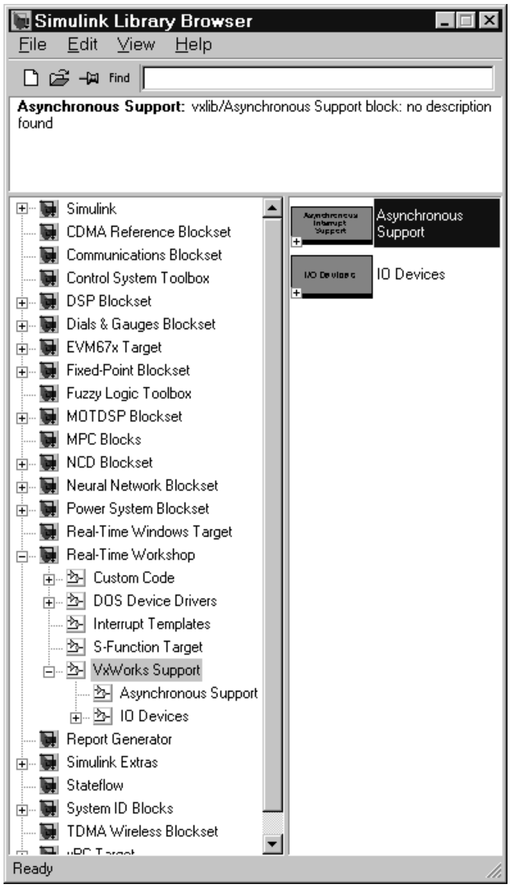
- Configuring device driver blocks and makefile templates
- Building the program
- Downloading the object file to the VxWorks target
- Executing the program on the VxWorks target
- Using Simulink external mode to change model parameters and download them to the executing program on the VxWorks target
- Using the StethoScope data acquisition and graphical monitoring tool, which is available as an option with VxWorks. It allows you to access the output of any block in the model (in the real-time program) and display the data on the host.

Confirming Your Tornado Setup Is Operational

Before beginning, you must install and configure Tornado on your host and target hardware, as discussed in the Tornado documentation. You should then run one of the VxWorks demonstration programs to ensure you can boot your VxWorks target and download object files to it. See the *Tornado User's Guide* for additional information about installation and operation of VxWorks and Tornado products.

VxWorks Library

Selecting **VxWorks Support** under the **Real-Time Workshop** library in the Simulink Library Browser opens the **VxWorks Support** library.



Asynchronous Support: vxlib/Asynchronous Support block: no description found

The blocks discussed in this chapter are located in the Asynchronous Support library, a sublibrary of the VxWorks Support library:

- Interrupt Control
- Rate Transition
- Read Side

- Task Synchronization
- Write Side

A second sublibrary, the IO Devices library, contains support for these drivers:

- Matrix MS-AD12
- Matrix MS-DA12
- VME Microsystems VMIVM-3115-1
- Xycom XVME-500/590
- Xycom XVME-505/595

Each of these blocks has online help available through the **Help** button on the block's dialog box. Refer to the *Tornado User's Guide* for detailed information on these blocks.

Run-Time Architecture Overview

In a typical VxWorks-based real-time system, the hardware consists of a UNIX or PC host running Simulink and the Real-Time Workshop, connected to a VxWorks target CPU via Ethernet. In addition, the target chassis may contain I/O boards with A/D and D/A converters to communicate with external hardware. The following diagram shows the arrangement.

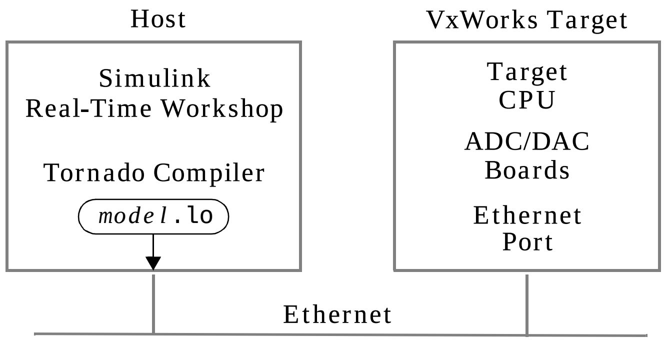


Figure 12-1: Typical Hardware Setup for a VxWorks Application

The real-time code is compiled on the UNIX or PC host using the cross compiler supplied with the VxWorks package. The object file (*model*.lo) output from the Real-Time Workshop program builder is downloaded, using `WindSh` (the command shell) in Tornado, to the VxWorks target CPU via an Ethernet connection.

The real-time program executes on the VxWorks target and interfaces with external hardware via the I/O devices installed on the target.

Parameter Tuning and Monitoring

You can change program parameters from the host and monitor data with Scope blocks while the program executes using Simulink external mode. You can also monitor program outputs using the StethoScope data analysis tool.

Using Simulink external mode and/or StethoScope in combination allows you to change model parameters in your program, and to analyze the results of these changes, in real time.

External Mode

Simulink external mode provides a mechanism to download new parameter values to the executing program and to monitor signals in your model. In this mode, the external link MEX-file sends a vector of new parameter values to the real-time program via the network connection. These new parameter values are sent to the program whenever you make a parameter change without requiring a new code generation or build iteration.

You can use the `BlockIOSignals` code generation option to monitor signals in VxWorks. See “Interfacing Parameters and Signals” on page 17-65 for further information and example code.

The real-time program (executing on the VxWorks target) runs a low priority task that communicates with the external link MEX-file and accepts the new parameters as they are passed into the program.

Communication between Simulink and the real-time program is accomplished using the sockets network API. This implementation requires an Ethernet network that supports TCP/IP. See Chapter 5, “External Mode” for more information on external mode.

Changes to the block diagram structure (for example, adding or removing blocks) require generation of model and execution of the build process.

Configuring VxWorks to Use Sockets

If you want to use Simulink external mode with your VxWorks program, you must configure your VxWorks kernel to support sockets by including the `INCLUDE_NET_INIT`, `INCLUDE_NET_SHOW`, and `INCLUDE_NETWORK` options in your VxWorks image. For more information on configuring your kernel, see the *VxWorks Programmer’s Guide*.

Before using external mode, you must ensure that VxWorks can properly respond to your host over the network. You can test this by using the host command

```
ping <target_name>
```

Note You may need to enter a routing table entry into VxWorks if your host is not on the same local network (subnet) as the VxWorks system. See `routeAdd` in the *VxWorks Reference Guide* for more information.

Configuring Simulink to Use Sockets

Simulink external mode uses a MEX-file to communicate with the VxWorks system. The MEX-file is

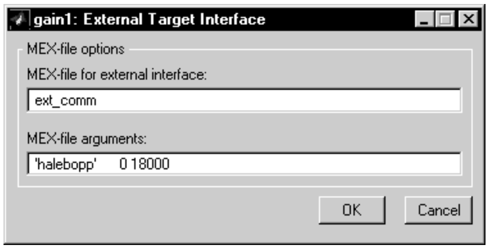
```
matlabroot/toolbox/rtw/ext_comm.*
```

where * is a host-dependent MEX-file extension. See Chapter 5, "External Mode" for more information.

To use external mode with VxWorks, specify `ext_comm` as the **MEX-file for external interface** in the **External Target Interface** dialog box (accessed from the **External Mode Control Panel**). In the **MEX-file arguments** field you must specify the name of the VxWorks target system and, optionally, the verbosity and TCP port number. Verbosity can be 0 (the default) or 1 if extra information is desired. The TCP port number ranges from 256 to 65535 (the default is 17725). If there is a conflict with other software using TCP port 17725, you can change the port that you use by editing the third argument of the **MEX-file for external interface** on the **External Target Interface** dialog box. The format for the MEX-file arguments field is

```
'target_network_name' [verbosity] [TCP port number]
```

For example, this picture shows the **External Target Interface** dialog box configured for a target system called `halebopp` with default verbosity and the port assigned to 18000.



MEX-file for external interface: ext_comm
MEX-file arguments: 'halebopp' 0 18000

StethoScope

With StethoScope, you can access the output of any block in the model (in the real-time program) and display this data on a host. Signals are installed in StethoScope by the real-time program using the `BlockIOSignals` data structure (See “Interfacing Parameters and Signals” on page 17-65 for information on `BlockIOSignals`), or interactively from the `WindSh` while the real-time program is running. To use StethoScope interactively, see the *StethoScope User’s Manual*.

To use StethoScope you must specify the proper options with the build command. See “Code Generation Options” on page 12-18 for information on these options.

Run-Time Structure

The real-time program executes on the VxWorks target while Simulink and StethoScope execute on the same or different host workstations. Both Simulink and StethoScope require tasks on the VxWorks target to handle communication.

This diagram illustrates the structure of a VxWorks application using Simulink external mode and StethoScope.

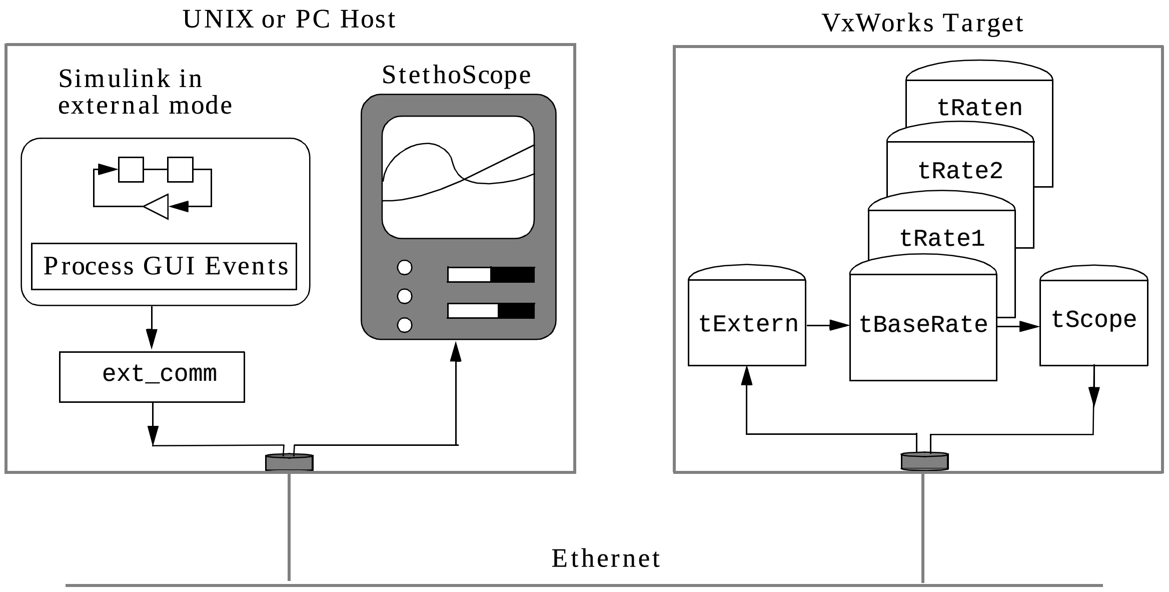


Figure 12-2: The Run-Time Structure

The program creates VxWorks tasks to run on the real-time system: one communicates with Simulink, the others execute the model. StethoScope creates its own tasks to collect data.

Host Processes

There are two processes running on the host side that communicate with the real-time program:

- Simulink running in external mode. Whenever you change a parameter in the block diagram, Simulink calls the external link MEX-file to download any new parameter values to the VxWorks target.
- The StethoScope user interface module. This program communicates with the StethoScope real-time module running on the VxWorks target to retrieve model data and plot time histories.

VxWorks Tasks

You can run the real-time program in either singletasking or multitasking mode. The code for both modes is located in

```
matlabroot/rtw/c/tornado/rt_main.c
```

The Real-Time Workshop compiles and links `rt_main.c` with the model code during the build process.

Singletasking. By default, the model is run as one task, `tSingleRate`. This may actually provide the best performance (highest base sample rate) depending on the model.

The `tSingleRate` task runs at the base rate of the model and executes all necessary code for the slower sample rates. Execution of the `tSingleRate` task is normally blocked by a call to the VxWorks `semTake` routine. When a clock interrupt occurs, the interrupt service routine calls the `semGive` routine, which causes the `semTake` call to return. Once enabled, the `tSingleRate` task executes the model code for one time step. The loop then waits at the top by again calling `semTake`. For more information about the `semTake` and `semGive` routines, refer to the *VxWorks Reference Manual*. By default, it runs at a relatively high priority (30), which allows it to execute without interruption from background system activity.

Multitasking. Optionally, the model can run as multiple tasks, one for each sample rate in the model:

- `tBaseRate` — This task executes the components of the model code run at the base (highest) sample rate. By default, it runs at a relatively high priority (30), which allows it to execute without interruption from background system activity.
- `tRate`*n* — The program also spawns a separate task for each additional sample rate in the system. These additional tasks are named `tRate1`, `tRate2`, …, `tRaten`, where `n` is slowest sample rate in the system. The priority of each additional task is one lower than its predecessor (`tRate1` has a lower priority than `tBaseRate`).

Supporting Tasks. If you select external mode and/or StethoScope during the build process, these tasks will also be created:

- `tExtern` — This task implements the server side of a socket stream connection that accepts data transferred from Simulink to the real-time program. In this implementation, `tExtern` waits for a message to arrive from Simulink. When a message arrives, `tExtern` retrieves it and modifies the specified parameters accordingly.

  `tExtern` runs at a lower priority than `tRaten`, the lowest priority model task. The source code for `tExtern` is located in *matlabroot*`/rtw/c/src/ext_svr.c`.
- `tScopeDaemon` and `tScopeLink` — StethoScope provides its own VxWorks tasks to enable real-time data collection and display. In singletasking mode, tSingleRate collects signals; in multitasking mode, tBaseRate collects them. Both perform the collection on every base time step. The StethoScope tasks then send the data to the host for display when there is idle time, that is, when the model is waiting for the next time step to occur. `rt_main.c` starts these tasks if they are not already running.

Implementation Overview

To implement and run a VxWorks-based real-time program using the Real-Time Workshop, you must:

- Design a Simulink model for your particular application.
- Add the appropriate device driver blocks to the Simulink model, if desired.
- Configure the `tornado.tmf` template makefile for your particular setup.
- Establish a connection between the host running Simulink and the VxWorks target via Ethernet.
- Use the automatic program builder to generate the code and the custom makefile, invoke the `make` command to compile and link the generated code, and load and activate the tasks required.

The figure below shows the Real-Time Workshop Tornado run-time interface modules and the generated code for the `f14` example model.

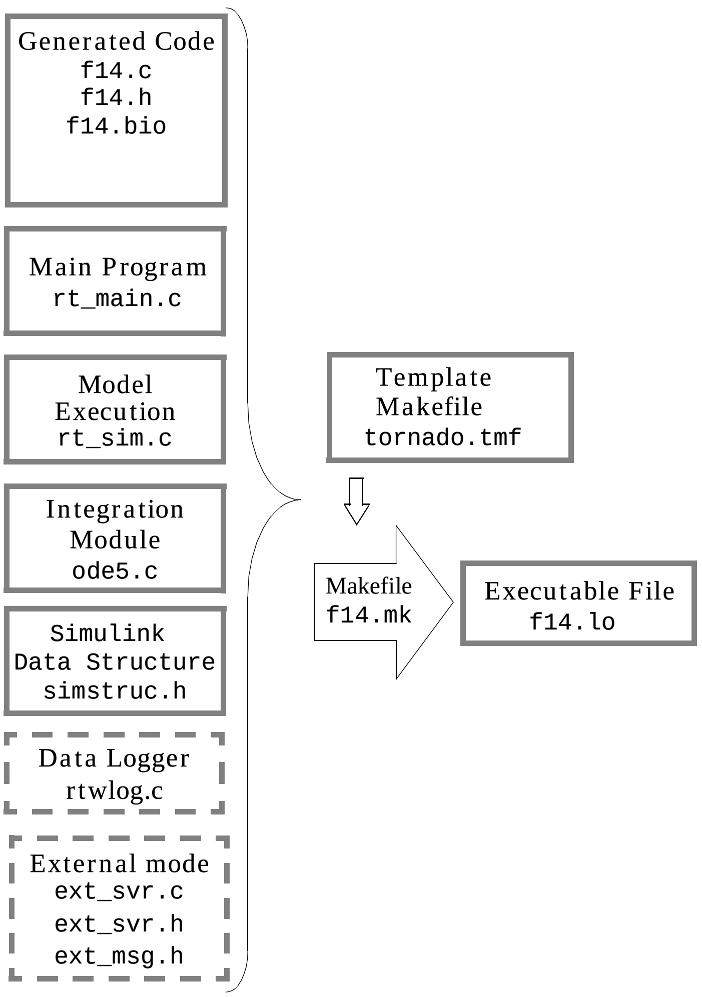


Figure 12-3: Source Modules Used to Build the VxWorks Real-Time Program

This diagram illustrates the code modules used to build a VxWorks real-time program. Dashed boxes indicate optional modules.

Adding Device Driver Blocks

The real-time program communicates with the I/O devices installed in the VxWorks target chassis via a set of device drivers. These device drivers contain the necessary code that runs on the target processor for interfacing to specific I/O devices.

To make device drivers easy to use, they are implemented as Simulink S-functions using C code MEX-files. This means you can connect them to your model like any other block and the code generator automatically includes a call to the block's C code in the generated code.

You can also inline S-functions via the Target Language Compiler. Inlining allows you to restrict function calls to only those that are necessary for the S-function. This can greatly increase the efficiency of the S-function. For more information about inlining S-functions, see *Writing S-Functions* and the *Target Language Compiler Reference Guide*.

You can have multiple instances of device driver blocks in your model. See *Targeting Real-Time Systems* for more information about creating device drivers.

Configuring the Template Makefile

To configure the VxWorks template, `tornado.tmf`, you must specify information about the environment in which you are using VxWorks. This section lists the lines in the file that you must edit.

VxWorks Configuration

To provide information used by VxWorks, you must specify the type of target and the specific CPU on the target. The target type is then used to locate the correct cross compiler and linker for your system.

The CPU type is used to define the `CPU` macro which is in turn used by many of the VxWorks header files. Refer to the *VxWorks Programmer's Guide* for information on the appropriate values to use.

This information is in the section labeled

```
#------------- VxWorks Configuration -------------
```

Edit the following lines to reflect your setup.

```
VX_TARGET_TYPE = 68k
CPU_TYPE = MC68040
```

Downloading Configuration

In order to perform automatic downloading during the build process, the target name and host name that the Tornado target server will run on must be specified. Modify these macros to reflect your setup.

```
#------------- Macros for Downloading to Target-------------
TARGET = targetname
TGTSVR_HOST = hostname
```

Tool Locations

In order to locate the Tornado tools used in the build process, the following three macros must either be defined in the environment or specified in the template makefile. Modify these macros to reflect your setup.

```
#------------- Tool Locations -------------
WIND_BASE = c:/Tornado
WIND_HOST_TYPE = x86–win32
WIND_REGISTRY = $(COMPUTERNAME)
```

Building the Program

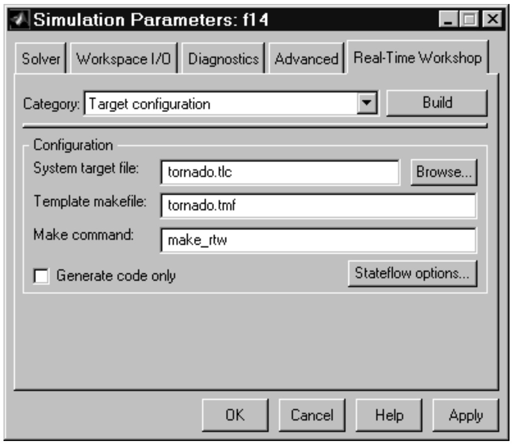
Once you have created the Simulink block diagram, added the device drivers, and configured the makefile template, you are ready to set the build options and initiate the build process.

Specifying the Real-Time Build Options

Set the real-time build options using the Solver and Real-Time Workshop pages of the **Simulation Parameters** dialog box. To access this dialog box, select **Parameters** from the Simulink **Simulation** menu.

In the Solver page, for models with continuous blocks, set the **Type** to **Fixed-step**, the **Step Size** to the desired integration step size, and select the integration algorithm. For models that are purely discrete, set the integration algorithm to **discrete**.

Next, use the System Target File Browser to select the correct Real-Time Workshop page settings for Tornado. To access the browser, open the Real-Time Workshop page of the **Simulation Parameters** dialog box and select **Target configuration** from the **Category** menu.



Category: Target configuration

Configuration

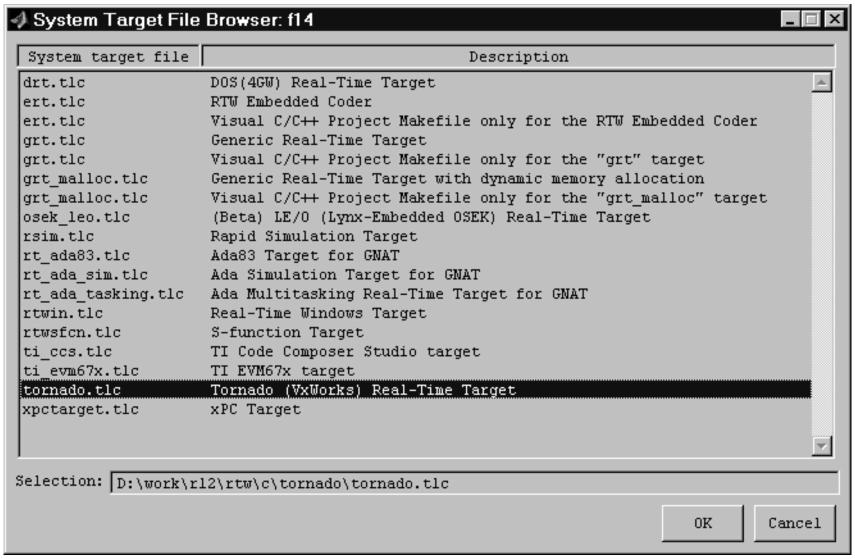
System target file: tornado.tlc

Template makefile: tornado.tmf

Make command: make_rtw

☐ Generate code only

Then click the **Browse** button. The System Target Browser opens.



System target file	Description
drt.tlc	DOS(4GW) Real-Time Target
ert.tlc	RTW Embedded Coder
ert.tlc	Visual C/C++ Project Makefile only for the RTW Embedded Coder
grt.tlc	Generic Real-Time Target
grt.tlc	Visual C/C++ Project Makefile only for the "grt" target
grt_malloc.tlc	Generic Real-Time Target with dynamic memory allocation
grt_malloc.tlc	Visual C/C++ Project Makefile only for the "grt_malloc" target
osek_leo.tlc	(Beta) LE/O (Lynx-Embedded OSEK) Real-Time Target
rsim.tlc	Rapid Simulation Target
rt_ada83.tlc	Ada83 Target for GNAT
rt_ada_sim.tlc	Ada Simulation Target for GNAT
rt_ada_tasking.tlc	Ada Multitasking Real-Time Target for GNAT
rtwin.tlc	Real-Time Windows Target
rtwsfcn.tlc	S-function Target
ti_ccs.tlc	TI Code Composer Studio target
ti_evm67x.tlc	TI EVM67x target
tornado.tlc	Tornado (VxWorks) Real-Time Target
xpctarget.tlc	xPC Target

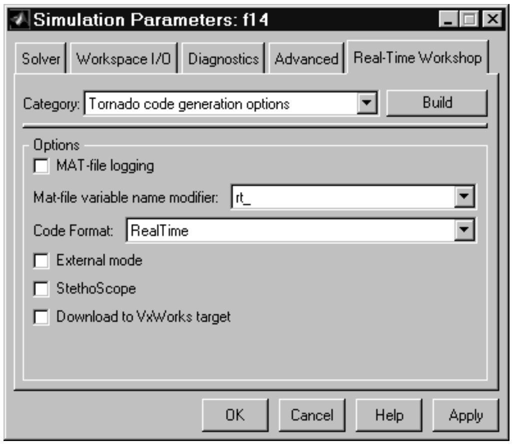
Selection: D:\work\r12\rtw\c\tornado\tornado.tlc

Select Tornado (VxWorks) Real-Time Target and click **OK**. This sets the **Target configuration** options correctly:

- **System target file** — `tornado.tlc`
- **Template makefile** — `tornado.tmf` template, which you must configure according to the instructions in "Configuring the Template Makefile" on page 12-15. (You can rename this file; simply change the dialog box accordingly.)
- **Make command** — `make_rtw`

You can optionally inline parameters for the blocks in the C code, which can improve performance. Inlining parameters is not allowed when using external mode.

Code Generation Options. To specify code generation options specific to Tornado, open the Real-Time Workshop page and select **Tornado code generation options** from the **Category** menu.



Category: Tornado code generation options
Options
☐ MAT-file logging
Mat-file variable name modifier: rt_
Code Format: RealTime
☐ External mode
☐ StethoScope
☐ Download to VxWorks target

The Real-Time Workshop provides flags that set the appropriate macros in the template makefile, causing any necessary additional steps to be performed during the build process.

The flags and switches are as follows:

- **MAT-file logging**: Select this option to enable data logging during program execution. The program will create a file named *model*`.mat` at the end of program execution; this file will contain the variables that you specified in the Solver page of the **Simulation Parameters** dialog box.

Real-Time Workshop adds a prefix or suffix to the names of the Solver page variables that you select for logging. The **MAT-file variable name modifier** menu lets you select the prefix or suffix.

By default, the MAT-file is created in the root directory of the current default device in VxWorks. This is typically the host file system that VxWorks was booted from. Other remote file systems can be used as a destination for the MAT-file using `rsh` or `ftp` network devices or NFS. See the *VxWorks Programmer's Guide* for more information. If a device or filename other than the default is desired, add `"-DSAVEFILE=filename"` to the `OPTS` flag to the `make` command. For example,

```
make_rtw OPTS="-DSAVEFILE=filename"
```

- **External mode**: Select this option to enable the use of external mode in the generated executable. You can optionally enable a verbose mode of external mode by appending `-DVERBOSE` to the `OPTS` flag in the `make` command. For example,

  ```
  make_rtw OPTS="-DVERBOSE"
  ```

  causes parameter download information to be printed to the console of the VxWorks system.
- **Code format**: Selects `RealTime` or `RealTimeMalloc` code generation format.
- **StethoScope**: Select this option to enable the use of StethoScope with the generated executable. When starting `rt_main`, there are two command line arguments that control the block names used by StethoScope; you can use them when starting the program on VxWorks. See the section, "Running the Program" on page 12-21 for more information on these arguments.
- **Download to VxWorks target**: Enables automatic downloading of the generated program.

Additional options are available on the Real-Time Workshop page. See "Using the Real-Time Workshop Page" on page 3-4 for information.

Initiating the Build

To build the program, click on the **Build** button in the Real-Time Workshop page of the **Simulation parameters** dialog. The resulting object file is named with the `.lo` extension (which stands for *loadable object*). This file has been compiled for the target processor using the cross compiler specified in the makefile. If automatic downloading (**Download to VxWorks target**) is enabled in the **Tornado code generation** options, the target server is started and the object file is downloaded and started on the target. If **StethoScope** was checked on the **Tornado code generation** options, you can now start StethoScope on the host. The StethoScope object files, `libxdr.so`, `libutilstssip.so`, and `libscope.so`, will be loaded on the VxWorks target by the automatic download. See the *StethoScope User's Manual* for more information.

Downloading and Running the Executable Interactively

If automatic downloading is disabled, you must use the Tornado tools to complete the process. This involves three steps:

1. Establishing a communication link to transfer files between the host and the VxWorks target
2. Transferring the object file from the host to the VxWorks target
3. Running the program

Connecting to the VxWorks Target

After completing the build process, you are ready to connect the host workstation to the VxWorks target. The first step is starting the target server that is used for communication between the Tornado tools on the host and the target agent on the target. This is done either from the DOS command line or from within the Tornado development environment. From the DOS command line use

```
tgtsvr target_network_name
```

Downloading the Real-Time Program

To download the real-time program, use the VxWorks `ld` routine from within `WindSh`. `WindSh` (wind shell) can also be run from the command line or from within the Tornado development environment. (For example, if you want to download the file `vx_equal.lo`, which is in the `/home/my_working_dir` directory, use the following commands at the `WindSh` prompt.

```
cd "/home/my_working_dir"
ld <vx_equal.lo
```

You will also need to load the StethoScope libraries if the StethoScope option was selected during the build. The *Tornado User's Guide* describes the `ld` library routine.

Running the Program

The real-time program defines a function, `rt_main()`, that spawns the tasks to execute the model code and communicate with Simulink (if you selected external mode during the build procedure.) It also initializes StethoScope if you selected this option during the build procedure.

The `rt_main` function is defined in the `rt_main.c` application module. This module is located in the *matlabroot*`/rtw/c/tornado` directory.

The `rt_main` function takes six arguments, and is defined by the following ANSI C function prototype.

```
SimStruct *rt_main(void (*model)(SimStruct *),
                   char *optStr,
                   char *scopeInstallString,
                   int scopeFullNames,
                   int priority,
                   int TCPport);
```

Table 12-1 lists the arguments to this function.

Table 12-1: Arguments to the rt_main simStruct

Argument	Description
`model`	A pointer to the entry point function in the generated code. This function has the same name as the Simulink model. It registers the local functions that implement the model code by adding function pointers to the model's `SimStruct`. See Chapter 6, "Program Architecture" for more information.
`optStr`	The options string used to specify a stop time (`-tf`) and whether to wait (`-w`) in external mode for a message from Simulink before starting the simulation. An example options string is `"-tf 20 -w"` The `-tf` option overrides the stop time that was set during code generation. If the value of the `-tf` option is `inf`, the program runs indefinitely.
`scopeInstallString`	A character string that determines which signals are installed to StethoScope. Possible values are: • `NULL` — Install no signals. This is the default value. • `"*"` — Install all signals. • `"[A-Z]*"` — Install signals from blocks whose names start with an uppercase letter. Specifying any other string installs signals from blocks whose names start with that string.
`scopeFullNames`	This argument determines whether StethoScope uses full hierarchical block names for the signals it accesses or just the individual block name. Possible values are: • `1` Use full block names. • `0` Use individual block names. This is the default value. It is important to use full block names if your program has multiple instances of a model or S-function.

Table 12-1: Arguments to the rt_main simStruct (Continued)

Argument	Description
`priority`	The priority of the program's highest priority task (`tBaseRate`). Not specifying any value (or specifying a value of zero) assigns `tBaseRate` to the default priority, 30.
`TCPport`	The port number that the external mode sockets connection should use. The valid range is 256 to 65535. When nothing is specified, the port number defaults to 17725.

Passing optStr Via the Template Makefile. You can also pass the `-w` and `-tf` options (see `optStr` in Table 12-1) to `rt_main` by using the `PROGRAM_OPTS` macro in `tornado .tmf`. `PROGRAM_OPTS` passes a string of the form

```
-opt1 val1 -opt2 val2
```

For example, the following sets an infinite stop time and instructs the program to wait for a message from Simulink before starting the simulation.

```
PROGRAM_OPTS = "-tf inf -w”
```

Calling rt_main. To begin program execution, call `rt_main` from `WindSh`. For example,

```
sp(rt_main, vx_equal, "-tf 20 -w", "*", 0, 30, 17725)
```

- Begins execution of the `vx_equal` model
- Specifies a stop time of 20 seconds
- Provides access to all signals (block outputs) in the model by StethoScope
- Uses only individual block names for signal access (instead of the hierarchical name)
- Uses the default priority (30) for the `tBaseRate` task
- Uses TCP port 17725, the default

Targeting DOS for Real-Time Applications

Introduction

This section provides information on using the Real-Time Workshop in a DOS environment.

Note The xPC Target and the Real-Time Windows Target provide significantly greater capabilities than does the DOS target. We recommend use of these targets for real-time development on PC platforms. The DOS target is provided only as an unsupported example. Also, note that the DOS target requires the Watcom C compiler. See "A Note on the Watcom Compiler" on page 13-6.

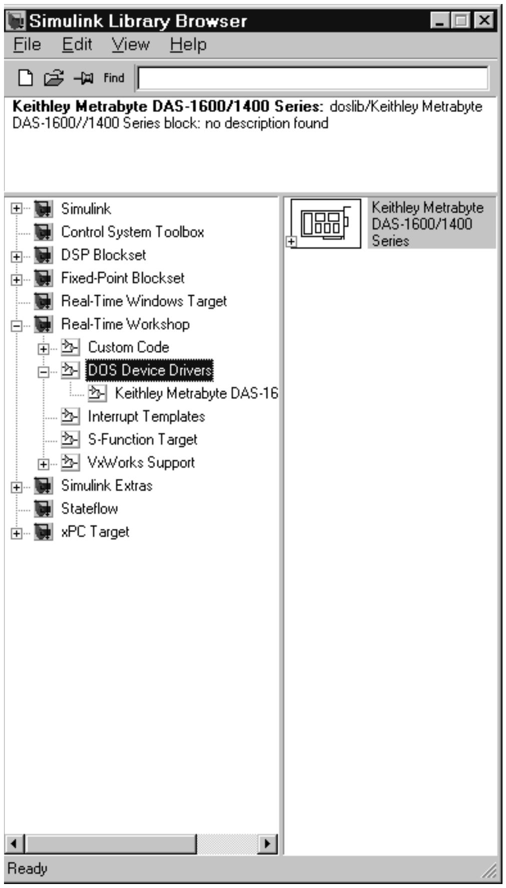
This chapter includes a discussion of:

- DOS-based Real-Time Workshop applications
- Supported compilers and development tools
- Device driver blocks — adding them to your model and configuring them for use with your hardware
- Building the program

The DOS target creates an executable for DOS, using Watcom for DOS. The executable runs on a computer running the DOS operating system. It will not run under the Microsoft Windows DOS command prompt. This executable installs interrupt service routines and effectively takes over the computer, which allows the generated code to run in real time. If you want to run the generated code in real time under Microsoft Windows, you should use the Real-Time Windows Target. See the *Real-Time Windows Target User's Guide* for more information about this product.

DOS Device Drivers Library

The Real-Time Workshop provides DOS-compatible analog and digital I/O device driver blocks in the DOS Device Drivers library. Select **DOS Device Drivers** under the Real-Time Workshop library in the Simulink Library Browser to open the DOS Device Drivers library.



Keithley Metrabyte DAS-1600/1400 Series: doslib/Keithley Metrabyte DAS-1600//1400 Series block: no description found

Implementation Overview

The Real-Time Workshop includes DOS run-time interface modules designed to implement programs that execute in real time under DOS. These modules, when linked with the code generated from a Simulink model, build a complete program that is capable of executing the model in real time. The DOS run-time interface files can be found in the *matlabroot*/rtw/c/dos/rti directory.

Real-Time Workshop DOS run-time interface modules and the generated code for the f14 demonstration model are shown in Figure 13-1.

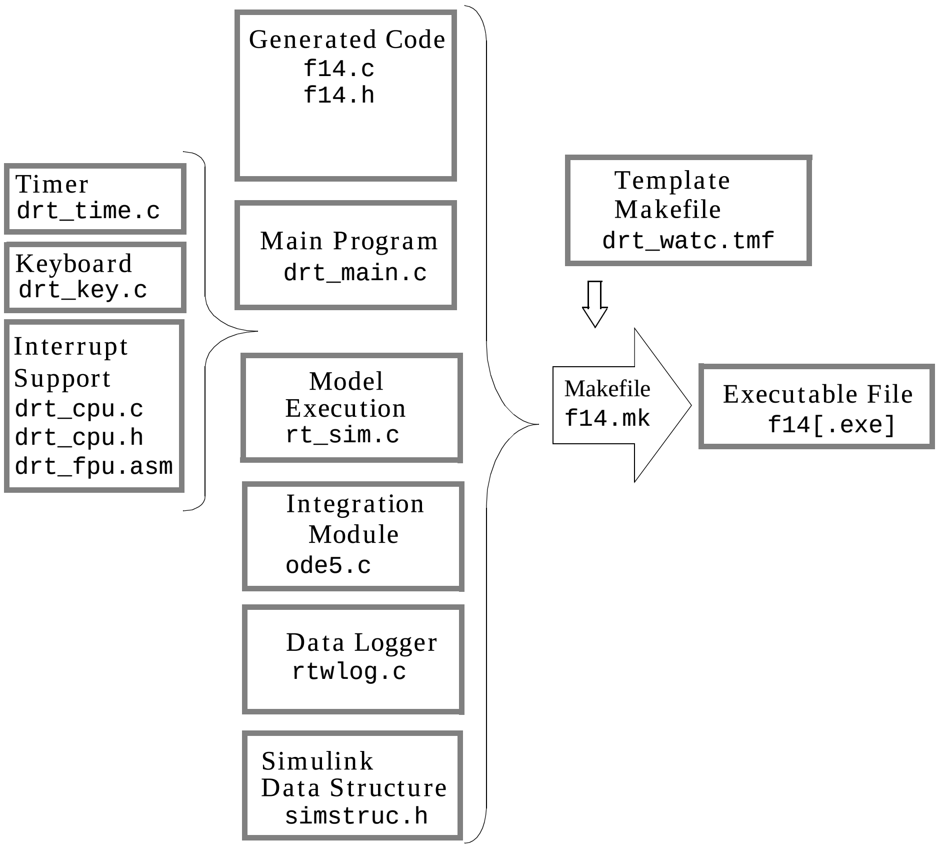


Figure 13-1: Source Modules Used to Build the DOS Real-Time Program

This diagram illustrates the code modules that are used to build a DOS real-time program.

To execute the code in real time, the program runs under the control of an interrupt driven timing mechanism. The program installs its own interrupt service routine (ISR) to execute the model code periodically at predefined sample intervals. The PC-AT's 8254 Programmable Interval Timer is used to time these intervals.

In addition to the modules shown in Figure 13-1, the DOS run-time interface also consists of device driver modules to read from and write to I/O devices installed on the DOS target.

Figure 13-2 shows the recommended hardware setup for designing control systems using Simulink, and then building them into DOS real-time applications using the Real-Time Workshop. The figure shows a robotic arm being controlled by a program (i.e., the controller) executing on the target PC. The controller senses the arm position and applies inputs to the motors accordingly, via the I/O devices on the target PC. The controller code executes on the PC and communicates with the apparatus it controls via I/O hardware.

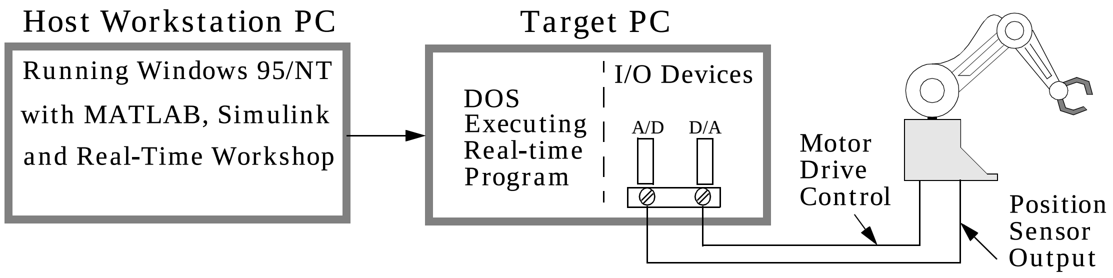

Host Workstation PC
Running Windows 95/NT with MATLAB, Simulink and Real-Time Workshop
Target PC
DOS Executing Real-time Program
I/O Devices
A/D
D/A
Motor Drive Control
Position Sensor Output

Figure 13-2: Typical Hardware Setup

System Configuration

You can use the Real-Time Workshop with a variety of system configurations, as long as these systems meet the following hardware and software requirements.

Hardware Requirements

The hardware needed to develop and run a real-time program includes:

- A workstation running Windows 95, Windows 98, or Windows NT and capable of running MATLAB/Simulink. This workstation is the *host* where the real-time program is built.
- A PC-AT (386 or later) running DOS. This system is the *target*, where the real-time program executes.
- I/O boards, which include analog to digital converter and digital to analog converters (collectively referred to as I/O devices), on the target.
- Electrical connections from the I/O devices to the apparatus you want to control (or to use as inputs and outputs to the program in the case of hardware-in-the-loop simulations).

Once built, you can run the executable on the target hardware as a stand-alone program that is independent of Simulink.

Software Requirements

The development host must have the following software:

- MATLAB and Simulink to develop the model, and Real-Time Workshop to create the code for the model. You also need the run-time interface modules included with the Real-Time Workshop. These modules contain the code that handles timing, interrupts, data logging, and background tasks.
- Watcom C/C++ compiler, Version 10.6 or 11.0. (see “A Note on the Watcom Compiler” below.)

The target PC must have the following software:

- DOS4GW extender `dos4gw.exe`, included with your Watcom compiler package) must be on the search path on the DOS-targeted PC.

You can compile the generated code (i.e., the files *model*`.c`, *model*`.h`, etc.) along with user-written code using other compilers. However, the use of 16-bit compilers is not recommended for any application.

A Note on the Watcom Compiler

As of this writing, the Watcom C compiler is no longer available from the manufacturer. The Real-Time Workshop continues to ship Watcom-related target configurations at this time. However, this policy may be subject to change in the future.

Device Drivers

If your application needs to access its I/O devices on the target, then the real-time program must contain device driver code to handle communication with the I/O boards. The Real-Time Workshop DOS run-time interface includes source code of the device drivers for the Keithley Metrabyte DAS-1600/1400 Series I/O boards. See “Device Driver Blocks” on page 13-10 for information on how to use these blocks.

Simulink Host

The development host must have Windows 95, Windows 98, or Windows NT to run Simulink. However, the real-time target requires only DOS, since the executable built from the generated code is not a Windows application. The real-time target will not run in a “DOS box” (i.e., a DOS window on Windows 95/98/NT).

Although it is possible to reboot the host PC under DOS for real-time execution, the computer would need to be rebooted under Windows 95/NT for any subsequent changes to the block diagram in Simulink. Since this process of repeated rebooting the computer is inconvenient, we recommend a second PC running only DOS as the real-time target.

Sample Rate Limits

Program timing is controlled by installing an interrupt service routine that executes the model code. The target PC’s CPU is then interrupted at the specified rate (this rate is determined from the step size).

The rate at which interrupts occur is controlled by application code supplied with the Real-Time Workshop. This code uses the PC-AT’s 8254 Counter/Timer to determine when to generate interrupts.

The code that sets up the 8254 Timer is in `drt_time.c`, which is in the `matlabroot\rtw\c\dos\rti` directory. It is automatically linked in when you build the program using the DOS real-time template makefile.

The 8254 Timer is a 16-bit counter that operates at a frequency of 1.193 MHz. However, the timing module `drt_time.c` in the DOS run-time interface can extend the range by an additional 16 bits in software, effectively yielding a

32-bit counter. This means that the slowest base sample rate your model can have is

$$1.193\times10^{6} \div (2^{32} - 1) \approx \frac{1}{3600} Hz$$

This corresponds to a maximum base step size of approximately one hour.

The fastest sample rate you can define is determined by the minimum value from which the counter can count down. This value is 3, hence the fastest sample rate that the 8254 is capable of achieving is

$$1.193 \times 10^{6} \div 3 \approx 4 \times 10^{5}\,\mathrm{Hz}$$

This corresponds to a minimum base step size of

$$1 \div 4 \times 10^{5} \approx 2.5 \times 10^{-6}\,\mathrm{seconds}$$

However, note that the above number corresponds to the fastest rate at which the timer can generate interrupts. It does not account for execution time for the model code, which would substantially reduce the fastest sample rate possible for the model to execute in real time. Execution speed is machine dependent and varies with the type of processor and the clock rate of the processor on the target PC.

The slowest and fastest rates computed above refer to the base sample times in the model. In a model with more than one sample time, you can define blocks that execute at slower rates as long as the sample times are an integer multiple of the base sample time.

Modifying Program Timing

If you have access to an alternate timer (e.g., some I/O boards include their own clock devices), you can replace the file `drt_time.c` with an equivalent file that makes use of the separate clock source. See the comments in `drt_time.c` to understand how the code works.

You can use your version of the timer module by redefining the `TIMER_OBJS` macros with the build command. For example, in the Real-Time Workshop page of the **Simulation parameters** dialog box, changing the build command to

```
make_rtw TIMER_OBJS=my_timer.obj
```

replaces the file `drt_time.c` with `my_timer.c` in the list of source files used to build the program.

Device Driver Blocks

The real-time program communicates with external hardware via a set of device drivers. These device drivers contain the necessary code for interfacing to specific I/O devices.

The Real-Time Workshop includes device drivers for commercially available Keithley Metrabyte DAS-1600/1400 Series I/O boards. These device drivers are implemented as C MEX S-functions to interface with Simulink. This means you can add them to your model like any other block.

In addition, each of these S-function device drivers has a corresponding target file to inline the device driver in the model code. See “Creating Device Drivers” on page 17-34 for information on implementing your own device drivers.

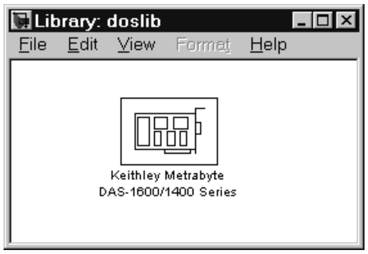
Since the device drivers are provided as source code, you can use these device drivers as a template to serve a a starting point for creating custom device drivers for other I/O boards.

Device Driver Block Library

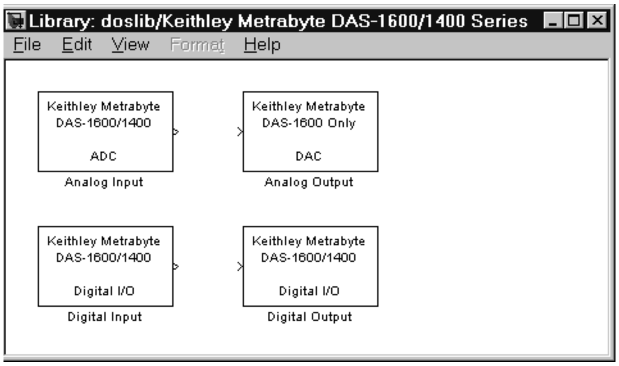
The device driver blocks for the Keithley Metrabyte DAS-1600/1400 Series I/O boards designed for use with DOS applications are contained in a block library called `doslib` (`matlabroot\toolbox\rtw\doslib.mdl`). To display this library, type

```
doslib
```

at the MATLAB prompt. This window will appear.



To access the device driver blocks, double-click on the sublibrary icon.


Library: doslib/Keithley Metrabyte DAS-1600/1400 Series
File Edit View Format Help
Keithley Metrabyte DAS-1600/1400 ADC
Analog Input
Keithley Metrabyte DAS-1600 Only DAC
Analog Output
Keithley Metrabyte DAS-1600/1400 Digital I/O
Digital Input
Keithley Metrabyte DAS-1600/1400 Digital I/O
Digital Output

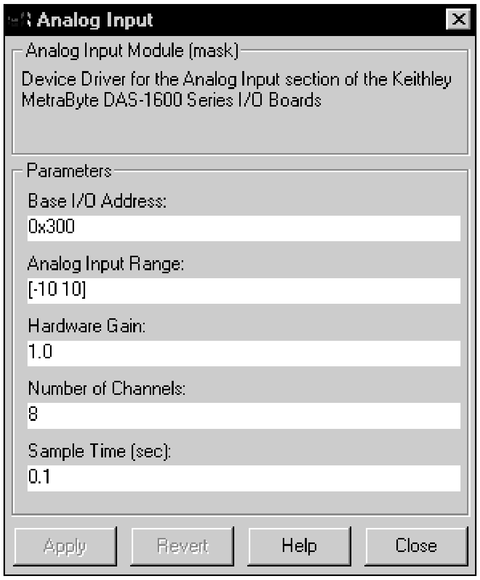
The blocks in the library contain device drivers that can be used for the DAS-1600/1400 Series I/O boards. The DAS-1601/1602 boards have 16 analog input (ADC) channels, two 12-bit analog output (DAC) channels and 4-bits of digital I/O. The DAS-1401/1402 boards do not have DAC channels. The DAS-1601/1401 boards have high programmable gains (1, 10, 100 and 500), while the DAS-1602/1402 boards offer low programmable gains (1, 2, 4 and 8).

For more information, contact the manufacturer via the Web site: `http://www.keithley.com`. The documentation for the DAS-1600/1400 Series I/O boards is the *DAS-1600/1400 Series User's Guide, Revision B* (Part Number: 80940).

Configuring Device Driver Blocks

Each device driver block has a dialog box that you use to set configuration parameters. As with all Simulink blocks, double-clicking on the block displays the dialog box. Some of the device driver block parameters (such as Base I/O Address) are hardware specific and are set either at the factory or configured via DIP switches at the time of installation.

Analog Input (ADC) Block Parameters



Analog Input Module (mask)

Device Driver for the Analog Input section of the Keithley MetraByte DAS-1600 Series I/O Boards

Parameters

Base I/O Address:
0x300

Analog Input Range:
[-10 10]

Hardware Gain:
1.0

Number of Channels:
8

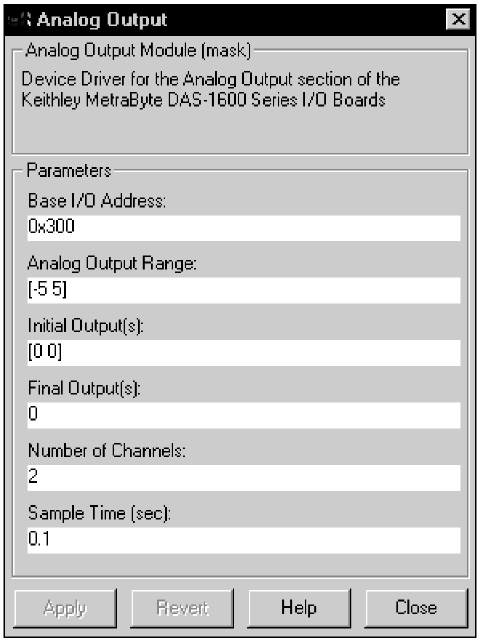
Sample Time (sec):
0.1

- **Base I/O Address**: The beginning of the I/O address space assigned to the board. The value specified here must match the board's configuration. Note that this parameter is a hexadecimal number and must be entered in the dialog as a MATLAB string (e.g., `'0x300'`).
- **Analog Input Range**: This two-element vector specifies the range of values supported by the ADC. The specified range must match the I/O board's settings. Specifically, the DAS-1600/1400 Series boards switches can be configured to either `[0 10]` for unipolar or `[-10 10]` for bipolar input signals.
- **Hardware Gain**: This parameter specifies the programmable gain that is applied to the input signal before presenting it to the ADC. Specifically, the DAS-1601/1401 boards have programmable gains of 1, 10, 100, and 500. The DAS-1602/1402 boards have programmable gains of 1, 2, 4, and 8. Configure the Analog Input Range and the Hardware Gain parameters depending on the type and range of the input signal being measured. For example, a DAS-1601 board in bipolar configuration with a programmable gain of 100 is best suited to measure input signals in the range between [±10v] ÷ 100 = ±0.1v.

  Voltage levels beyond this range will saturate the block output form the ADC block. Please adhere to manufacturers' electrical specifications to avoid damage to the board.
- **Number of Channels**: The number of analog input channels enabled on the I/O board. The DAS-1600/1400 Series boards offer up to 16 ADC channels when configured in unipolar mode (8 ADC channels if you select differential mode). The output port width of the ADC block is equal to the number of channels enabled.

- **Sample Time (sec)**: Device drivers are discrete blocks that require you to specify a sample time. In the generated code, these blocks are executed at the specified rate. Specifically, when the ADC block is executed, it causes the ADC to perform a single conversion on the enabled channels, and the converted values are written to the block output vector.

Analog Output (DAC) Block Parameters



Analog Output Module (mask)

Device Driver for the Analog Output section of the Keithley MetraByte DAS-1600 Series I/O Boards

Parameters

Base I/O Address: 0x300

Analog Output Range: [-5 5]

Initial Output(s): [0 0]

Final Output(s): 0

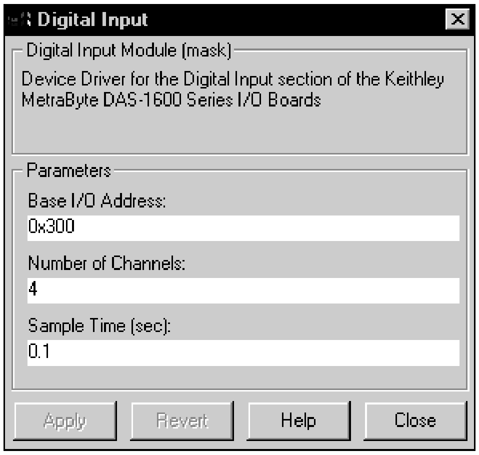
Number of Channels: 2

Sample Time (sec): 0.1

- **Base I/O Address**: The beginning of the I/O address space assigned to the board. The value specified here must match the board's configuration. Note that this parameter is a hexadecimal number and must be entered in the dialog as a MATLAB string (e.g., `'0x300'`).
- **Analog Output Range**: This parameter specifies the output range settings of the DAC section of the I/O board. Typically, unipolar ranges are between `[0 10]` volts and bipolar ranges are between `[-10 10]` volts. Refer to the DAS-1600 documentation for other supported output ranges.
- **Initial Output(s)**: This parameter can be specified either as a scalar or as an N element vector, where N is the number of channels. If a single scalar value is entered, the same scalar is applied to output. The specified initial output(s) is written to the DAC channels in the `mdlInitializeConditions` function.
- **Final Output(s)**: This parameter is specified in a manner similar to the Initial Output(s) parameter except that the specified final output values are written out to the DAC channels in the `mdlTerminate` function. Once the generated code completes execution, the code sets the final output values prior to terminating execution.

- **Number of Channels**: Number of DAC channels enabled. The DAS-1600 Series I/O boards have two 12-bit DAC channels. The DAS-1400 Series I/O boards do not have any DAC channels. The input port width of this block is equal to the number of channels enabled.
- **Sample Time (sec)**: DAC device drivers are discrete blocks that require you to specify a sample time. In the generated code, these blocks are executed at the specified rate. Specifically, when the DAC block is executed, it causes the DAC to convert a single value on each of the enabled DAC channels, which produces a corresponding voltage on the DAC output pin(s).

Digital Input Block Parameters


Digital Input

Digital Input Module (mask)

Device Driver for the Digital Input section of the Keithley MetraByte DAS-1600 Series I/O Boards

Parameters

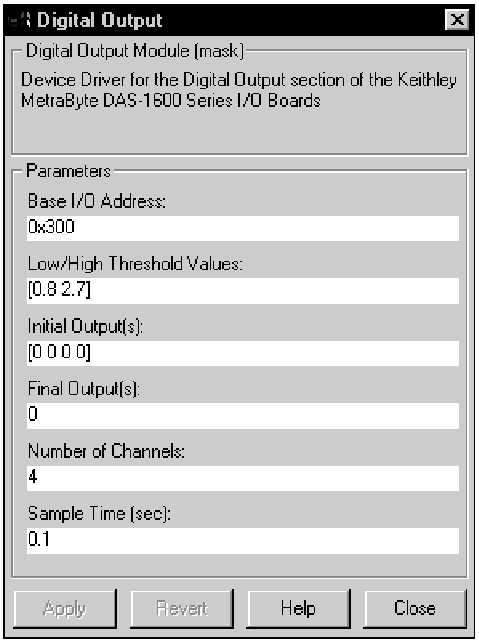
Base I/O Address:
0x300

Number of Channels:
4

Sample Time (sec):
0.1

- **Base I/O Address**: The beginning of the I/O address space assigned to the board. The value specified here must match the board's configuration. Note that this parameter is a hexadecimal number and must be entered in the dialog as a MATLAB string (e.g., `'0x300'`).
- **Number of Channels**: This parameter specifies the number of 1-bit digital input channels being enabled. This parameter also determines the output port width of the block in Simulink. Specifically, the DAS-1600/1400 Series boards provide four bits (i.e., channels) for digital I/O.
- **Sample Time (sec)**: Digital input device drivers are discrete blocks that require you to specify a sample time. In the generated code, these blocks are executed at the specified rate. Specifically, when the digital input block is executed, it reads a boolean value from the enabled digital input channels. The corresponding input values are written to the block output vector.

Digital Output Block Parameters



Digital Output Module (mask)

Device Driver for the Digital Output section of the Keithley MetraByte DAS-1600 Series I/O Boards

Parameters

Base I/O Address: 0x300

Low/High Threshold Values: [0.8 2.7]

Initial Output(s): [0 0 0 0]

Final Output(s): 0

Number of Channels: 4

Sample Time (sec): 0.1

- **Base I/O Address**: The beginning of the I/O address space assigned to the board. The value specified here must match the board's configuration. Note that this parameter is a hexadecimal number and must be entered in the dialog as a MATLAB string (e.g., `'0x300'`).
- **Low/High Threshold Values**: This parameter specifies the threshold levels, `[lo hi]`, for converting the block inputs into 0/1 digital values. The signal in the block diagram connected to the block input should rise above the high threshold level for a 0 to 1 transition in the corresponding digital output channel on the I/O board. Similarly, the input should fall below the low threshold level for a 1 to 0 transition.
- **Initial Output(s)**: Same as the Analog Output block, except the specified values are converted to 0 or 1 based on the lower threshold value before they are written to the corresponding digital output channel.
- **Final Output(s)**: Same as the Analog Output block, except the specified values are converted to 0 or 1 based on the lower threshold value before they are written to the corresponding digital output channel on the I/O board.
- **Number of Channels**: This parameter specifies the number of 1-bit digital I/O channels being enabled. This parameter also determines the output port width of the block. Specifically, the DAS-1600/1400 Series boards provide four bits (i.e., channels) for digital I/O.
- **Sample Time (sec)**: Digital output device drivers are discrete blocks that require you to specify a sample time. In the generated code, these blocks are executed at the specified rate. Specifically, when the digital output block is

executed, it causes corresponding boolean values to be output from the board's digital I/O channels.

Adding Device Driver Blocks to the Model

Add device driver blocks to the Simulink block diagram as you would any other block — simply drag the block from the block library and insert it into the model. Connect the ADC or Digital Input block to the model's inputs and connect the DAC or Digital Output block to the model's outputs.

Including Device Driver Code

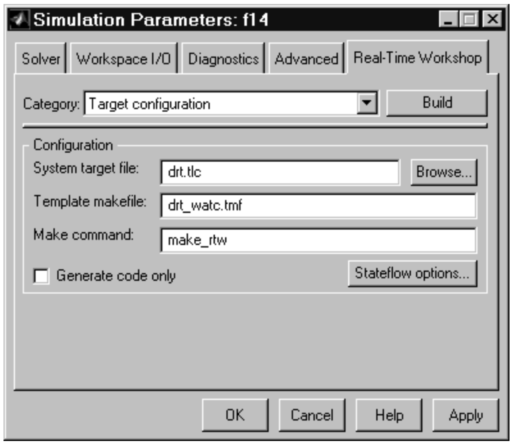
Device driver blocks are implemented as S-functions written in C. The C code for a device driver block is compiled as a MEX-file so that it can be called by Simulink. See the *MATLAB Application Program Interface Guide* for information on MEX-files.

The same C code can also be compiled and linked to the generated code just like any other C-coded, S-function. However, by using the target (`.tlc`) file that corresponds to each of the C file S-functions, the device driver code is inlined in the generated code.

The `matlabroot\rtw\c\dos\devices` directory contains the MEX-files, C files, and target files (`.tlc`) for the device driver blocks included in `doslib`. This directory is automatically added to your MATLAB path when you include any of the blocks from `doslib` in your model.

Building the Program

Once you have created your Simulink model and added the appropriate device driver blocks, you are ready to build a DOS target application. To do this, select the Real-Time Workshop page of the **Simulink parameters** dialog box, and select **Target configuration** from the **Category** menu.



Category: Target configuration

Configuration

System target file: drt.tlc

Template makefile: drt_watc.tmf

Make command: make_rtw

☐ Generate code only

Click **Browse to** open the System Target File Browser. Select `drt.tlc`; this automatically fills in the correct files as shown above:

- `drt.tlc` as the **System target file**
- `drt_watc.tmf` as the **Template makefile**. This is used with the Watcom compiler, assembler, linker, and `WMAKE` utility.
- `make_rtw` as the **Make command**

You can specify Target Language Compiler options in the **System target file** field following `drt.tlc`. You can also specify and make options in the **Make command** field. See Chapter 3, “Code Generation and the Build Process” for descriptions of the available Target Language Compiler and make options.

The DOS system target file, `drt.tlc`, and the template makefile, `drt_watc.tmf`, are located in the `matlab\rtw\c\dos` directory.

The template makefile assumes that the Watcom C/386 compiler, assembler, and linker have been correctly installed on the host workstation. You can verify this by checking the environment variable, WATCOM, which correctly points to the directory where the Watcom files are installed.

The program builder invokes the Watcom wmake utility on the generated makefile, so the directory where wmake is installed must be on your path.

Running the Program

The result of the build process is a DOS 32-bit protected-mode executable. The default name of the executable is *model*.exe, where *model* is the name of your Simulink model. You must run this executable in DOS; you cannot run the executable in Windows 95/98/NT.

Custom Code Blocks

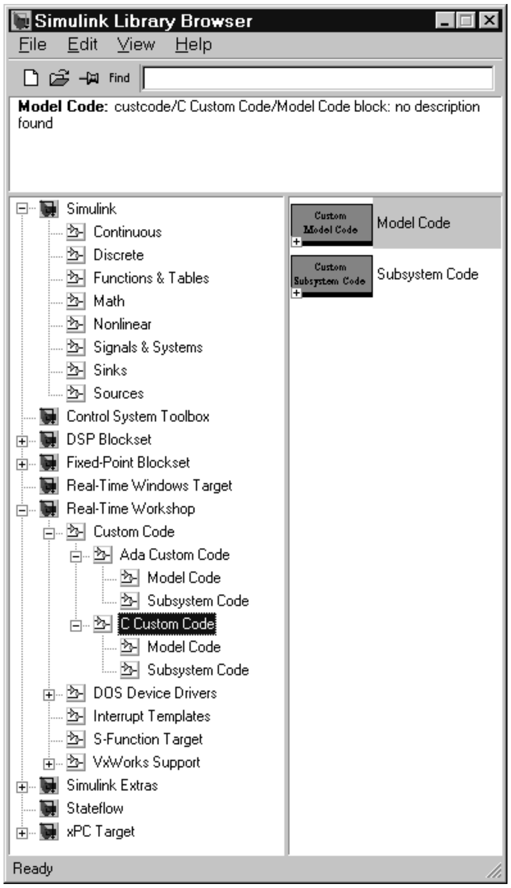
Introduction

This chapter discusses the Custom Code library, a collection of blocks that allow you to insert custom code into the generated source code files and/or functions associated with your model.

The Custom Code library is part of the Real-Time Workshop library. You can access the Real-Time Workshop library via the Simulink Library Browser as shown in Figure 14-1. Alternatively, you can access the Real-Time Workshop library by typing the command

```
rtwlib
```

at the MATLAB command prompt.



Model Code: custcode/C Custom Code/Model Code block: no description found

- Simulink
  - Continuous
  - Discrete
  - Functions & Tables
  - Math
  - Nonlinear
  - Signals & Systems
  - Sinks
  - Sources
- Control System Toolbox
- DSP Blockset
- Fixed-Point Blockset
- Real-Time Windows Target
- Real-Time Workshop
  - Custom Code
    - Ada Custom Code
      - Model Code
      - Subsystem Code
    - C Custom Code
      - Model Code
      - Subsystem Code
  - DOS Device Drivers
  - Interrupt Templates
  - S-Function Target
  - VxWorks Support
- Simulink Extras
- Stateflow
- xPC Target

Model Code

Subsystem Code

Figure 14-1: Custom Code Library and its Sublibraries

This chapter discusses only the C Custom Code library and its sublibraries.

Note that, depending on which MathWorks products you have installed, your browser may show a different collection of libraries.

There are four other sublibraries in the Real-Time Workshop library:

- DOS Device Drivers — Blocks for use with DOS. See Chapter 13, “Targeting DOS for Real-Time Applications” for information.

- Interrupt Templates — A collection of blocks that you can use as templates for building your own asynchronous interrupts. See Chapter 15, “Asynchronous Support” for information.
- S-Function Target — The S-Function Target sublibrary contains only one block type, the Real-Time Workshop S-Function block. This block is intended for use with generated S-functions. See Chapter 10, “The S-Function Target” for more information.
- VxWorks Support — A collection of blocks that support VxWorks (Tornado). See Chapter 12, “Targeting Tornado for Real-Time Applications” for information on VxWorks.

Custom Code Library

The Custom Code library contains blocks that allow you to place your own code, in C or in Ada, inside the code generated by the Real-Time Workshop.

Figure 14-1 illustrates the hierarchy of sublibraries under the Real-Time Workshop library. First, the Custom Code library has two sublibraries:

- C Custom Code
- Ada Custom Code

Both the C Custom Code and Ada Custom Code libraries, in turn, have identically named sublibraries:

- Model Code
- Subsystem Code

Both sublibraries contain blocks that target specific files and subsystems within which you can place your code.

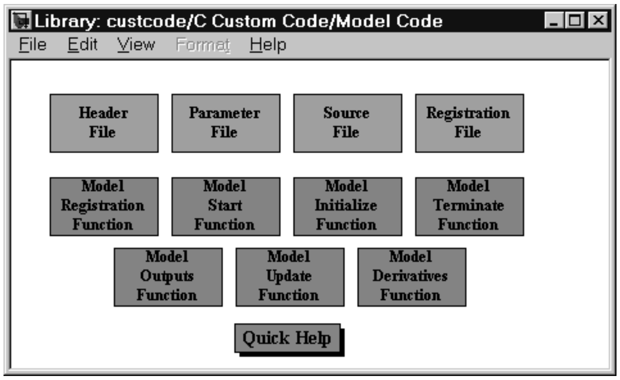
The following sections discuss the Model Code and Subsystem Code sublibraries of the C Custom Code library.

Model Code Sublibrary

The Model Code sublibrary contains 10 blocks that insert custom code into the generated model files and functions. You can view the blocks either by:

- Expanding the Model Code sublibrary (under C Custom Code) in the Simulink Library Browser
- Right-clicking on the Model Code sublibrary icon in the right pane of the Simulink Library Browser

The latter method opens this window.



The four blocks on the top row contain texts fields to insert custom code at the top and bottom of the following files:

- `model.h` — Header File block
- `model_prm.h` — Parameter File block
- `model.c` — Source File block
- `model_reg.h` — Registration File block

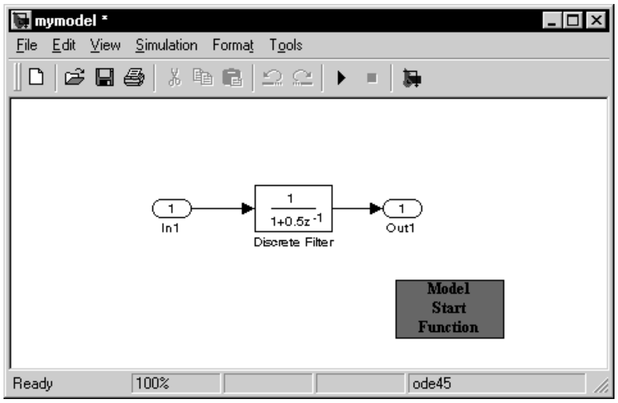
The six function blocks in the second and third rows contain text fields to insert critical code sections at the top and bottom of these designated model functions:

- `Registration function` — Registration Function block
- `MdlStart` — MdlStart Function block
- `MdlTerminate` — MdlTerminate Function block
- `MdlOutputs` — MdlOutputs Function block
- `MdlUpdate` — MdlUpdate Function block
- `MdlDerivatives` — MdlDerivatives Function block

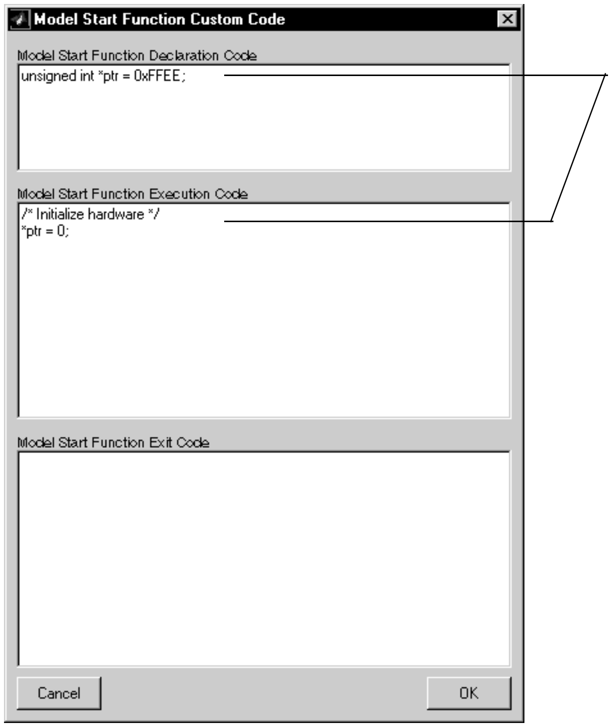
Each block provides a dialog box that contains three fields.

Example: Using a Custom Code Block

The following example uses an MdlStart Function block to introduce code into the `MdlStart` function. The diagram below shows a simple model with the Model Start Function block inserted.



Double-clicking the Model Start Function block opens the **Model Start Function Custom Code** dialog box.



Model Start Function Declaration Code

```
unsigned int *ptr = 0xFFEE;
```

Model Start Function Execution Code

```
/* Initialize hardware */
*ptr = 0;
```

Model Start Function Exit Code

The Real-Time Workshop inserts the code entered here into the MdlStart function in the generated code.

You can insert custom code into any or all of the available text fields.

The code below is the `MdlStart` function for this example (`mymodel`).

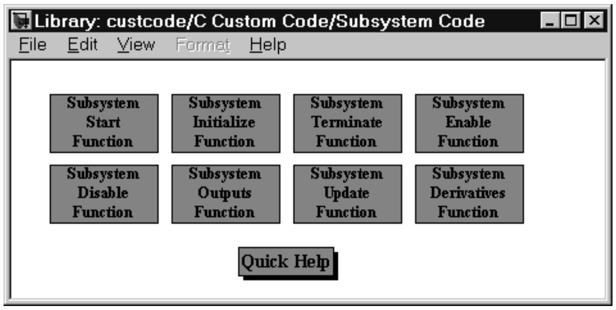
```
void MdlStart(void)
{
  /* user code (Start function Header) */
  /* System: <Root> */
  unsigned int *ptr = 0xFFEE;

  /* user code (Start function Body) */
  /* System: <Root> */
  /* Initialize hardware */
  *ptr = 0;

  /* state initialization */
  /* DiscreteFilter Block: <Root>/Discrete Filter */
  rtX.d.Discrete_Filter = 0.0;
}
```

The custom code entered in the **Model Start Function Custom Code** dialog box is embedded directly in the generated code. Note that each block of custom code is tagged with a comment such as

```
/* user code (Start function Header) */
```

Subsystem Code Sublibrary

The Subsystem Code sublibrary contains eight blocks to insert critical code sections into system functions.


Library: custcode/C Custom Code/Subsystem Code
File Edit View Format Help
Subsystem Start Function
Subsystem Initialize Function
Subsystem Terminate Function
Subsystem Enable Function
Subsystem Disable Function
Subsystem Outputs Function
Subsystem Update Function
Subsystem Derivatives Function
Quick Help

Each of these blocks has a dialog box containing two text fields that allow you to place data at the top and the bottom of system functions. The eight blocks are:

- Subsystem Start
- Subsystem Initialize
- Subsystem Terminate
- Subsystem Enable
- Subsystem Disable
- Subsystem Outputs
- Subsystem Update
- Subsystem Derivatives

The location of the block in your model determines the location of the custom code. In other words, the code is local to the subsystem that you select. For example, the Subsystem Outputs block places code in `mdlOutputs` when the code block resides in the root model. If the Subsystem Outputs block resides in a triggered or enabled subsystem, however, the code is placed in the subsystem's Outputs function.

The ordering for a triggered or enabled system is:

1. Output entry
2. Output exit
3. Update entry
4. Update exit code

Asynchronous Support

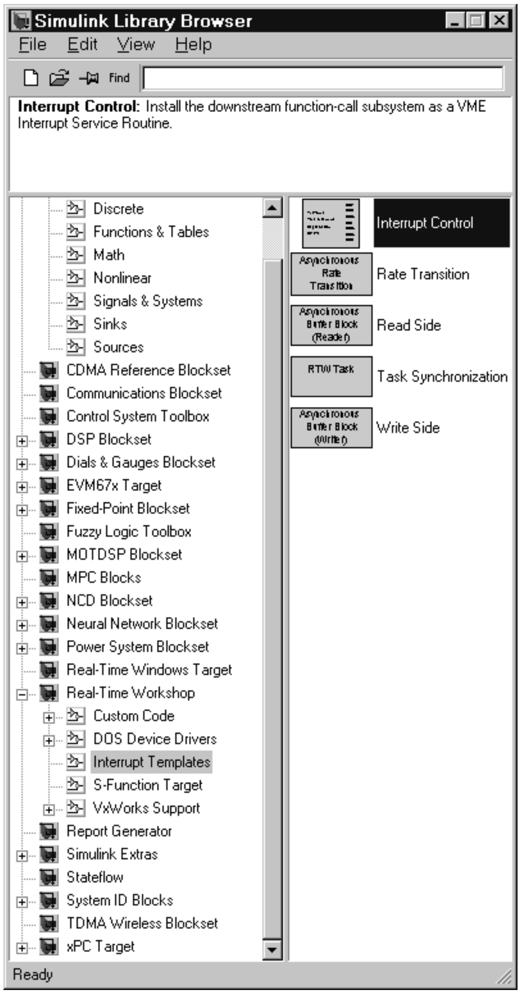
Introduction

The Interrupt Templates are blocks that you can use as templates for building your own asynchronous interrupts.

The Interrupt Templates library is part of the Real-Time Workshop library. You can access the Real-Time Workshop library via the Simulink Library Browser as shown in Figure 15-1. Alternatively, you can access the Real-Time Workshop library by typing the command

```
rtwlib
```

at the MATLAB command prompt.



Interrupt Control: Install the downstream function-call subsystem as a VME Interrupt Service Routine.

Figure 15-1: Interrupt Templates in Simulink Library Browser

Note that, depending on which MathWorks products you have installed, your browser may show a different collection of libraries.

Other sublibraries in the Real-Time Workshop library are:

- DOS Device Drivers: Blocks for use with DOS. See Chapter 13, “Targeting DOS for Real-Time Applications” for information.
- Custom Code Blocks: Blocks that allow you to insert custom code into the generated source code files and/or functions associated with your model. See Chapter 14, “Custom Code Blocks.” for information.
- S-Function Target: The S-Function Target sublibrary contains only one block type, the RTW S-Function block. This block is intended for use with generated S-functions. See Chapter 10, “The S-Function Target” for more information.
- VxWorks Support: A collection of blocks that support VxWorks (Tornado). See Chapter 12, “Targeting Tornado for Real-Time Applications” for information on VxWorks.

Interrupt Handling

The blocks in the Interrupt Templates library allow you to model synchronous/asynchronous event handling, including interrupt service routines (ISRs). These blocks include:

- Asynchronous Buffer block (read)
- Asynchronous Buffer block (write)
- Asynchronous Interrupt block
- Asynchronous Rate Transition block
- Task Synchronization block

Using these blocks, you can create models that handle asynchronous events, such as hardware generated interrupts and asynchronous read and write operations. The following sections discuss each of these blocks in the context of VxWorks Tornado operating system.

Asynchronous Interrupt Block

Interrupt service routines (ISR) are realized by connecting the outputs of the VxWorks Asynchronous Interrupt block to the control input of a function-call subsystem, the input of a VxWorks Task Synchronization block, or the input to a Stateflow chart configured for a function-call input event.

The Asynchronous Interrupt block installs the downstream (destination) function-call subsystem as an ISR and enables the specified interrupt level. The current implementation of the VxWorks Asynchronous Interrupt block supports VME interrupts 1-7 and uses the VxWorks system calls `sysIntEnable`, `sysIntDisable`, `intConnect`, `intLock` and `intUnlock`. Ensure that your target architecture (BSP) for VxWorks supports these functions.

When a function-call subsystem is connected to an Asynchronous Interrupt block output, the generated code for that subsystem becomes the ISR. For large subsystems, this can have a large impact on interrupt response time for interrupts of equal and lower priority in the system. As a general rule, it is best to keep ISRs as short as possible. To do this, you should only connect function-call subsystems that contain few blocks.

A better solution for large systems is to use the Task Synchronization block to synchronize the execution of the function-call subsystem to an event. The Task Synchronization block is placed between the Asynchronous Interrupt block and the function-call subsystem (or Stateflow chart). The Asynchronous Interrupt block then installs the Task Synchronization block as the ISR, which releases a synchronization semaphore (performs a `semGive`) to the function-call subsystem and then returns. See the VxWorks Task Synchronization block for more information.

Using the Asynchronous Interrupt Block

The Asynchronous Interrupt block has two modes that help support rapid prototyping:

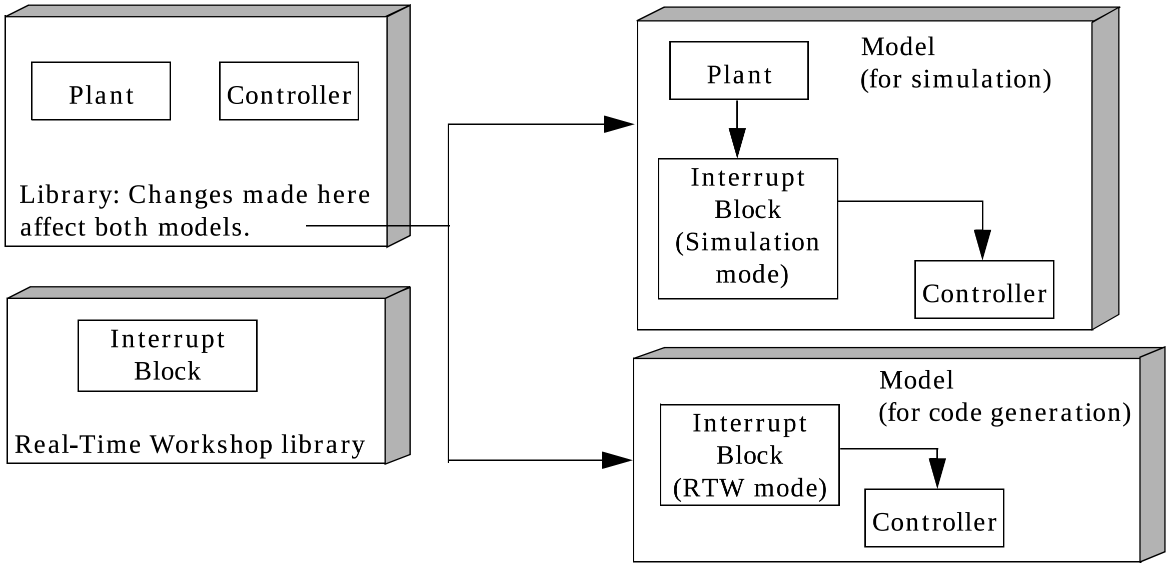
- *RTW mode*. In RTW mode, the Asynchronous Interrupt block configures the downstream system as an ISR and enables interrupts during model startup. You can select this mode using the Asynchronous Interrupt block dialog box when generating code.
- *Simulation mode*. In Simulation mode, simulated Interrupt Request (IRQ) signals are routed through the Asynchronous Interrupt block's trigger port. Upon receiving a simulated interrupt, the block calls the associated system.

  You should select this mode when simulating, in Simulink, the effects of an interrupt signal. Note that there can only be one VxWorks Asynchronous Interrupt block in a model and all desired interrupts should be configured by it.

In both RTW and Simulation mode, in the event that two IRQ signals occur simultaneously, the Asynchronous Interrupt block executes the downstream systems according to their priority interrupt level.

The Asynchronous Interrupt block provides these two modes to make the development and implementation of real-time systems that include ISRs easier and quicker. You can develop two models, one that includes a plant and a controller for simulation, and one that only includes the controller for code generation.

Using the Library feature of Simulink, you can implement changes to both models simultaneously. Figure 15-1 illustrates how changes made to the plant or controller, both of which are in a library, propagate to the models.



Plant
Controller
Library: Changes made here affect both models.
Interrupt Block
Real-Time Workshop library
Model (for simulation)
Plant
Interrupt Block (Simulation mode)
Controller
Model (for code generation)
Interrupt Block (RTW mode)
Controller

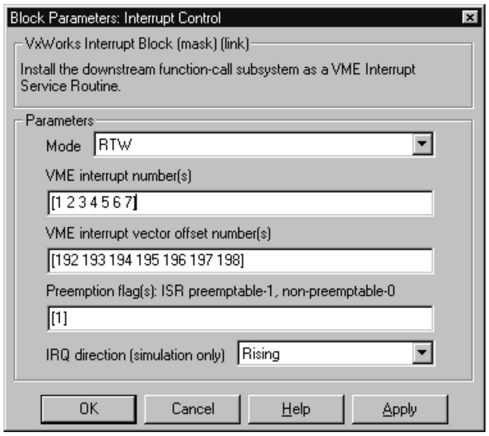
Figure 15-1: Using the Asynchronous Interrupt Block with Simulink Library Feature in Rapid Prototyping Process

Real-Time Workshop models normally run from a periodic interrupt. All blocks in a model run at their desired rate by executing them in multiples of the timer interrupt rate. Asynchronous blocks, on the other hand, execute based on other interrupt(s) that may or may not be periodic.

The hardware that generates the interrupt is not configured by the Asynchronous Interrupt block. Typically, the interrupt source is a VME I/O board, which generates interrupts for specific events (e.g., end of A/D conversion). The VME interrupt level and vector are set up in registers or by using jumpers on the board. You can use the `mdlStart` routine of a user-written device driver (S-function) to set up the registers and enable interrupt generation on the board. You must match the interrupt level and vector specified in the Asynchronous Interrupt block dialog to the level and vector setup on the I/O board.

Asynchronous Interrupt Block Parameters

The picture below shows the VxWorks Asynchronous Interrupt block dialog box.



Block Parameters: Interrupt Control

VxWorks Interrupt Block (mask) (link)

Install the downstream function-call subsystem as a VME Interrupt Service Routine.

Parameters

Mode: RTW

VME interrupt number(s): [1 2 3 4 5 6 7]

VME interrupt vector offset number(s): [192 193 194 195 196 197 198]

Preemption flag(s): ISR preemptable-1, non-preemptable-0: [1]

IRQ direction (simulation only): Rising

Parameters associated with the Asynchronous Interrupt block are:

- Mode: In Simulation mode, the ISRs are executed nonpreemptively. If they occur simultaneously, signals are executed in the order specified by their number (1 being the highest priority). Interrupt mapping during simulation is left to right, top to bottom. That is, the first control input signal maps to the topmost ISR. The last control input signal maps to the bottom most ISR.

  In RTW mode, the Real-Time Workshop uses `vxinterrupt.tlc` to realize asynchronous interrupts in the generated code. The ISR is passed one argument, the root `SimStruct`, and the Simulink definition of the function-call subsystem is remapped to conform with the information in the `SimStruct`.
- VME Interrupt Number(s): Specify the VME interrupt numbers for the interrupts to be installed. The valid range is 1-7; for example: `[4 2 5]`).
- VME Interrupt Vector Offset Number(s): The Real-Time Workshop uses this number in the call to `intConnect(INUM_TO_IVEC(#),...)`. You should specify a unique vector offset number for each interrupt number.
- Preemption Flag(s): By default, higher priority interrupts can preempt lower priority interrupts in VxWorks. If desired, you can lock out interrupts during

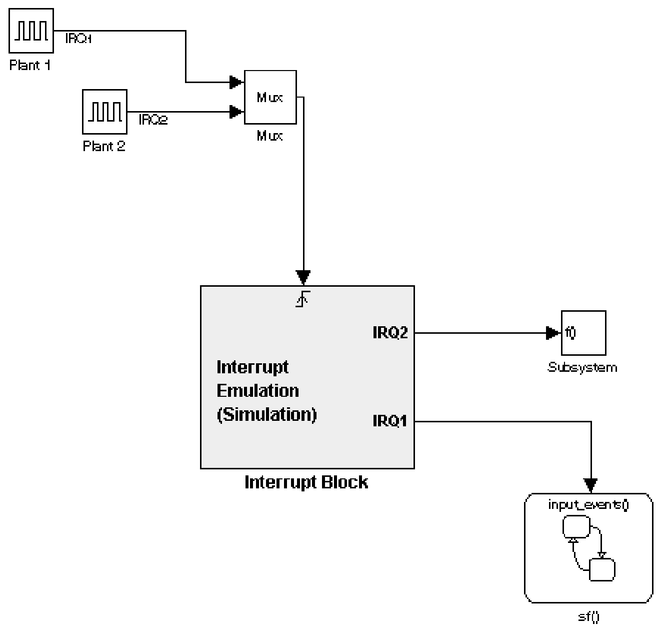
the execution of a ISR by setting the preemption flag to 0. This causes `intLock()` and `intUnlock()` calls to be inserted at the beginning and end of the ISR respectively. This should be used carefully since it increases the system's interrupt response time for all interrupts at the `intLockLevelSet()` level and below.

- IRQ Direction: In simulation mode, a scalar IRQ direction is applied to all control inputs, and is specified as 1 (rising), -1 (falling), or 0 (either). Configuring inputs separately in simulation is done prior to the control input. For example, a Gain block set to -1 prior to a specific IRQ input will change the behavior of one control input relative to another. In RTW mode the IRQ direction parameter is ignored.

Asynchronous Interrupt Block Example - Simulation Mode

This example shows how the Asynchronous Interrupt block works in simulation mode.

Simulated Interrupt Signals

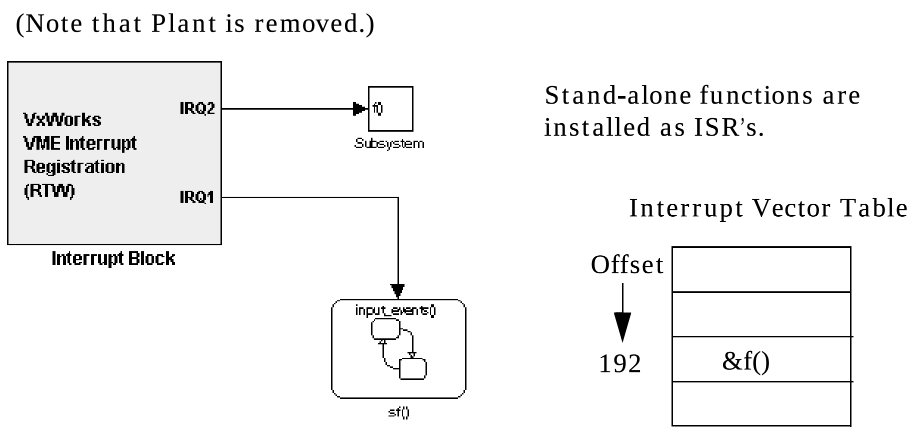


The Asynchronous Interrupt block works as a “handler” that routes signals and sets priority. If two interrupts occur simultaneously, the rule for handling which signal is sent to which port is left to right and top to bottom. This means that IRQ2 receives the signal from Plant 1 and IRQ1 receives the signal from Plant 2 simultaneously. IRQ1 still has priority over IRQ2 in this situation.

Note that the Asynchronous Interrupt block executes during simulation by processing incoming signals and executing downstream functions. Also, interrupt preemption cannot be simulated.

Asynchronous Interrupt Block Example - RTW Mode

This example shows the Asynchronous Interrupt block in RTW mode.



In this example, the simulated plant signals that were included in the previous example have been removed. In RTW mode, the Asynchronous Interrupt block receives interrupts directly from the hardware.

During the Target Language Compiler phase of code generation, the Asynchronous Interrupt block installs the code in the Stateflow chart and the Subsystem block as interrupt service routines. Configuring a function-call subsystem as an ISR requires two function calls, `int_connect` and `int_enable`. For example, the function `f(u)` in the Function block requires that the Asynchronous Interrupt block inserts a call to `int_connect` and `sysIntEnable` in the `mdlStart` function, as shown below.

```
/* model start function */
MdlStart()
{
  . . .
  int_connect(f,192,1);
  . . .
  sysIntEnable(1);
  . . .

}
```

Locking and Unlocking ISRs. It is possible to lock ISRs so that they are not preempted by a higher priority interrupt. Configuring the interrupt as nonpreemptive has this effect. The following code fragment shows where the Real-Time Workshop places the `int_lock` and `int_unlock` functions to configure the interrupt as nonpreemptive.

```
f()

{
  lock = int_lock();   }
  . . .                }  Real-Time Workshop code
  . . .                }
  . . .
  int_unlock(lock);
}
```

Finally, the model's terminate function disables the interrupt:

```
/* model terminate function */
MdlTerminate()
{
  ...
  int_disable(1);
  ...
}
```

Task Synchronization Block

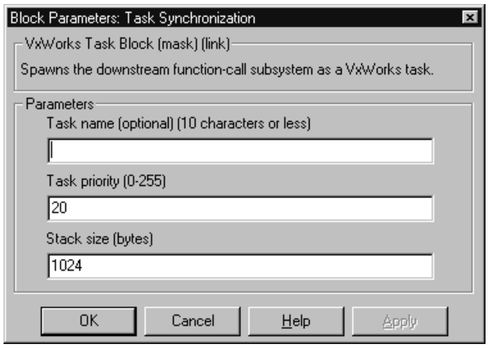
The VxWorks Task Synchronization block is a function-call subsystem that spawns, as an independent VxWorks task, the function-call subsystem connected to its output. Typically it would be placed between the VxWorks Asynchronous Interrupt block and a function-call subsystem block or a Stateflow chart. Another example would be to place the Task Synchronization block at the output of a Stateflow diagram that has an Event, “Output to Simulink,” configured as a function-call.

The VxWorks Task Synchronization block performs the following functions:

- The downstream function-call subsystem is spawned as an independent task using the VxWorks system call `taskSpawn()`. The task is deleted using `taskDelete()` during model termination.
- A semaphore is created to synchronize the downstream system to the execution of the Task Synchronization block.
- Code is added to this spawned function-call subsystem to wrap it in an infinite while loop.
- Code is added to the top of the infinite while loop of the spawned task to wait on a the semaphore, using `semTake()`. When `semTake()` is first called, `NO_WAIT` is specified. This allows the task to determine if a second `semGive()` has occurred prior to the completion of the function-call subsystem. This would indicate the interrupt rate is too fast or the task priority is too low.
- Synchronization code, i.e., `semgive()`, is generated for the Task Synchronization block (a masked function-call subsystem). This allows the output function-call subsystem to run. As an example, if you connect the Task Synchronization block to the output of a VxWorks Asynchronous Interrupt block, only a `semGive()` would occur inside an ISR.

Task Synchronization Parameters

The picture below shows the VxWorks Task Synchronization block dialog box.



Block Parameters: Task Synchronization

VxWorks Task Block (mask) (link)

Spawns the downstream function-call subsystem as a VxWorks task.

Parameters

Task name (optional) (10 characters or less)

Task priority (0-255): 20

Stack size (bytes): 1024

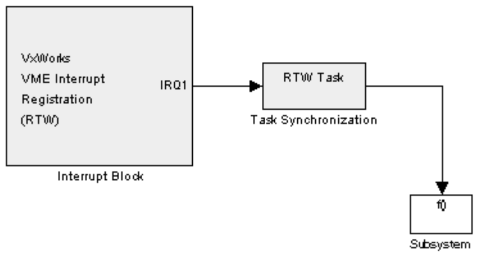
Parameters associated with the Task Synchronization block are:

- Task Name — An optional name, which if provided, is used as the first argument to the `taskSpawn()` system call. This name is used by VxWorks routines to identify the task they are called from to aid in debugging.
- Task Priority — The task priority is the VxWorks priority that the function-call subsystem task is given when it is spawned. The priority can be a very important consideration in relation to other tasks priorities in the VxWorks system. In particular, the default priority of the model code is 30 and, when multitasking is enabled, the priority of the each subrate task increases by one from the default model base rate. Other task priorities in the system should also be considered when choosing a task priority. VxWorks priorities range from 0 to 255 where a lower number is a higher priority.
- Stack Size — The function-call subsystem is spawned with the stack size specified. This is maximum size to which the task's stack can grow. The value should be chosen based on the number of local variables in the task.

  By default, Real-Time Workshop limits the number of bytes for local variables in all of the generated code to 8192 bytes (see assignment of `MaxStackSize` in *matlabroot*`/rtw/c/tornado/tornado.tlc`). As a rule, providing twice 8192 bytes (16384) for the one function that is being spawned as a task should be sufficient.

Task Synchronization Block Example

This example shows a Task Synchronization block as a simple ISR.



The Task Synchronization block inserts this code during the Target Language Compiler phase of code generation:

- In `MdlStart`, the Task Synchronization block is registered by the Asynchronous Interrupt block as an ISR. The Task Synchronization block creates and initializes the synchronization semaphore. It also spawns the function-call subsystem as an independent task.

```
/* Create and spawn task: <Root>/Faster Rate(.015) */
if ((*(SEM_ID *)rtPWork.s6_S_Function.SemID =
  semBCreate(SEM_Q_PRIORITY, SEM_EMPTY)) == NULL)
  ssSetErrorStatus(rtS,"semBCreate call failed "
                   "for block <Root>/Faster Rate(.015).\n ");
}
if ((rtIWork.s6_S_Function.TaskID = taskSpawn("root_Faster_", 20, VX_FP_TASK,
    1024, (FUNCPTR)Sys_root_Faster__OutputUpdate,
    (int_T)rtS, 0, 0, 0, 0, 0, 0, 0, 0, 0)) == ERROR) {
      ssSetErrorStatus(rtS,"taskSpawn call failed for block <Root>/ Faster Rate "
                           "(.015).\n");
  }
```

- The Task Synchronization block modifies the downstream function-call subsystem by wrapping it inside an infinite loop and adding semaphore synchronization code.

```
/* Output and update for function-call system: <Root>/Faster    Rate(.015) */
void Sys_root_Faster__OutputUpdate(void *reserved, int_T
                                   controlPortIdx, int_T tid)
{
  /* Wait for semaphore to be released by system: <Root>/Task Synchronization */
  for(;;) {
    if (semTake(*(SEM_ID *)rtPWork.s6_S_Function.SemID,NO_WAIT) != ERROR) {
      logMsg("Rate for function-call subsystem"
             "Sys_root_Faster__OutputUpdate() fast.\n",0,0,0,0,0,0);
#if STOPONOVERRUN
```

```
          logMsg("Aborting real-time simulation.\n",0,0,0,0,0,0);
          semGive(stopSem);
          return(ERROR);
    #endif
        } else {

          semTake(*(SEM_ID *)rtPWork.s6_S_Function.SemID, WAIT_FOREVER);
        }
        /* UniformRandomNumber Block: <S3>/Uniform Random Number */
        rtB.s3_Uniform_Random_Number =
        rtRWork.s3_Uniform_Random_Number.NextOutput;
       .
       .
       .
   }
```

Asynchronous Buffer Block

The VxWorks Asynchronous Buffer blocks are meant to be used to interface signals to asynchronous function-call subsystems in a model. This is needed whenever a function-call subsystem has input or output signals and its control input ultimately connects (sources) to the VxWorks Asynchronous Interrupt block or Task Synchronization block.

Because an asynchronous function-call subsystem can preempt or be preempted by other model code, an inconsistency arises when more than one signal element is connected to it. The issue is that signals passed to and/or from the function-call subsystem can be in the process of being written or read when the preemption occurs. Thus, partial old and partial new data will be used.

This situation can also occur with scalar signals in some cases. For example, if a signal is a double (8 bytes), the read or write operation may require two assembly instructions.

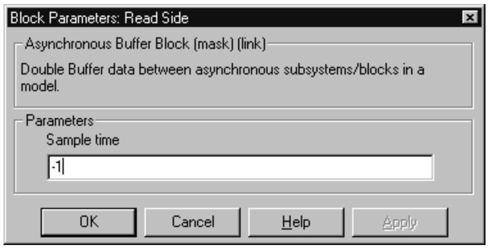
The Asynchronous Buffer blocks can be used to guarantee the data passed to and/or from the function-call subsystem is all from the same iteration.

The Asynchronous Buffer blocks are used in pairs, with a write side driving the read side. To ensure the data integrity, no other connections are allowed between the two Asynchronous Buffer blocks. The pair works by using two buffers (“double buffering”) to pass the signal and, by using mutually exclusive control, allow only exclusive access to each buffer. For example, if the write side is currently writing into one buffer, the read side can only read from the other buffer.

The initial buffer is filled with zeros so that if the read side executes before the write side has had time to fill the other buffer, the read side will collect zeros from the initial buffer.

Asynchronous Buffer Block Parameters

There are two kinds of Asynchronous Buffer blocks, a reader and a writer. The picture below shows the Asynchronous Buffer block's dialog boxes.


Block Parameters: Read Side

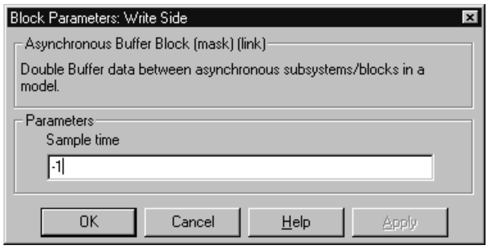
Asynchronous Buffer Block (mask) (link)

Double Buffer data between asynchronous subsystems/blocks in a model.

Parameters

Sample time

-1


Block Parameters: Write Side

Asynchronous Buffer Block (mask) (link)

Double Buffer data between asynchronous subsystems/blocks in a model.

Parameters

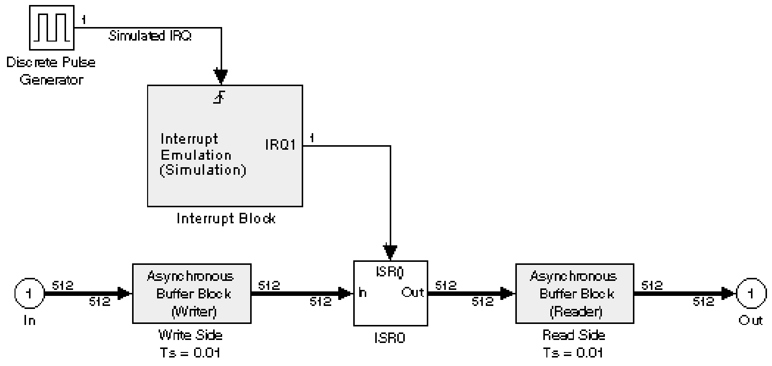
Sample time

-1

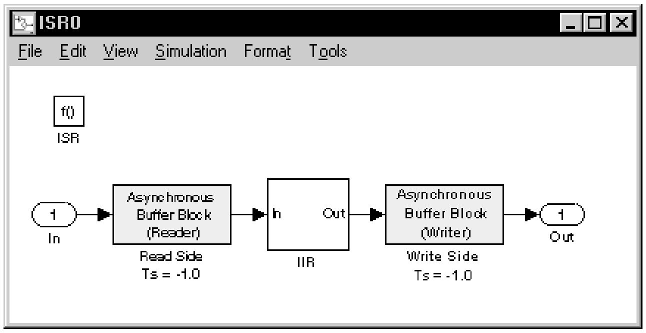
Both blocks require the Sample Time parameter. The sample time should be set to -1 inside a function call and to the desired time otherwise.

Asynchronous Buffer Block Example

This example shows how you might use the Asynchronous Buffer block to control the data flow in an interrupt service routine.



The `ISR0` subsystem block, which is configured as a function-call subsystem, contains another set of Asynchronous Buffer blocks.


ISR0
File Edit View Simulation Format Tools
f()
ISR
1
In
Asynchronous Buffer Block (Reader)
Read Side Ts = -1.0
In Out
IIR
Asynchronous Buffer Block (Writer)
Write Side Ts = -1.0
1
Out

Rate Transition Block

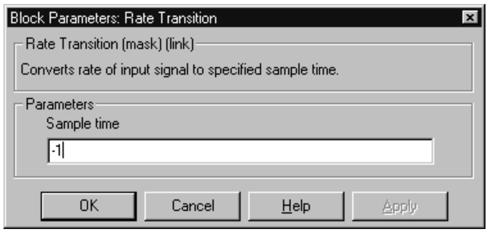
The VxWorks Rate Transition block provides a sample time for blocks connected to an asynchronous function-call subsystem when double buffering is not required. There are two options for connecting I/O to an asynchronous function-call subsystem:

- Use the Rate Transition block, or some other block that requires a sample time to be set, at the input or output of the asynchronous function-call subsystem. This will cause blocks up- or downstream from it, which would otherwise inherit from the function-call subsystem, to use the sample time specified. Note that if the signal width is greater than 1, data consistency is not guaranteed, which may or may not an issue. See next option.

  The Rate Transition block does not introduce any system delay. It only specifies the sample time of the downstream blocks. It also informs Simlink to allow a non-buffered asynchronous connection. This block is typically used for scalar signals that do not require double buffering.
- Use the Asynchronous Buffer block pair. This not only will set the sample time of the blocks up or downstream that would otherwise inherit from the function-call subsystem, and also guarantees consistency of the data on the signal. See the Asynchronous Buffer block for more information on data consistency.

Rate Transition Block Parameters

This picture shows the VxWorks Rate Transition block's dialog box.



Block Parameters: Rate Transition

Rate Transition (mask) (link)

Converts rate of input signal to specified sample time.

Parameters

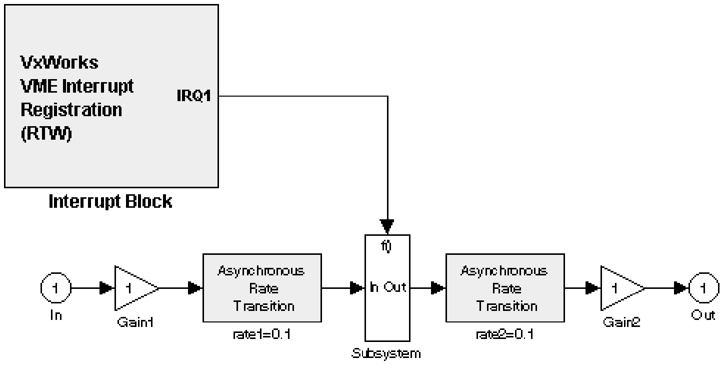
Sample time

-1

The Sample time parameter sets the sample time to the desired rate.

Rate Transition Block Example

This picture shows a sample application of the Rate Transition block in an ISR.



In this example, the Rate Transition block on the input to the function-call subsystem causes both the In and Gain1 blocks to run at the 0.1 second rate. The Rate Transition block on the output of the function-call subsystem causes both the Gain2 and Out blocks to run at the 0.2 second rate. Using this scheme informs Simlink to allow non-buffered connections to an asynchronous function-call subsystem.

Creating a Customized Asynchronous Library

You can use the Real-Time Workshop's VxWorks asynchronous blocks as templates that provide a starting point for creating your own asynchronous blocks. Templates are provided for these blocks:

- Asynchronous Buffer block
- Asynchronous Interrupt block
- Rate Transition block
- Task Synchronization block

You can customize each of these blocks by implementing a set of modifications to files associated with each template. These files are:

- The block's underlying S-function C MEX-file
- The block's mask and the associated mask M-file
- The TLC files that control code generation of the block

At a minimum, you must rename the system calls generated by the TLC files to the correct names for the new real-time operating system (RTOS) and supply the correct arguments for each file. There is a collection of files that you must copy (and rename) from *matlabroot*`/rtw/c/tornado/devices` into a new directory, for example, *matlabroot*`/rtw/c/`*my_os*`/devices`. These files are:

- Asynchronous Buffer block — `vxdbuffer.tlc`, `vxdbuffer.c`
- Asynchronous Interrupt block — `vxinterrupt.tlc`, `vxinterrupt.c`, `vxintbuild.m`
- O/S include file — `vxlib.tlc`
- Task Synchronization block — `vxtask.tlc`, `vxtask.c`

Real-Time Workshop Ada Coder

Introduction

This chapter presents an introduction to the Real-Time Workshop Ada Coder. It compares and contrasts the Real-Time Workshop Ada Coder with the Real-Time Workshop, shows you how to use the product by presenting an example, and concludes with a discussion of code validation.

Note The Real-Time Workshop Ada Coder is a separate product from the Real-Time Workshop.

Like the Real-Time Workshop, the Real-Time Workshop Ada Coder provides a real-time development environment that features:

- A rapid and direct path from system design to hardware implementation
- Seamless integration with MATLAB and Simulink
- A simple, easy-to-use interface
- An open and extensible architecture

The package includes application modules that allow you to build complete programs targeting a wide variety of environments. Program building is fully automated. Automatic program building provides a standard means to create programs for real-time applications. This chapter contains examples of automatic program building on DOS and UNIX platforms.

The Real-Time Workshop Ada Coder is an automatic Ada language code generator. It produces Ada83 or Ada95 code directly from Simulink models and automatically builds programs that can be run in real time in a variety of environments. The Real-Time Workshop Ada Coder is an extension of the Real-Time Workshop.

With the Real-Time Workshop Ada Coder, you can run your Simulink model in real time on a remote processor. You can run accelerated, stand-alone simulations on your host machine or on an external computer.

Real-Time Workshop Ada Coder Applications

Like the Real-Time Workshop, the Real-Time Workshop Ada Coder supports a variety of real-time applications:

- Real-Time Control — You can design your control system using MATLAB and Simulink and generate Ada code from your block diagram model. You can then compile and download the Ada code directly to your target hardware.
- Hardware-in-the-Loop Simulation — You can use Simulink to model real-life measurement and actuation signals. You can use the code generated from the model on special-purpose hardware to provide a real-time representation of the physical system. Applications include control system validation, training simulation, and fatigue testing using simulated load variations.

Supported Compilers

The generated code will work with any validated Ada83 or Ada95 compiler. Real-Time Workshop provides an example target that uses the GNAT Ada95 compiler. You can download a free version of this compiler from the GNAT ftp site (`ftp://cs.nyu.edu/pub/gnat`). You can also purchase a professional version from Ada Core Technologies (`www.gnat.com`).

Supported Targets

The Real-Time Workshop Ada Coder supports the following targets:

- Ada Simulation Target — Useful for validating generated code. This does not use Ada tasking primitives.
- Ada Multitasking Real-Time Target — Useful as a starting point for targeting real-time systems. This uses Ada tasking primitives.
- Ada83 target — The generated Ada83 code does not support data logging, but is suitable for generating embedded code for use with legacy Ada83 compilers. The Ada83 target is available from the System Target File Browser by selecting the system target file `rt_ada83.tlc` (Ada83 Target for GNAT). The accompanying template makefile `gnat83.tmf` uses the `-gnat83` switch in the GNAT Ada95 compiler.

You can also add your own target by creating a system target file, make process, and run-time interface files along with any device drivers using inlined Ada or C S-functions.

The Generated Code

The generated code (i.e., the model code) is highly optimized, fully commented, and can be generated from any discrete-time Simulink model — linear or nonlinear.

All Simulink blocks are automatically converted to code, with the exception of:

- MATLAB function blocks
- Any continuous sample time blocks
- S-functions that are not inlined using the Target Language Compiler

Types of Output

The Real-Time Workshop Ada Coder's interface supports two forms of output:

- Ada code – Generate code that contains system equations and initialization functions for the Simulink model. You can use this code in real-time applications.
- A real-time program – Transform the generated code into a real-time program suitable for use with dedicated real-time hardware. The resulting code is designed to interface with an external clock source and hence runs at a fixed, user-specified sample rate.

Supported Blocks

See "Supported Blocks" on page 16–21 for a complete list of the Simulink blocks supported by the Real-Time Workshop Ada Coder.

Restrictions

The Real-Time Workshop Ada Coder has the same constraints imposed upon it as the Real-Time Workshop Embedded Coder target. The code generator does not produce code that solves algebraic loops, and Simulink blocks that are dependent on absolute time can be used only if the program is not intended to run for an indefinite period of time.

There are additional constraints for the Ada code generation. The Real-Time Workshop Ada Coder does not provide:

- Nonreal-time variable step integration models
- Continuous-time integration
- Since Ada does not support implicit upcasting of data types, all numerical operations in your model must be of homogeneous data types for Ada code generation. You can, however, perform explicit upcasting using the Data Type Conversion block in Simulink.
- You must inline all Ada S-functions with a corresponding TLC file (see the *Target Language Compiler Reference Guide* for more information about inlining S-functions)
- External mode is not supported with the Real-Time Workshop Ada Coder.

Getting Started

This section illustrates, through a simple example, how to transform a Simulink model into a stand-alone executable program. This program runs independently of Simulink, allowing accelerated execution on the development host or a different target computer.

Generating Ada code from a Simulink model is very similar to generating C code. Begin by typing

```
countersdemo
```

at the MATLAB prompt. This block diagram appears.

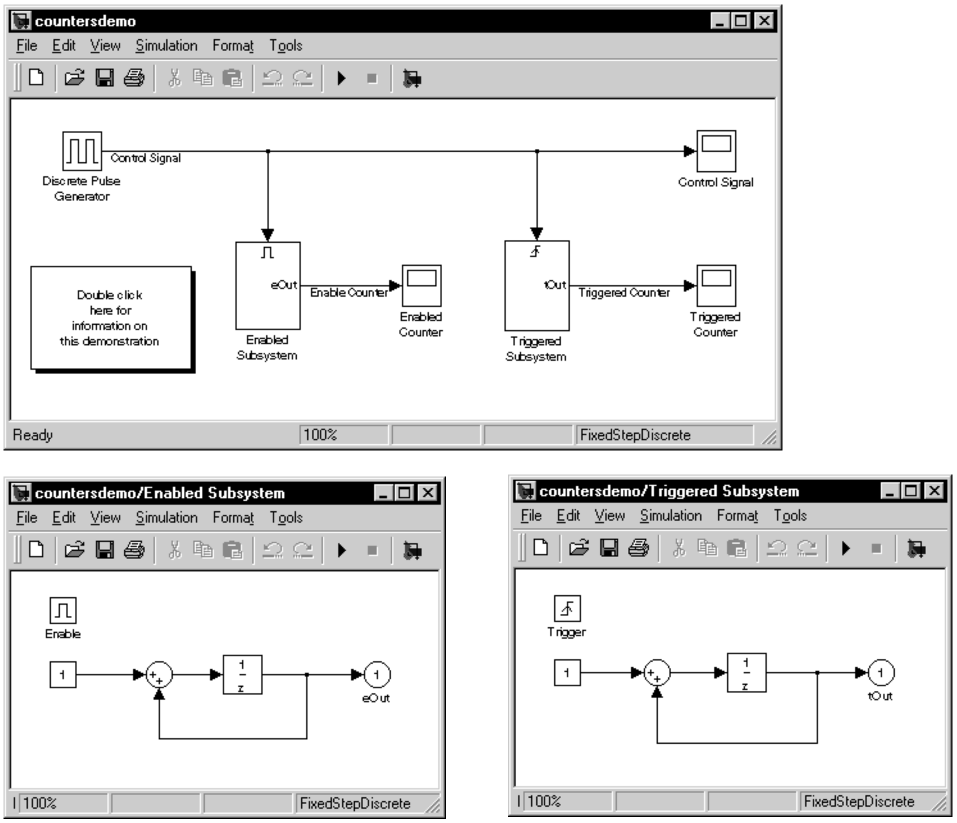


Figure 16-1: Counter Demonstration with Subsystems Open

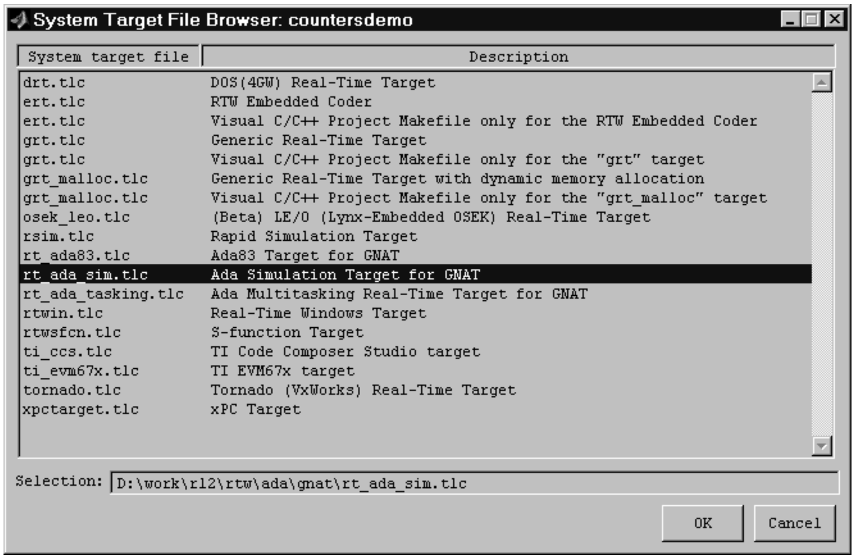
Setting Options for Ada Code Generation

You must specify the correct options before you generate Ada code from this model. These are the steps:

1. Select **Parameters** under the **Simulation** menu. This opens the **Simulation Parameters** dialog box.
2. On the Solver page, set the **Solver options** to **Fixed-step discrete (no continuous states)**
3. Select the Real-Time Workshop page. In the **Category** menu, select **Target configuration**.
4. Click the **Browse** button. This opens the S**ystem Target File Browser**.

5 Select **Ada Simulation Target for GNAT** and click **OK**. This automatically sets the correct **System target file**, **Template makefile**, and **Make command** fields for Ada code generation.

   Figure 16-2 shows the System Target File Browser with the correct selection for Ada code generation.



System target file	Description
drt.tlc	DOS(4GW) Real-Time Target
ert.tlc	RTW Embedded Coder
ert.tlc	Visual C/C++ Project Makefile only for the RTW Embedded Coder
grt.tlc	Generic Real-Time Target
grt.tlc	Visual C/C++ Project Makefile only for the "grt" target
grt_malloc.tlc	Generic Real-Time Target with dynamic memory allocation
grt_malloc.tlc	Visual C/C++ Project Makefile only for the "grt_malloc" target
osek_leo.tlc	(Beta) LE/O (Lynx-Embedded OSEK) Real-Time Target
rsim.tlc	Rapid Simulation Target
rt_ada83.tlc	Ada83 Target for GNAT
rt_ada_sim.tlc	Ada Simulation Target for GNAT
rt_ada_tasking.tlc	Ada Multitasking Real-Time Target for GNAT
rtwin.tlc	Real-Time Windows Target
rtwsfcn.tlc	S-function Target
ti_ccs.tlc	TI Code Composer Studio target
ti_evm67x.tlc	TI EVM67x target
tornado.tlc	Tornado (VxWorks) Real-Time Target
xpctarget.tlc	xPC Target

Selection: D:\work\r12\rtw\ada\gnat\rt_ada_sim.tlc

Figure 16-2: The System Target File Browser

Alternatively, you can specify the settings on the Real-Time Workshop page manually by following these steps:

1 Select **Options** under the **Real-Time Workshop** submenu of the **Tools** menu. This opens the Real-Time Workshop page of the **Simulation Parameters** dialog box.

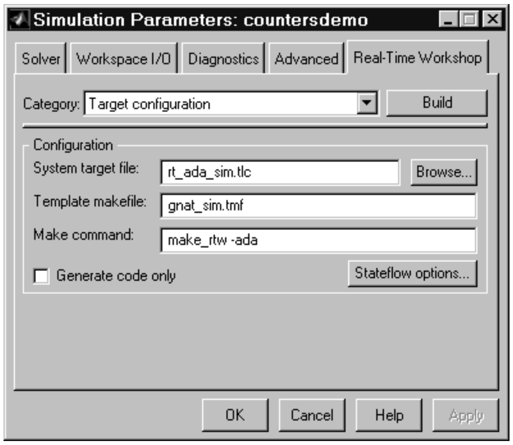
2 In the **Category** menu, select **Target configuration**.

3 Specify `rt_ada_sim.tlc` as the **System target file**.

4 Specify `gnat_sim.tmf` as the **Template makefile**.

5 Specify `make_rtw -ada` as the **Make command**.

Figure 16-3 shows the Real-Time Workshop page with the correct settings.



Category: Target configuration

System target file: rt_ada_sim.tlc

Template makefile: gnat_sim.tmf

Make command: make_rtw -ada

☐ Generate code only

Figure 16-3: Target Configuration Settings in the Real-Time Workshop Page

In addition, you can use the `make` command to pass compiler switches to the code compilation phase. For example, if you want to compile with debugging symbols, add a `-g` after the `-ada` switch in the **Make command** field (there must be a space between each switch). This switch is applied on a model basis; for more permanent changes, see "Configuring the Template Makefile" on page 14-13.

Generating Ada Code

To generate Ada code and build an Ada executable, open the Real-Time Workshop page. In the **Category** menu, select **Target configuration**. Click the **Build** button.

Alternatively, select **Build Model** under the **Real-Time Workshop** submenu of the **Tools** menu.

Generated Files

The Real-Time Workshop Ada Coder creates output files in two directories during the build process:

- The working directory

  If an executable is created, it is written to your working directory. The executable is named *model*`.exe` (on PC) or *model* (on UNIX).·
- The build directory

  The build process creates a subdirectory, called the build directory, within your working directory. The build directory name is *model*`_ada`*XX*`_rtw`, where *model* is the name of the source model, and `ada`*XX* is either `ada83` or `ada95`, depending on the selected target. The build directory stores generated source code and all other files (other than the executable) created during the build process.

This table lists the Ada files generated by the Real-Time Workshop Ada Coder from the counter demonstration (`countersdemo`).

Table 16-1: Ada Files Generated by the Real-Time Workshop Ada Coder

Filename	Description
`countersdemo.adb`	Package body with the implementation details of the model.
`countersdemo.ads`	Package specification that defines the callable procedures of the model and any external inputs and outputs to the model.
`countersdemo_types.ads`	Package specification of data types used by the model. The Real-Time Ada Coder derives the data types from the block name and signal width.
`register.ads`	Package specification that defines model rate information and renames program entry points in `countersdemo.ads`.

Table 16-1: Ada Files Generated by the Real-Time Workshop Ada Coder (Continued)

Filename	Description
`register2.ads`	Package specification that contains the subset of `register.ads` required for elimination of circular compilation dependencies.
`rt_engine-rto_data.ads`	Package specification that contains the timing information for executing the model encapsulated in the real-time object.

Models with S-Functions

Real-Time Workshop Ada Coder does not currently support non-inlined S-functions. For Real-Time Workshop code generation purposes, you can create a wrapper S-function that calls an Ada S-function.

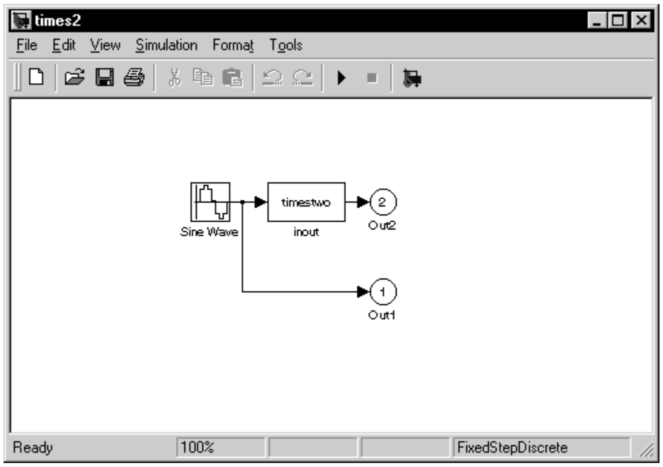
It is possible, however, to generate Ada code for models with inlined Ada or C S-functions. To do so, you must create a TLC file that incorporates the algorithm from your S-function.

The following example shows how to write a TLC file to inline a simple Ada S-function. For more information on writing TLC files, see the *Target Language Compiler Reference Guide*.

Create model `times2` using these blocks:

- Sine Wave (sample time = 0.1)
- `timestwo` Ada S-function (provided in the *matlabroot*`/toolbox/simulink/blocks/tlc_ada` directory) with no parameters set (i.e., leave the **Parameters** field blank)
- Two Outport blocks

Your model should look like this picture.



The times2 model contains a simple S-function, called timestwo, that takes the input sine wave signal and doubles its amplitude. The TLC file corresponding to the S-function is shown below.

```
%% Copyright (c) 1990-2000 by The MathWorks, Inc.
%%
%% Abstract:
%%      TLC file for timestwo.c used in Real-Time Workshop
%%      S-Function test.

%implements "timestwo" "Ada"

%% Function: Outputs
==========================================================
%function Outputs(block, system) Output
  -- %<Type> Block: %<Name>
  -- Multiply input by two
  %<LibBlockOutputSignal(0, "", "", 0)> := ...
    %<LibBlockInputSignal(0, "", "", 0) > * 2.0;

%endfunction

%% [EOF] timestwo.tlc
```

The key line in this TLC file is the assignment of two times the `LibBlockInputSignal` to the `LibBlockOutputSignal`. This line directs the Target Language Compiler to place the algorithm directly into the generated Ada code.

This TLC file is located in *matlabroot*`/toolbox/simulink/blocks/tlc_ada/timestwo.tlc`.

Generating the Ada Code

The build process is similar to the case where your model does not contain any S-functions. The only additional requirement for generating Ada code for models containing S-functions is to provide a TLC file with the same name as the S-function. Otherwise, the build procedure is exactly the same.

Configuring the Template Makefile

Template makefiles specify the compiler, link, and make directives native to the target computer's operating system and compiler you are using. Two examples of template makefiles are provided in the directory `matlab/rtw/ada/gnat`. File `gnat_sim.tmf` is the template makefile that builds that real-time Ada simulation program. The automatic build process expands the macros defined at the top of the `.tmf` file to create the call to `gnatmake`. `gnatmake` then compiles and links the program.

There are two ways to make permanent changes to the template makefile if you need to make modifications either to target a different Ada95 compiler or to make minor adjustments to the `gnat` make directive:

- Copy the template makefile into the directory where the model is located.
- Copy the template makefile to a unique name in `matlab/rtw/ada/gnat` and make modifications to the new file.

Data Logging

You can use the Ada real-time simulation target (`rt_ada_sim`) to perform the same data logging as a Simulink simulation. To enable MAT-file logging, select Ada-specific code generation options item in the **Category** menu of the Real-Time Workshop page. Then select the **MAT-file logging** option. When this option is selected, the Ada program executes for the duration specified by the **Stop time** field on the Solver page of the **Simulation Parameters** dialog box.

When the Ada Coder finishes its run, a *model*`.mat` file is created that contains all workspace variables that would have been created by running a Simulink simulation. The names of these workspace variables are the same as the names that would have been created by Simulink, except that an `rt_` prefix is attached. See "Workspace I/O Options and Data Logging" in Chapter 3 for more information about data logging.

Note If you do not select MAT-file logging, the stop time is ignored and the Ada program runs without stopping.

Generating Block Comments

When the **Insert block descriptions in code** option is selected, comments are inserted into the code generated for any blocks that have text in their **Description** fields. To generate block comments:

1 Right-click on the block you want to comment. Select **Block Properties** from the context menu. The **Block Properties** dialog box opens.

2 Type the comment into the **Description** field.

3 Select the **Insert block descriptions in code** option in the Ada-specific code generation options category of the Real-Time Workshop page.

Note For virtual blocks or blocks that have been removed due to block reduction optimizations, no comments are generated.

Application Modules Required for the Real-Time Program

Building the real-time program requires a number of support files in addition to the generated code. These support files contain:

- A main program
- Code to drive execution of the model code
- Code to carry out data logging

The makefile automatically compiles and links these source modules. This diagram shows the modules used to build the `countersdemo` example.

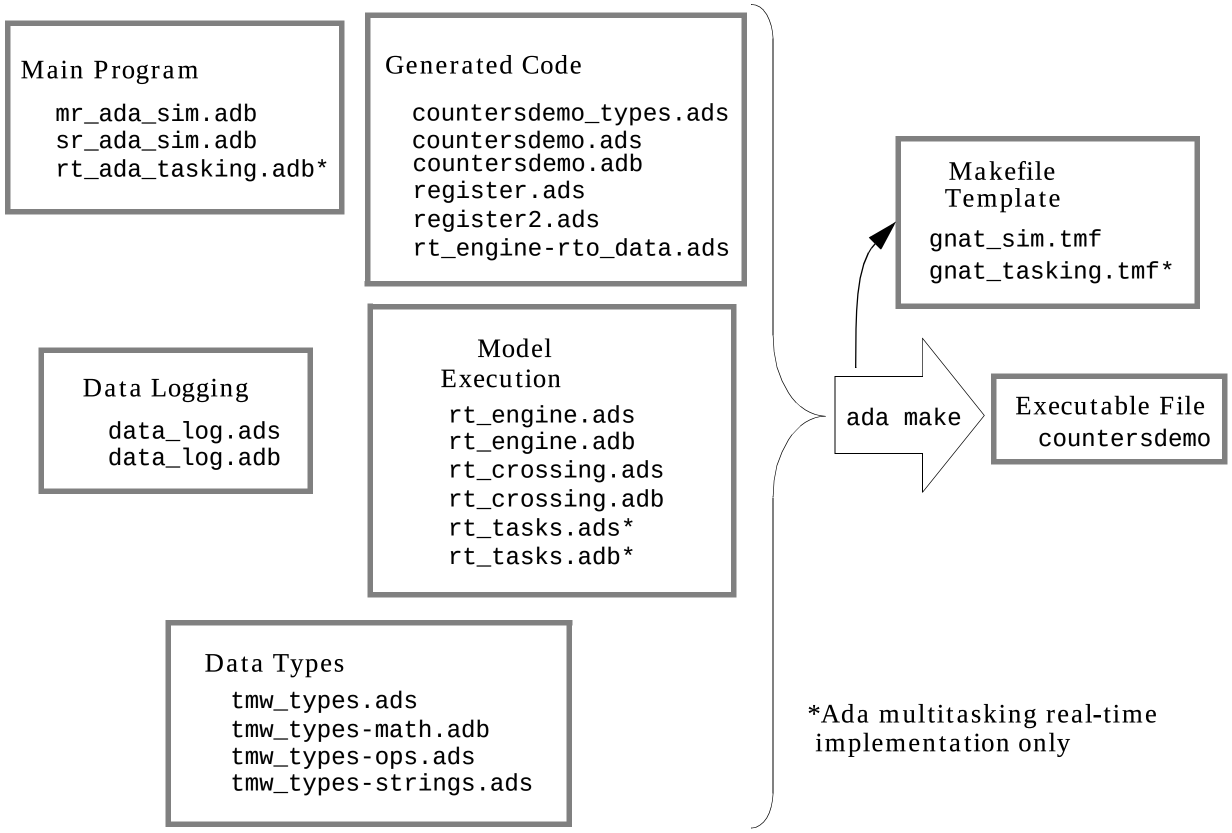


Figure 16-4: Source Modules Used to Build the countersdemo Program

Configuring and Interfacing Parameters and Signals

Model Parameter Configuration

The **Model Parameter Configuration** dialog provides a mechanism for interfacing your model's block parameters to code that you have written. This is useful in situations where you want your hand-written code to change parameter values while the generated program executes.

Read "Parameters: Storage, Interfacing, and Tuning" on page 3-51 to learn about general parameter storage concepts and the **Model Parameter Configuration** dialog.

Using the **Model Parameter Configuration** dialog does not differ greatly between between Ada and other target languages such as C. However, you should note the following Ada-specific differences:

- The **Storage Type Qualifier** field is only used when specifying the package specification to fully qualify the variable name for the `ImportedExtern` option. This field is ignored in all other cases.
- `ImportedExtern` variables are assumed to be declared in the package specification entered in the **Storage Type Qualifier** field. The generated code accesses this variable as `Your_Package.Your_variable`.
- `ImportedExternPointer` storage class is not permitted in Ada.

Signal Properties

The Simulink **Signal Properties** dialog lets you interface selected signals to externally written code. In this way, your hand-written code has access to such signals for monitoring or other purposes.

Read "Signals: Storage, Optimization, and Interfacing" on page 3-65 to learn about general signal storage concepts and the **Signal Properties** dialog. Then note the Ada-specific differences described below.

The Real-Time Workshop Ada Coder has logic for supporting Simulink signal labels. This logic automatically maps signal labels to Simulink blocks based on the block name, signal name, and connectivity. You can override the default behavior and either specify an external declaration for the signal name or direct the Real-Time Workshop Ada Coder to declare a unique declaration of the signal that is visible in the generated model package specification. The heuristics are implemented on a signal basis as specified by the **Signal name**, **RTW storage class**, and **RTW storage type qualifier** (externally declared signals only).

The options relevant to the Real-Time Workshop Ada Coder are:

- **SimulinkGlobal (Test Point)**: Clicking this check box directs the Real-Time Workshop Ada Coder to place the signal in a unique global memory location (`rtB` structure). This is useful for testing purposes since it eliminates the possibility of overwriting the signal data. Note that selecting this option forces the **RTW storage class** to be `Auto`.
- **RTW storage class** : The storage class options are:
  - `Auto`: directs the Real-Time Workshop to store the signal in a persistent data structure. Specifically, an element called `Signal_Name` is declared in the `External_Inputs` structure defined in the `Model_Types` package specification. The generated code accesses this signal as `RT_U.Signal_Name`.
  - `ExportedGlobal`: Declares the signal as a global variable that can be accessed from outside the generated code. The signal is declared in the model package specification but not in the `External_Inputs` structure. The generated code accesses this signal as `Signal_Name`. The signal will be globally visible as `Model.Signal_Name`.
  - `ImportedExtern`: The signal is assumed to be declared in the package specification entered in the **RTW storage type qualifier** field. The generated code accesses this signal as `Your_Package.Signal_Name`.
  - `ImportedExternPointer`: This is not permitted in Ada.
- **RTW storage type qualifier** : This is only used when specifying the package specification to qualify fully the signal name for the `Imported Extern` option. This field is ignored in all other cases.

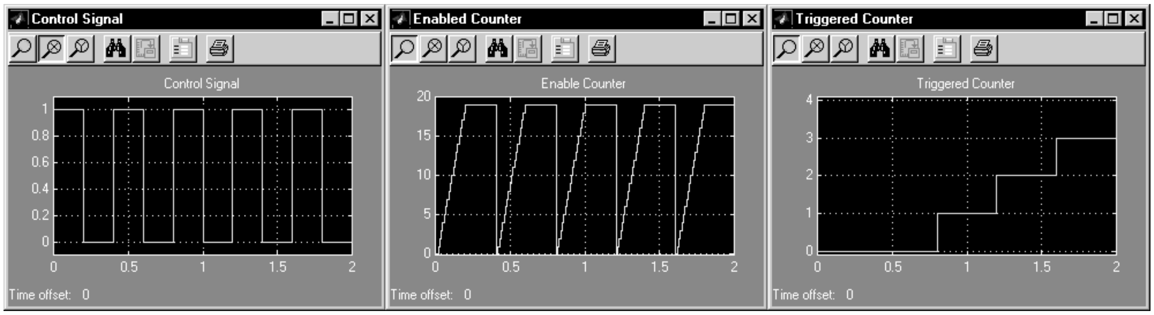
These cases are useful if you want to link Real-Time Workshop Ada Coder generated code to other Ada code (i.e., code that the Real-Time Workshop Ada Coder did not generate).

Code Validation

After completing the build process, the stand-alone version of the countersdemo model is ready for comparison with the Simulink model. The data logging options selected with the Workspace I/O page of the **Simulation Parameters** dialog box cause the program to save the control signal, enabled counter, triggered counter, and simulation time. You can now use MATLAB to produce plots of the same data that you see on the three Simulink scopes.

In both the Simulink and the stand-alone executable version of the countersdemo model, the control input is simulated with a discrete-pulse generator producing a 10 Hz, fifty percent duty cycle waveform.

Open the control signal, enabled counter, and triggered counter scopes. Running the Simulink simulation from T=0 to T=2 produces these outputs.



Type who at the MATLAB prompt to view the variable names from Simulink simulation.

```
who

Your variables are:
Enable_Signal       Triggered_Counter
Enabled_Counter     tout
```

Now run the stand-alone program from MATLAB.

```
!countersdemo
```

The “!” character passes the command that follows it to the operating system. This command, therefore, runs the stand-alone version of countersdemo (not the M-file).

To obtain the data from the stand-alone program, load the file countersdemo.mat.

```
load countersdemo
```

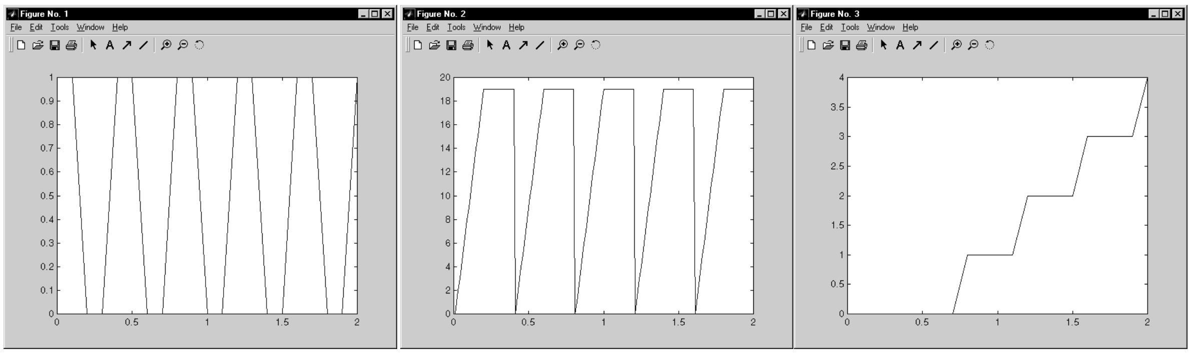
Then look at the workspace variables.

```
who
Your variables are:
Enable_Signal                 rt_Triggered_Counter
Enabled_Counter               rt_tout
Triggered_Counter             tout
rt_Enable_Signal
rt_Enabled_Counter
```

The stand-alone Ada program prepends rt_ to the logged variable names to distinguish them from the variables Simulink logged.

You can now use MATLAB to plot the three workspace variables as a function of time.

```
plot(rt_Enable_Signal(:,1),rt_Enable_Signal(:,2))
figure
plot(rt_Enabled_Counter(:,1),rt_Enabled_Counter(:,2))
figure
plot(rt_Triggered_Counter(:,1),rt_Triggered_Counter(:,2))
```



Analyzing Data with MATLAB

Points to consider when data logging:

- Ada95 code supports all data logging formats (matrix, structure, and structure/time).
- Ada83 code does not support data logging.
- To Workspace blocks log data at the frequency of the driving block and do not log time.
- Scope blocks log data at the frequency of the driving block and log time in the first column of the matrix.
- Root Outport blocks are updated at the frequency of the driving block but are logged at the base rate of the model.

Supported Blocks

The Real-Time Workshop Ada Coder supports the following Simulink blocks.

Discrete Blocks	
Discrete-Time Integrator	Discrete Zero-Pole
Discrete Filter	Unit Delay
Discrete State-Space	Zero-Order Hold
Discrete Transfer Fcn	

Functions & Tables	
Direct Look-Up Table (n-D)	Look-Up Table (2-D)
Fcn	Look-Up Table (n-D)
Interpolation (n-D) Using PreLook-Up Index Search	PreLook-Up Index Search
Look-Up Table	S-Function — Only Target Language Compiler inlined S-functions are supported

Math Blocks	
Abs	MinMax
Bitwise Logical Operator	Product — matrix multiplication and element-wise multiplication and division are supported. Matrix division is *not* supported.
Combinatorial Logic	Real-Imag to Complex
Complex to Magnitude-Angle	Relational Operator
Complex to Real-Imag	Rounding Function

Math Blocks (Continued)	
Dot Product	Sign
Gain (inluding matrix/ element-wise)	Slider Gain
Logic Operator	Sum
Magnitude-Angle to Complex	Trigonometric Function
Math Function	

Nonlinear Blocks	
Backlash	Quantizer
Coulomb & Viscous Friction	Relay
DeadZone	Saturation
Manual Switch (must Break Library Link and use discrete sample time)	Switch
Multiport Switch	

Signals & Systems Blocks	
Bus Selector	Initial Condition (IC)
Configurable Subsystem	Inport
DataStore Memory	Matrix Concatenation
DataStore Read	Merge
DataStore Write	ModelInfo
Data Type Conversion	Outport
Demux	Probe

Signals & Systems Blocks (Continued)	
Enable	Reshape
From	Selector
Goto Tag Visibility	Subsystem
Goto	Terminator
Ground	Trigger Width
Hit Crossing	

Sinks	
Display — no code is generated for this block	To File
Scope	To Workspace
Stop Simulation	

Sources	
Band-Limited White Noise	Ramp — You must break the library link and replace the clock with a discrete clock and manually set the sample time step to match the discrete clock.
Chirp Signal — (you must break the library link and use a discrete clock)	Random Number
Constant	Sine Wave

Sources (Continued)	
Digital Clock	Repeating Sequence — You must break the library link and replace the clock with a discrete clock and manually set the sample time step to match the discrete clock.
Discrete Pulse Generator	Step
From File	Uniform Random Number

Note All element-wise operation blocks are suported by the Real-Time Workshop Ada Coder.

Targeting Real-Time Systems

Introduction

The target configurations bundled with the Real-Time Workshop are suitable for many different applications and development environments. Third-party targets provide additional versatility. However, a number of users find that they require a custom target configuration.You may want to implement a custom target configuration for any of the following reasons:

- To support custom hardware and incorporate custom device driver blocks into your models.
- To customize a bundled target configuration — such as the generic real-time (GRT) or Real-Time Workshop Embedded Coder targets — to your needs.
- To configure the build process for a special compiler (such as a compiler for an embedded microcontroller or DSP board).

As part of your custom target implementation, you may also need to:

- Interface generated model code with existing supervisory or supporting code that calls the generated code.
- Interface signals and parameters within generated code to your own code.
- Combine code generated from multiple models into a single system.
- Implement external mode communication via your own low-level protocol layer.

The following sections provide the information necessary to accomplish these tasks:

- "Components of a Custom Target Configuration" on page 17-4 gives an overview of the code and control files that make up a custom target configuration.
- "Tutorial: Creating a Custom Target Configuration" on page 17–9 is a hands-on exercise in building a custom rapid prototyping target.
- "Customizing the Build Process" on page 17–16 provides information on the structure of system target files ("System Target File Structure" on page 17–16) and template makefiles ("Template Makefiles" on page 17–25).
- "Adding a Custom Target to the System Target File Browser" on page 17–24 shows you how to make your custom target configuration available to users via the System Target File Browser.

- “Creating Device Drivers” on page 17–34 discusses the implementation of device drivers as S-Function blocks, covering both inlined and noninlined drivers.
- “Interfacing Parameters and Signals” on page 17–65 contains guidelines for use of the Real-Time Workshop signal monitoring and parameter tuning APIs.
- “Creating an External Mode Communication Channel” on page 17–73 provides information you will need to support external mode on your custom target, using your own low-level communications layer.
- “Combining Multiple Models” on page 17–82 discusses strategies for combining several models (or several instances of the same model) into a single executable.

Components of a Custom Target Configuration

The components of a custom target configuration are:

- Code to supervise and support execution of generated model code
- Control files:
  - A system target file to control the code generation process
  - A template makefile to build the real-time executable

This section summarizes key concepts and terminology you will need to know to begin developing each component. References to more detailed information sources are provided, in case any of these topics are unfamiliar to you.

Code Components

A Real-Time Workshop program containing code generated from a Simulink model consists of a number of code modules and data structures. These fall into two categories.

Application Components

Application components are those which are specific to a particular model; they implement the functions represented by the blocks in the model. Application components are not specific to the target. Application components include:

- Modules generated from the model
- User-written blocks (S-functions)
- Parameters of the model that are visible, and can be interfaced to, external code

Run-Time Interface Components

A number of code modules and data structures, referred to collectively as the *run-time interface*, are responsible for managing and supporting the execution of the generated program. The run-time interface modules are not automatically generated. To develop a custom target, you must implement certain parts of the run-time interface. Table 17-1 summarizes the run-time interface components.

Table 17-1: Run-Time Interface Components

User Provides:	Real-Time Workshop Provides:
Customized main program	Generic main program
Timer interrupt handler to run model	Execution engine and integration solver (called by timer interrupt handler)
Other interrupt handlers	Example interrupt handlers (Asynchronous Interrupt Blocks)
Device drivers	Example device drivers
Data logging and signal monitoring user interface	Data logging, parameter tuning, signal monitoring, and external mode support

The components of the run-time interface vary, depending upon whether the target is an embedded system or a rapid prototyping environment.

User-Written Run-Time Interface Code

Most of the run-time interface is provided by Real-Time Workshop. You must implement the following elements:

- A timer *interrupt service routine* (ISR). The timer runs at the program's base sample rate. The timer ISR is responsible for operations that must be completed within a single clock period, such as computing the current output sample.The timer ISR usually calls the Real-Time Workshop supplied function, `rt_OneStep`.
- The *main program*. Your main program initializes the blocks in the model, installs the ISR, and executes a background task or loop. The timer periodically interrupts the main loop. If the main program is designed to run for a finite amount of time, it is also responsible for cleanup operations - such as memory deallocation and masking the timer interrupt - before terminating the program.
- *Device drivers* to communicate with your target hardware.

Run-Time Interface for Rapid Prototyping

The run-time interface for a rapid prototyping target includes:

- Supervisory logic
  - The main program
  - Execution engine and integration solver
- Supporting logic
  - I/O drivers
  - Code to handle timing, and interrupts
- Monitoring, tuning, and debugging support
  - Data logging code
  - Signal monitoring
  - Real-time parameter tuning
  - External mode communications

The structure of the rapid prototyping run-time interface, and the execution of rapid prototyping code, are detailed in Chapter 6, “Program Architecture” and Chapter 7, “Models with Multiple Sample Rates.”

Development of a custom rapid prototyping target generally begins with customization of one of the generic main programs, `grt_main.c` or `grt_malloc_main.c`. As described in “User-Written Run-Time Interface Code” above, you must modify the main program for real-time interrupt-driven execution. You must also supply device drivers (optionally inlined).

Run-Time Interface for Embedded Targets

The run-time interface for an embedded (production) target includes:

- Supervisory logic
  - The main program
  - Execution engine and integration solver
- Supporting logic
  - I/O drivers
  - Code to handle timing, and interrupts

- Monitoring and debugging support
  - Data logging code
  - Access to tunable parameters and external signals

Development of a custom embedded target generally begins with customization of the Real-Time Workshop Embedded Coder main program, `ert_main.c`. Chapter 9, "Real-Time Workshop Embedded Coder" details the structure of the Real-Time Workshop Embedded Coder run-time interface and the execution of Real-Time Workshop Embedded Coder code, and provides guidelines for customizing `ert_main.c`.

Control Files

System Target Files

The Target Language Compiler (TLC) generates target-specific C or Ada code from an intermediate description of your Simulink block diagram (*model*`.rtw`). The Target Language Compiler reads *model*`.rtw` and executes a program consisting of several target files (`.tlc` files.) The output of this process is a number of source files, which are fed to your development system's make utility.

The system target file controls the code generation process. You will need to create a customized system target file to set code generation parameters for your target. We recommend that you copy, rename, and modify one of the standard system target files:

- The generic real-time (GRT) target file, *matlabroot*`/rtw/c/grt/grt.tlc`, for rapid prototyping targets
- The Real-Time Workshop Embedded Coder target file, *matlabroot*`/rtw/c/ert/ert.tlc`, for embedded (production) targets

Chapter 2, "Technical Overview"2 and Chapter 3, "Code Generation and the Build Process" describe the role of the system target file in the code generation and build process. Guidelines for creating a custom system target file are given in "Customizing the Build Process" on page 17-16.

Template Makefiles

A template makefile (`.tmf` file) provides information about your model and your development system. Real-Time Workshop uses this information to create an appropriate makefile (`.mk` file) to build an executable program. Real-Time Workshop provides a large number of template makefiles suitable for different types of targets and development systems. The standard template makefiles are described in “Template Makefiles and Make Options” on page 3–102.

If one of the standard template makefiles meets your requirements, you can simply copy and rename it in accordance with the conventions of your project. If you need to make more extensive modifications, see “Template Makefiles” on page 17-25 for a full description of the structure of template makefiles.

Tutorial: Creating a Custom Target Configuration

This tutorial walks through the task of creating a skeletal rapid prototyping target. This exercise illustrates several tasks that are usually required when creating a custom target:

- Incorporating a noninlined S-function into a model for use in simulation.
- Inlining the S-function in the generated code, using a corresponding TLC file.

  In a real-world application, you would incorporate inlined and noninlined device driver S-functions into the model and the generated code. In this tutorial, we inline a simple S-function that multiplies its input by two.
- Making minor modifications to a standard system target file and template makefile.
- Generating code from the model by invoking your customized system target file and template makefile.

You can use this process as a starting point for your own projects.

This example uses the LCC compiler under Windows. LCC is distributed with Real-Time Workshop. If you use a different compiler, you can set up LCC temporarily as your default compiler by typing the MATLAB command

```
mex -setup
```

A command prompt window will open; follow the prompts and select LCC.

Note On UNIX systems, make sure that you have a C compiler installed. You can then do this exercise substituting appropriate UNIX directory syntax.

In this example, the code is generated from `targetModel.mdl`, a very simple fixed-step model (see Figure 17-1). The resultant program behaves exactly as if it had been built for the generic real-time target.

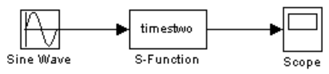


Figure 17-1: targetModel.mdl

The S-Function block will use the source code from the `timestwo` example. See the *Writing S-Functions* manual for a complete discussion of this S-function. The *Target Language Compiler Reference Guide* discusses `timestwo.tlc`, the inlined version of `timestwo`.

To create the skeletal target system:

1. Create a directory to store your C source files and `.tlc` and `.tmf` files. We refer to this directory as `d:/work/mytarget`.

2. Add `d:/work/mytarget` to your MATLAB path.

   ```
   addpath d:/work/mytarget
   ```

3. Make `d:/work/mytarget` your working directory. Real-Time Workshop writes the output files of the code generation process into a build directory within the working directory.

4. Copy the `timestwo` S-function C source code from *matlabroot*`/toolbox/rtw/rtwdemos/tlctutorial/timestwo/solutions/timestwo.c` to `d:/work/mytarget`.

5. Build the `timestwo` MEX-file in `d:/work/mytarget`.

   ```
   mex timestwo.c
   ```

6. Create the model as illustrated in Figure 17-1. Use an S-Function block from the Simulink Functions & Tables library in the Library Browser. Set the solver options to `fixed-step` and `ode4`.

7. Double-click the S-Function block to open the **Block Parameters** dialog. Enter the S-function name `timestwo`. The block is now bound to the `timestwo` MEX-file. Click **OK**.

8 Open the Scope and run the simulation. Verify that the `timestwo` S-function multiplies its input by 2.0.

9 In order to generate inlined code from the `timestwo` S-Function block, you must have a corresponding TLC file in the working directory. If the Target Language Compiler detects a C-code S-function and a TLC file with the same name in the working directory, it generates inline code from the TLC file. Otherwise, it generates a function call to the external S-function.

To ensure that the build process generates inlined code from the `timestwo` block, copy the `timestwo` TLC source code from *matlabroot*`/toolbox/rtw/rtwdemos/tlctutorial/timestwo/solutions/timestwo.tlc` to `d:/work/mytarget`.

10 Make local copies of the main program and system target files. *matlabroot*`/rtw/c/grt` contains the main program (`grt_main.c`) and the system target file (`grt.tlc`) for the generic real-time target. Copy `grt_main.c` and `grt.tlc` to `d:/work/mytarget`. Rename them to `mytarget_main.c` and `mytarget.tlc`.

11 Remove the initial comment lines from `mytarget.tlc`. The lines to remove are shown below.

```
%% SYSTLC: Generic Real-Time Target \
%%    TMF: grt_default_tmf MAKE: make_rtw EXTMODE: ext_comm
%% SYSTLC: Visual C/C++ Project Makefile only for the "grt" target \
%%    TMF: grt_msvc.tmf MAKE: make_rtw EXTMODE: ext_comm
```

The initial comment lines have significance only if you want to add `my_target` to the System Target File Browser. For now you should remove them.

12 Real-Time Workshop creates a build directory in your working directory to store files created during the code generation process. The build directory is given the name of the model, followed by a suffix. This suffix is specified in the `rtwgensettings` structure in the system target file.

   To set the suffix to a more appropriate string, change the line

   ```
   rtwgensettings.BuildDirSuffix = '_grt_rtw'
   ```

   to

   ```
   rtwgensettings.BuildDirSuffix = '_mytarget_rtw'
   ```

   Your build directory will be named `targetModel__mytarget_rtw`.

13 Make a local copy of the template makefile. *matlabroot*`/rtw/c/grt` contains several compiler-specific template makefiles for the generic real-time target. The appropriate template makefile for the LCC compiler is `grt_lcc.tmf`. Copy `grt_lcc.tmf` to `d:/work/mytarget`, and rename it to `mytarget.tmf`.

Note Some of the template makefile modifications described in the next step are specific to the LCC template makefile. If you are using a different compiler and template makefile, the rules for the source (`REQ_SRCS`) and object file (`%.obj :`) lists may differ slightly.

---

14 Modify `mytarget.tmf`. The `SYS_TARGET FILE` parameter must be changed so that the correct file reference is generated in the `make` file. Change the line

```
SYS_TARGET FILE = grt.tlc
```

to

```
SYS_TARGET FILE = mytarget.tlc
```

Also, change the `source file` list to include `mytarget_main.c` instead of `grt_main.c`.

```
REQ_SRCS = $(MODEL).c $(MODULES) mytarget_main.c...
```

Finally, change the line

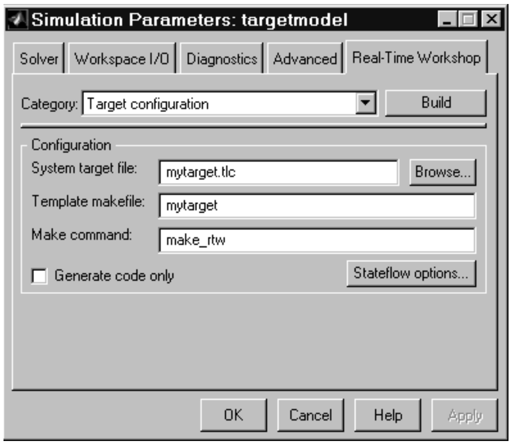
```
%.obj : $(MATLAB_ROOT)/rtw/c/grt/%.c
```

to

```
%.obj : d:/work/mytarget/%.c
```

15 This exercise requires no changes to `mytarget_main.c`. In an actual application, you would modify `mytarget_main.c` to execute your model code under the control of a timer interrupt, and make other changes.

16 Open the Real-Time Workshop page in the **Simulation Parameters** dialog. Select **Target configuration** from the **Category** menu. Enter the system target file, template makefile, and **Make command** parameters as below.



Category: Target configuration

Configuration

System target file: mytarget.tlc

Template makefile: mytarget

Make command: make_rtw

☐ Generate code only

17 Click the **Apply** button.

18 Click the **Build** button. If the build is successful, MATLAB will display the message below.

```
### Created executable: targetModel.exe
### Successful completion of Real-Time Workshop build procedure
for model: targetModel
```

Your working directory will contain the `targetModel.exe` file and the build directory, `targetModel_mytarget_rtw`.

19 Edit the generated file `d:/work/mytarget/targetModel_mytarget_rtw/targetModel.c` and locate the `MdlOutputs` function. Observe the inlined code.

```
/* S-Function Block: <Root>/S-Function (timestwo) */
rtB.S_Function = 2.0 * rtB.Sine_Wave;
```

Because the working directory contained a TLC file (`timestwo.tlc`) with the same name as the `timestwo` S-Function block, the Target Language Compiler generated inline code instead of a function call to the external C-code S-function.

20 As an optional final step to this exercise, you may want to add your custom target configuration to the System Target File Browser. See “Adding a Custom Target to the System Target File Browser” on page 17-24 to learn how to do this.

Customizing the Build Process

The Real-Time Workshop build process proceeds in two stages. The first stage is code generation. The system target file exerts overall control of the code generation stage. In the second stage, the template makefile generates a `.mk` file, which compiles and links code modules into an executable.

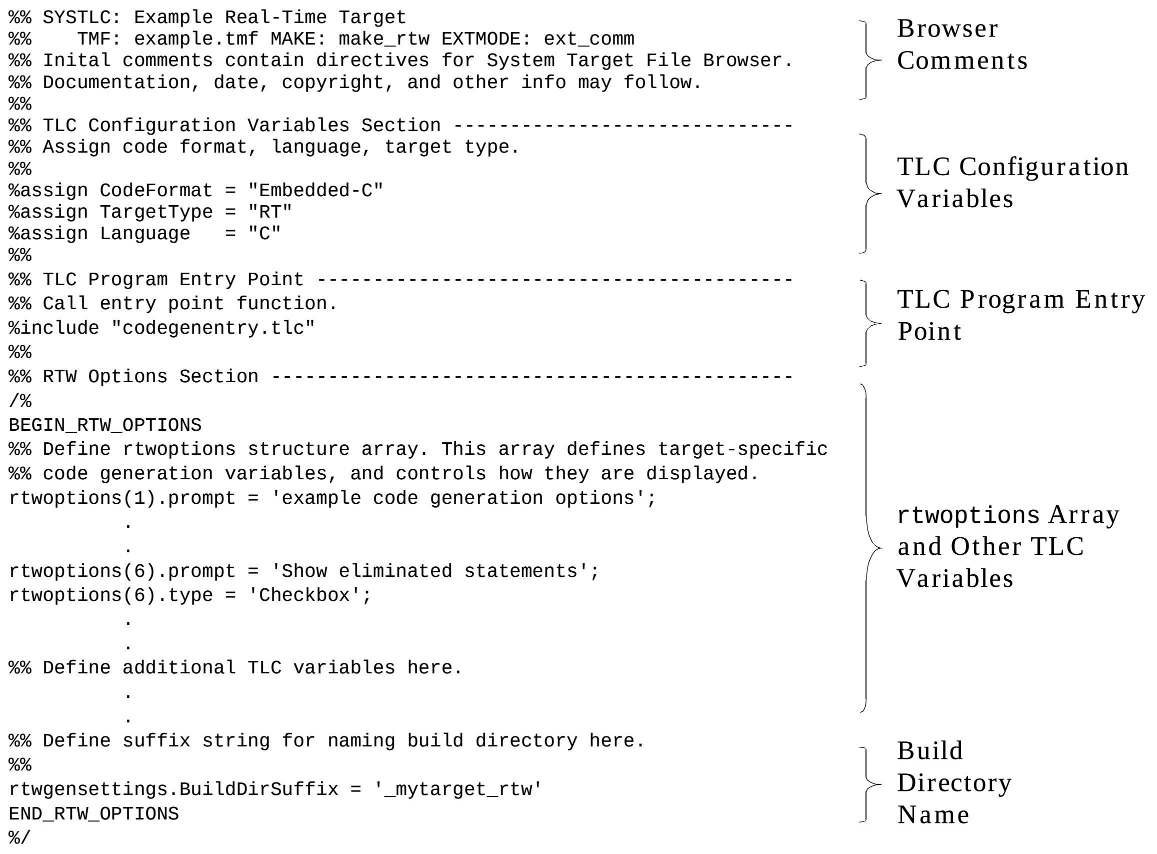
In developing your custom target, you may need to create a customized system target file and/or template makefile. This section provides information on the structure of these files, and guidelines for modifying them.

System Target File Structure

This section is a guide to the structure and contents of a system target file. You may want to refer to the system target files provided with the Real-Time Workshop while reading this section. Most of these files are stored in the target-specific directories under *matlabroot*`/rtw/c`. Additional system target files are stored in *matlabroot*`/toolbox/rtw/targets/rtwin/rtwin` and *matlabroot*`/toolbox/rtw/targets/xpc/xpc`.

Before creating or modifying a system target file, you should acquire a working knowledge of the Target Language Compiler. The *Target Language Compiler Reference Guide* documents the features and syntax of the language.

Figure 17-2 shows the general structure of a system target file.



```
%% SYSTLC: Example Real-Time Target
%%    TMF: example.tmf MAKE: make_rtw EXTMODE: ext_comm
%% Inital comments contain directives for System Target File Browser.
%% Documentation, date, copyright, and other info may follow.
%%
%% TLC Configuration Variables Section -----------------------------
%% Assign code format, language, target type.
%%
%assign CodeFormat = "Embedded-C"
%assign TargetType = "RT"
%assign Language   = "C"
%%
%% TLC Program Entry Point ----------------------------------------
%% Call entry point function.
%include "codegenentry.tlc"
%%
%% RTW Options Section ---------------------------------------------
/%
BEGIN_RTW_OPTIONS
%% Define rtwoptions structure array. This array defines target-specific
%% code generation variables, and controls how they are displayed.
rtwoptions(1).prompt = 'example code generation options';
          .
          .
rtwoptions(6).prompt = 'Show eliminated statements';
rtwoptions(6).type = 'Checkbox';
          .
          .
%% Define additional TLC variables here.
          .
          .
%% Define suffix string for naming build directory here.
%%
rtwgensettings.BuildDirSuffix = '_mytarget_rtw'
END_RTW_OPTIONS
%/
```

Browser Comments
TLC Configuration Variables
TLC Program Entry Point
rtwoptions Array and Other TLC Variables
Build Directory Name

Figure 17-2: Structure of a System Target File

Browser Comments

This section is optional. You can place comment lines at the head of the file to identify your system target file to the System Target File Browser. These lines have significance to the browser only. During code generation, the Target Language Compiler treats them as comments.

Note that you must place the browser comments at the head of the file, before any other comments or TLC statements.

The comments contain the following directives:

- `SYSTLC`: This string is a descriptor that appears in the browser.
- `TMF`: Name of the template makefile to use during build process. When the target is selected, this filename is displayed in the **Template makefile** field of the **Target configuration** section of the Real-Time Workshop page.
- `MAKE`: `make` command to use during build process. When the target is selected, this command is displayed in the **Make command** field of the **Target configuration** section of the Real-Time Workshop page.
- `EXTMODE`: Name of external mode interface file (if any) associated with your target. If your target does not support external mode, use `no_ext_comm`.

The following browser information comments are from *matlabroot*`/rtw/c/grt/grt.tlc`.

```
%% SYSTLC: Generic Real-Time Target
%%    TMF: grt_default_tmf MAKE: make_rtw EXTMODE: ext_comm
```

See "Adding a Custom Target to the System Target File Browser" on page 17-24 for further information.

Target Language Compiler Configuration Variables

This section assigns global TLC variables that affect the overall code generation process. The following variables must be assigned:

- `CodeFormat`: The `CodeFormat` variable selects one of the available code formats:
  - `RealTime`: Designed for rapid prototyping, with static memory allocation.
  - `RealTimeMalloc`: Similar to `RealTime`, but with dynamic memory allocation.
  - `Embedded-C`: Designed for production code, minimal memory usage, simplified interface to generated code.
  - `S-Function`: For use by S-function and Accelerator targets only.

- Ada: Designed for production code, minimal memory usage, simplified interface to generated code.

  The default CodeFormat value is RealTime.

  Chapter 4, “Generated Code Formats” summarizes available code formats and provides pointers to further details.

- Language: Selects code generation in one of the supported languages:
  - C
  - Ada

    When Ada is selected, Real-Time Workshop generates Ada95 code by default. To generate Ada83 code, set the variable AdaVersion as follows.

  ```
  %assign AdaVersion = “83”
  ```

  It is possible to generate code in a language other than C or Ada. To do this would require considerable development effort, including reimplementation of all block target files to generate the desired target language code. See the *Target Language Compiler Reference Guide* for a discussion of the issues.

- TargetType: The Real-Time Workshop defines the preprocessor symbols RT and NRT to distinguish simulation code from real-time code. These symbols are used in conditional compilation. The TargetType variable determines whether RT or NRT is defined.

  Most targets are intended to generate real-time code. They assign TargetType as follows.

  ```
  %assign TargetType = "RT"
  ```

  Some targets, such as the Simulink Accelerator, generate code for use in non real-time only. Such targets assign TargetType as follows.

  ```
  %assign TargetType = "NRT"
  ```

  See “Conditional Compilation for Simulink and Real-Time” on page 17–40 for further information on the use of these symbols.

Target Language Compiler Program Entry Point

The code generation process normally begins with codegenentry.tlc. The system target file invokes codegenentry.tlc as follows.

```
%include "codegenentry.tlc"
```

`codegenentry.tlc` in turn invokes other TLC files:

- `genmap.tlc` maps the block names to corresponding language-specific block target files.
- `commonsetup.tlc` sets up global variables.
- `commonentry.tlc` starts the process of generating code in the format specified by `CodeFormat`.

To customize the code generation process, you can call the lower-level TLC files explicitly and include your own TLC functions at each stage of the process. See the *Target Language Compiler Reference Guide* for guidelines.

Note `codegenentry.tlc` and the lower-level TLC files assume that `CodeFormat`, `TargetType`, and `Language` have been correctly assigned. Set these variables before including `codegenentry.tlc`.

RTW_OPTIONS Section

The `RTW_OPTIONS` section (see Figure 17-2) is bounded by the directives:

```
%/
BEGIN_RTW_OPTIONS
.
.
END_RTW_OPTIONS
/%
```

The first part of the `RTW_OPTIONS` section defines an array of `rtwoptions` structures. The `rtwoptions` structure is discussed in this section.

The second part of the `RTW_OPTIONS` section defines `rtwgensettings`, a structure defining the build directory name and other settings for the code generation process. See "Build Directory Name" on page 17-24 for information about `rtwgensettings`.

The rtwoptions Structure. The fields of the `rtwoptions` structure define variables and associated user interface elements to be displayed in the Real-Time Workshop page. Using the `rtwoptions` structure array, you can customize the **Category** menu in the Real-Time Workshop page, define the options displayed in each category, and specify how these options are processed.

When the Real-Time Workshop page opens, the `rtwoptions` structure array is scanned and the listed options are displayed. Each option is represented by an assigned user interface element (check box, edit field, pop-up menu, or pushbutton), which displays the current option value.

The user interface elements can be in an enabled or disabled (grayed-out) state. If the option is enabled, the user can change the option value.

The elements of the `rtwoptions` structure array are organized into groups that correspond to items in the **Category** menu in the Real-Time Workshop page. Each group of items begins with a header element of type `Category`. The `default` field of a `Category` header must contain a count of the remaining elements in the category.

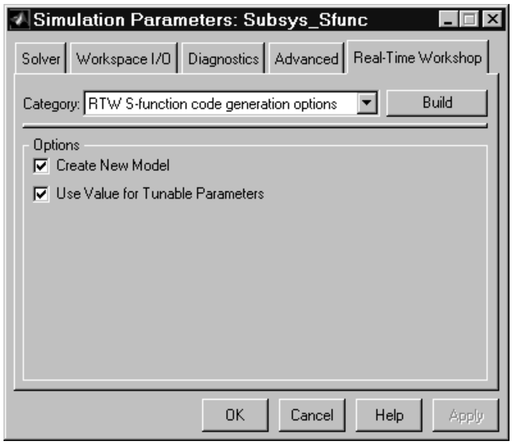
The header is followed by options to be displayed on the Real-Time Workshop page. The header in each category is followed by a maximum of seven elements.

Table 17-2 summarizes the fields of the `rtwoptions` structure.

The following example is excerpted from *matlabroot*`/rtw/c/rtwsfcn/rtwsfcn.tlc`, the system target file for the S-Function target. The code defines an `rtwoptions` structure array of three elements. The `default` field of the first (header) element is set to 2, indicating the number of elements that follow the header.

```
rtwoptions(1).prompt = 'RTW S-function code generation options';
rtwoptions(1).type = 'Category';
rtwoptions(1).enable = 'on';
rtwoptions(1).default = 2; % Number of items under this category
                           % excluding this one.

rtwoptions(2).prompt = 'Create New Model';
rtwoptions(2).type = 'Checkbox';
rtwoptions(2).default = 'on';
rtwoptions(2).tlcvariable = 'CreateModel';
rtwoptions(2).makevariable = 'CREATEMODEL';
rtwoptions(2).tooltip = ...
  ['Create a new model containing the generated RTW S-Function block inside it'];

rtwoptions(3).prompt = 'Use Value for Tunable Parameters';
rtwoptions(3).type = 'Checkbox';
rtwoptions(3).default = 'off';
rtwoptions(3).tlcvariable = 'UseParamValues';
rtwoptions(3).makevariable = 'USEPARAMVALUES';
rtwoptions(3).tooltip = ...
['Use value instead of variable name in generated block mask edit fields'];
```

The first element adds the **RTW S-function code generation options** item to the **Category** menu of the Real-Time Workshop page. The options defined in `rtwoptions(2)` and `rtwoptions(3)` display as shown in Figure 17-3.



Category: RTW S-function code generation options

Options

☑ Create New Model

☑ Use Value for Tunable Parameters

Figure 17-3: Code Generation Options for S-Function Target

If you want to define more than seven options, you can define multiple **Category** menu items within a single system target file. For an example, see the Tornado system target file, *matlabroot*`/rtw/c/tornado/tornado.tlc`.

For further examples of target-specific `rtwoptions` definitions, see the system target files in the other target directories under *matlabroot*`/rtw/c`.

Note that to verify the syntax of your `rtwoptions` definitions, you can execute the commands in MATLAB by copying and pasting them to the MATLAB command window.

The following table lists the fields of the `rtwoptions` structure.

Table 17-2: rtwoptions Structure Fields Summary

Field Name	Description
`callback`	Name of M-code function to call when value of option changes.
`default`	Default value of the option (empty if the `type` is `Pushbutton`).
`enable`	Must be `on` or `off`. If `on`, the option is displayed as an enabled item; otherwise, as a disabled item.
`makevariable`	Template makefile token (if any) associated with option. The `makevariable` will be expanded during processing of the template makefile. See “Template Makefile Tokens” on page 17-26.
`opencallback`	M-code to be executed when dialog opens. The purpose of the code is to synchronize the displayed value of the option with its previous setting. See *matlabroot*`/rtw/c/ert/ert.tlc` for an example.

`popupstrings`	If `type` is `Popup`, `popupstrings` defines the items in the pop-up menu. Items are delimited by the “|” (vertical bar) character. The following example defines the items of the **Function management** menu used by the GRT and other targets:
	`['None|Function splitting|File ', ...`
	` 'splitting|Function and file splitting']`

`prompt`	Label for the option.
`tlcvariable`	Name of TLC variable associated with the option.
`tooltip`	Help string displayed when mouse is over the item.
`type`	Type of element: `Checkbox`, `Edit`, `Popup`, `Pushbutton`, or `Category`.

Additional Code Generation Options

"Target Language Compiler Variables and Options" on page 3-93 describes additional code generation variables. For readability, it is recommended that you assign these variables in the **Configure RTW code generation settings** section of the system target file.

Alternatively, you can append statements of the form

```
-aVariable=val
```

to the **System target filename** field on the Real-Time Workshop page.

Build Directory Name

The final part of the system target file defines the `BuildDirSuffix` field of the `rtwgensettings` structure. The build process appends the `BuildDirSuffix` string to the model name to form the name of the build directory. For example, if you define `BuildDirSuffix` as follows

```
rtwgensettings.BuildDirSuffix = '_mytarget_rtw'
```

the build directories are named *model*`_mytarget_rtw`.

See the *Target Language Compiler Reference Manual* for further information on the `rtwgensettings` structure.

Adding a Custom Target to the System Target File Browser

As a convenience to end users of your custom target configuration, you can add a custom target configuration to the System Target File Browser. To do this:

1 Modify (or add) browser comments at the head of your custom system target file. For example,

```
%% SYSTLC: John's Real-Time Target \
%%    TMF: mytarget.tmf  MAKE: make_rtw EXTMODE: no_ext_comm
```

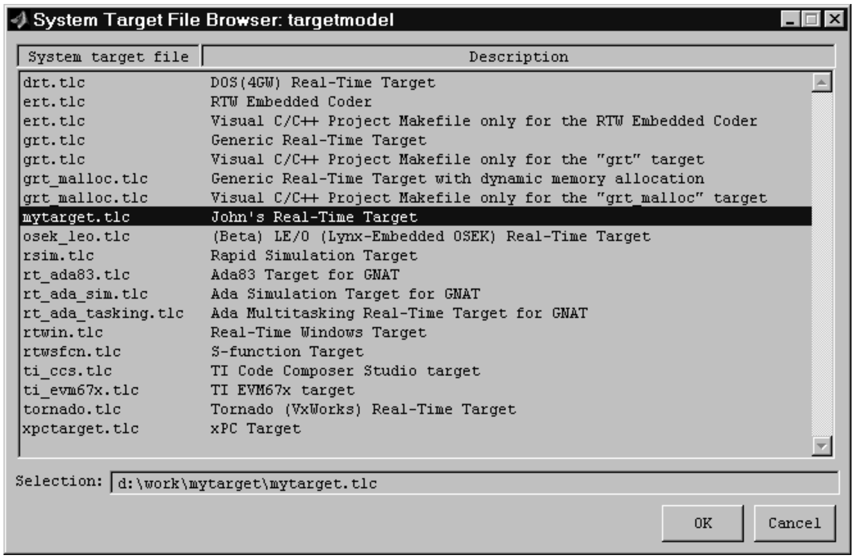
2 Create a directory `<targetname>` (e.g., `/mytarget`). Move your custom system target file, custom template makefile, and run-time interface files (such as your main program and S-functions) into the `<targetname>` subdirectory.

3 Add your target directory to the MATLAB path.

```
addpath <targetname>
```

If you want `<targetname>` included in the MATLAB path each time MATLAB starts up, include this `addpath` command in your `startup.m` file.

4 When the System Target File Browser opens, Real-Time Workshop detects system target files that are on the MATLAB path, and displays the target filenames and target description comments. Figure 17-4 shows how the target file `mytarget.tlc`, which contains the browser comments above, appears in the System Target File Browser.


System Target File Browser: targetmodel

System target file	Description
drt.tlc	DOS(4GW) Real-Time Target
ert.tlc	RTW Embedded Coder
ert.tlc	Visual C/C++ Project Makefile only for the RTW Embedded Coder
grt.tlc	Generic Real-Time Target
grt.tlc	Visual C/C++ Project Makefile only for the "grt" target
grt_malloc.tlc	Generic Real-Time Target with dynamic memory allocation
grt_malloc.tlc	Visual C/C++ Project Makefile only for the "grt_malloc" target
mytarget.tlc	John's Real-Time Target
osek_leo.tlc	(Beta) LE/O (Lynx-Embedded OSEK) Real-Time Target
rsim.tlc	Rapid Simulation Target
rt_ada83.tlc	Ada83 Target for GNAT
rt_ada_sim.tlc	Ada Simulation Target for GNAT
rt_ada_tasking.tlc	Ada Multitasking Real-Time Target for GNAT
rtwin.tlc	Real-Time Windows Target
rtwsfcn.tlc	S-function Target
ti_ccs.tlc	TI Code Composer Studio target
ti_evm67x.tlc	TI EVM67x target
tornado.tlc	Tornado (VxWorks) Real-Time Target
xpctarget.tlc	xPC Target

Selection: d:\work\mytarget\mytarget.tlc

Figure 17-4: Custom System Target File Displayed in Browser

Template Makefiles

To configure or customize template makefiles, you should be familiar with how the `make` command works and how the `make` command processes makefiles. You should also understand makefile build rules. For information of these topics, please refer to the documentation provided with the make utility you use. There are also several good books on the make utility.

Template makefiles are made up of statements containing tokens. The Real-Time Workshop build process expands tokens and creates a makefile, *model*.mk. Template makefiles are designed to generate makefiles for specific compilers on specific platforms. The generated *model*.mk file is specifically tailored to compile and link code generated from your model, using commands specific to your development system.

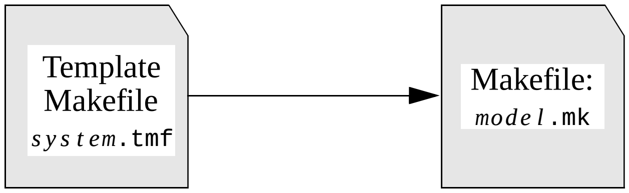


Figure 17-5: Creation of model.mk

Template Makefile Tokens

The make_rtw M-file command (or a different command provided with some targets) directs the process of generating *model*.mk. The make_rtw command processes the template makefile specified on the **Target configuration** section of the Real-Time Workshop page of the **Simulation Parameters** dialog. make_rtw copies the template makefile, line by line, expanding each token encountered. Table 17-3 lists the tokens and their expansions.

Table 17-3: Template Makefile Tokens Expanded by make_rtw

Token	Expansion
|>COMPUTER<|	Computer type. See the MATLAB computer command.
|>MAKEFILE_NAME<|	*model*.mk — The name of the makefile that was created from the template makefile.
|>MATLAB_ROOT<|	Path to where MATLAB is installed.
|>MATLAB_BIN<|	Location of the MATLAB executable.

Table 17-3: Template Makefile Tokens Expanded by make_rtw (Continued)

Token	Expansion
`|>MEM_ALLOC<|`	Either `RT_MALLOC` or `RT_STATIC`. Indicates how memory is to be allocated.
`|>MEXEXT<|`	MEX-file extension. See the MATLAB `mexext` command.
`|>MODEL_NAME<|`	Name of the Simulink block diagram currently being built.
`|>MODEL_MODULES<|`	Any additional generated source (`.c`) modules. For example, you can split a large model into two files, *model*`.c` and *model*`1.c`. In this case, this token expands to *model*`1.c`.
`|>MODEL_MODULES_OBJ<|`	Object filenames (`.obj`) corresponding to any additional generated source (`.c`) modules.
`|>MULTITASKING<|`	True (1) if solver mode is multitasking, otherwise False (0).
`|>NUMST<|`	Number of sample times in the model.
`|>RELEASE_VERSION<|`	The release version of MATLAB.
`|>S_FUNCTIONS<|`	List of noninlined S-function (`.c`) sources.
`|>S_FUNCTIONS_LIB<|`	List of S-function libraries available for linking.
`|>S_FUNCTIONS_OBJ<|`	Object (`.obj`) file list corresponding to noninlined S-function sources.
`|>SOLVER<|`	Solver source filename, e.g., `ode3.c`.
`|>SOLVER_OBJ<|`	Solver object (`.obj`) filename, e.g., `ode3.obj`.

Table 17-3: Template Makefile Tokens Expanded by make_rtw (Continued)

Token	Expansion
`|>TID01EQ<|`	True (1) if sampling rates of the continuous task and the first discrete task are equal, otherwise False (0).
`|>NCSTATES<|`	Number of continuous states.
`|>BUILDARGS<|`	Options passed to `make_rtw`. This token is provided so that the contents of your *model*`.mk` file will change when you change the build arguments, thus forcing an update of all modules when your build options change.
`|>EXT_MODE<|`	True (1) to enable generation of external mode support code, otherwise False (0).

These tokens are expanded by substitution of parameter values known to the build process. For example, if the source model contains blocks with two different sample times, the template makefile statement

```
NUMST = |>NUMST<|
```

expands to the following in *model*`.mk`.

```
NUMST = 2
```

In addition to the above, `make_rtw` expands tokens from other sources:

- Target-specific tokens defined via the **Target configuration** section of the Real-Time Workshop page of the **Simulation Parameters** dialog box.
- Structures in the **RTW Options** section of the system target file. Any structures in the `rtwoptions` structure array that contain the field `makevariable` are expanded.

  The following example is extracted from *matlabroot*`/rtw/c/grt/grt.tlc`. The section starting with `BEGIN_RTW_OPTIONS` contains M-file code that sets up `rtwoptions`. The directive

```
rtwoptions(2).makevariable = 'EXT_MODE'
```

causes the `|>EXT_MODE<|` token to be expanded into `1` (on) or `0` (off), depending on how you set the **External mode** option in the **Code generation options** section of the Real-Time Workshop page.

The Make Command

After creating *model.mk* from your template makefile, the Real-Time Workshop invokes a `make` command. To invoke `make`, the Real-Time Workshop issues this command.

```
makecommand -f model.mk
```

makecommand is defined by the `MAKE` macro in your system's template makefile (see Figure 17-6 on page 17-31). You can specify additional options to `make` in the **Make command** field of the Real-Time Workshop page. (see "Make Command Field" on page 3-8 and "Template Makefiles and Make Options" on page 3-102.)

For example, specifying `OPT_OPTS=-O2` in the **Make command** field causes `make_rtw` to generate the following `make` command.

```
makecommand -f model.mk OPT_OPTS=-O2
```

A comment at the top of the template makefile specifies the available `make` command options. If these options do not provide you with enough flexibility, you can configure your own template makefile.

Make Utilities

The make utility lets you control nearly every aspect of building your real-time program. There are several different versions of `make` available. The Real-Time Workshop provides the Free Software Foundation's GNU Make for both UNIX and PC platforms in the platform-specific subdirectories below *matlabroot*/`rtw/bin`.

It is possible to use other versions of `make` with the Real-Time Workshop, although GNU Make is recommended. To ensure compatibility with the Real-Time Workshop, make sure that your version of `make` supports the following command format.

```
makecommand –f model.mk
```

Structure of the Template Makefile

A template makefile has four sections:

- The first section contains initial comments that describe what this makefile targets.
- The second section defines macros that tell `make_rtw` how to process the template makefile. The macros are:
  - `MAKE` — This is the command used to invoke the make utility. For example, if `MAKE` = `mymake`, then the `make` command invoked is

    ```
    mymake –f model.mk
    ```
  - `HOST` — What platform this template makefile is targeted for. This can be `HOST=PC`, `UNIX`, `computer_name` (see the MATLAB `computer` command), or `ANY`.
  - `BUILD` — This tells `make_rtw` whether or not (`BUILD=yes` or `no`) it should invoke `make` from the Real-Time Workshop build procedure.
  - `SYS_TARGET_FILE` — Name of the system target file. This is used for consistency checking by `make_rtw` to verify that the correct system target file was specified in the **Target configuration** section of the Real-Time Workshop page of the **Simulation Parameters** dialog box.
  - `BUILD_SUCCESS` — An optional macro that specifies the build success string to be displayed on successful `make` completion on the PC. For example,

    ```
    BUILD_SUCCESS = ### Successful creation of
    ```
  - `BUILD_ERROR` — An optional macro that specifies the build error message to be displayed when an error is encountered during the `make` procedure. For example,

    ```
    BUILD_ERROR = ['Error while building ', modelName]
    ```

  The following `DOWNLOAD` options apply only to the Tornado target:

  - `DOWNLOAD` — An optional macro that you can specify as yes or no. If specified as yes (and `BUILD=yes`), then `make` is invoked a second time with the download target.

    ```
    make -f model.mk download
    ```
  - `DOWNLOAD_SUCCESS` — An optional macro that you can use to specify the download success string to be used when looking for a successful download. For example,

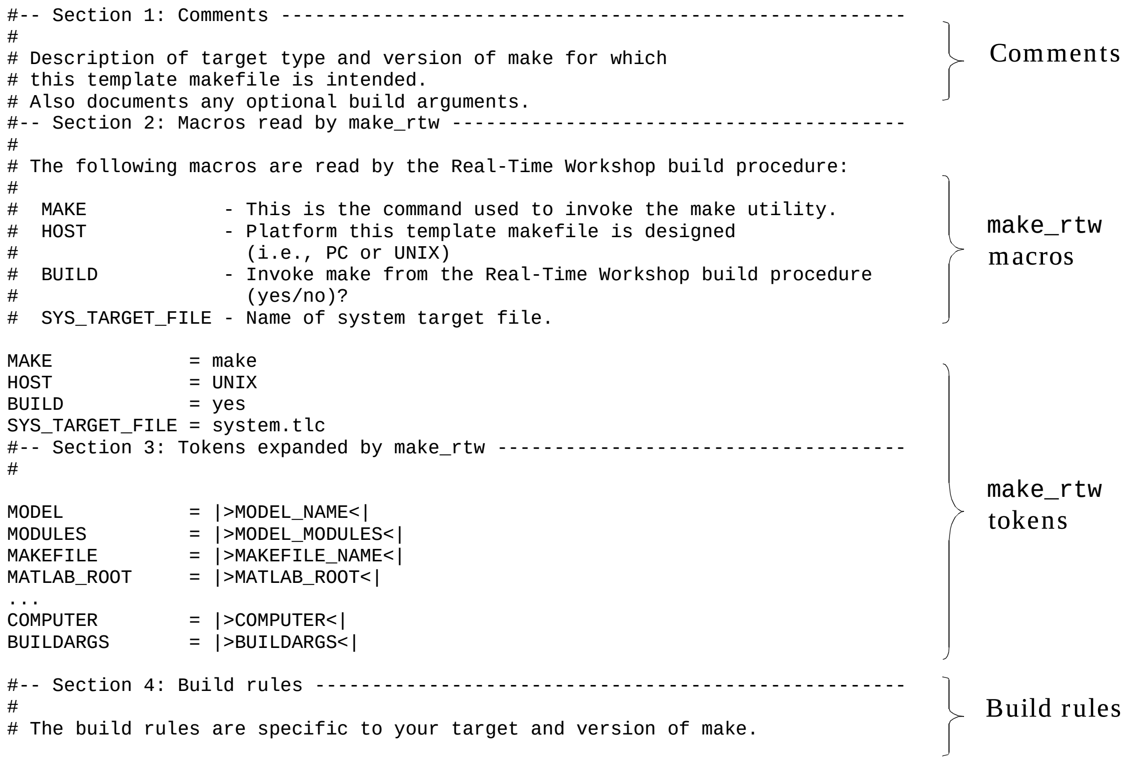
    ```
    DOWNLOAD_SUCCESS = ### Downloaded
    ```

  - DOWNLOAD_ERROR — An optional macro that you can use to specify the download error message to be displayed when an error is encountered during the download. For example,

    ```
    DOWNLOAD_ERROR = ['Error while downloading ', modelName]
    ```

- The third section defines the tokens `make_rtw` expands (see Table 17-3).
- The fourth section contains the make rules used in building an executable from the generated source code. The build rules are typically specific to your version of make.

Figure 17-6 shows the general structure of a template makefile.



```
#-- Section 1: Comments -------------------------------------------------------
#
# Description of target type and version of make for which
# this template makefile is intended.
# Also documents any optional build arguments.
#-- Section 2: Macros read by make_rtw ----------------------------------------
#
# The following macros are read by the Real-Time Workshop build procedure:
#
#  MAKE            - This is the command used to invoke the make utility.
#  HOST            - Platform this template makefile is designed
#                    (i.e., PC or UNIX)
#  BUILD           - Invoke make from the Real-Time Workshop build procedure
#                    (yes/no)?
#  SYS_TARGET_FILE - Name of system target file.

MAKE            = make
HOST            = UNIX
BUILD           = yes
SYS_TARGET_FILE = system.tlc
#-- Section 3: Tokens expanded by make_rtw ------------------------------------
#

MODEL           = |>MODEL_NAME<|
MODULES         = |>MODEL_MODULES<|
MAKEFILE        = |>MAKEFILE_NAME<|
MATLAB_ROOT     = |>MATLAB_ROOT<|
...
COMPUTER        = |>COMPUTER<|
BUILDARGS       = |>BUILDARGS<|

#-- Section 4: Build rules ----------------------------------------------------
#
# The build rules are specific to your target and version of make.
```

Figure 17-6: Structure of aTemplate Makefile

Customizing and Creating Template Makefiles

To customize or create a new template makefile, we recommend that you copy an existing template makefile to your local working directory and modify it.

This section shows, through an example, how to use macros and file-pattern-matching expressions in a template makefile to generate commands in the *model*.mk file.

The make utility processes the *model*.mk makefile and generates a set of commands based upon dependency rules defined in *model*.mk. After make generates the set of commands needed to build or rebuild test, make executes them.

For example, to build a program called test, make must link the object files. However, if the object files don't exist or are out of date, make must compile the C code. Thus there is a dependency between source and object files.

Each version of make differs slightly in its features and how rules are defined. For example, consider a program called test that gets created from two sources, file1.c and file2.c. Using most versions of make, the dependency rules would be

```
test: file1.o file2.o
        cc –o test file1.o file2.o

file1.o: file1.c
        cc –c file1.c

file2.o: file2.c
        cc –c file2.c
```

In this example, we assumed a UNIX environment. In a PC environment the file extensions and compile and link commands will be different.

In processing the first rule

```
test: file1.o file2.o
```

make sees that to build test, it needs to build file1.o and file2.o. To build file1.o, make processes the rule

```
file1.o: file1.c
```

If `file1.o` doesn't exist, or if `file1.o` is older than `file1.c`, `make` compiles `file1.c`.

The format of Real-Time Workshop template makefiles follows the above example. Our template makefiles use additional features of `make` such as macros and file-pattern-matching expressions. In most versions of `make`, a macro is defined via

```
MACRO_NAME = value
```

References to macros are made via `$(MACRO_NAME)`. When `make` sees this form of expression, it substitutes *value* for `$(MACRO_NAME)`.

You can use pattern matching expressions to make the dependency rules more general. For example, using GNU Make you could replace the two "`file1.o: file1.c`" and "`file2.o: file2.c`" rules with the single rule

```
%.o : %.c
        cc –c $<
```

Note that `$<` above is a special macro that equates to the dependency file (i.e., `file1.c` or `file2.c`). Thus, using macros and the "%" pattern matching character, the above example can be reduced to

```
SRCS = file1.c file2.c
OBJS = $(SRCS:.c=.o)

test: $(OBJS)
        cc –o $@ $(OBJS)

%.o : %.c
        cc –c $<
```

Note that the `$@` macro above is another special macro that equates to the name of the current dependency target, in this case `test`.

This example generates the list of objects (`OBJS`) from the list of sources (`SRCS`) by using the string substitution feature for macro expansion. It replaces the source file extension (`.c`) with the object file extension (`.o`). This example also generalized the build rule for the program, `test`, to use the special "`$@`" macro.

Creating Device Drivers

Device drivers that communicate with target hardware are essential to many real-time development projects. This section describes how to integrate device drivers into your target system. This includes incorporating drivers into your Simulink model and into the code generated from that model.

Device drivers are implemented as Simulink device driver blocks. A device driver block is an S-Function block that is bound to user-written driver code.

To implement device drivers, you should be familiar with the Simulink C MEX S-function format and API. The following documents contain more information about C MEX S-functions:

- *Writing S-Functions* describes S-functions, including how to write both inlined and noninlined S-functions and how to access parameters from a masked S-function. *Writing S-Functions* also describes how to use the special `mdlRTW` function to parameterize an inlined S-function.
- "External Interfaces" in the MATLAB online documentation explains how to write C and other programs that interact with MATLAB via the MEX API. Simulink's S-function API is built on top of this API. To pass parameters to your device driver block from MATLAB/Simulink you must use the MEX API. *MATLAB Application Program Interface Reference Guide* contains reference descriptions for the required MATLAB `mx*` routines.
- *Target Language Compiler Reference Guide* describes the Target Language Compiler. Knowledge of the Target Language Compiler is required in order to inline S-functions. The *Target Language Compiler Reference Guide* also describes the structure of the *`model`*`.rtw` file.
- "Using Masks to Customize Blocks" in *Using Simulink* describes how to create a mask for an S-function.

Note Device driver blocks must be implemented as C MEX S-functions, not as M-file S-functions. C MEX S-functions are limited to a subset of the features available in M-file S-functions. See "Limitations of Device Driver Blocks" on page 17-37 for information.

This section covers the following topics:

- Inlined and noninlined device drivers
- General requirements and limitations for device drivers
- Obtaining S-function parameter values from a dialog box
- Writing noninlined device drivers
- Writing inlined device drivers
- Building the device driver MEX-file

Inlined and Noninlined Drivers

In your target system, a device driver has a dual function. First, it functions as a code module that you compile and link with other code generated from your model by Real-Time Workshop. In addition, the driver must interact with Simulink during simulation. To meet both these requirements, you must incorporate your driver code into a Simulink device driver block.

You can build your driver S-function in several ways:

- As a MEX-file component, bound to an S-Function block, for use in a Simulink model. In this case, the Simulink engine calls driver routines in the MEX-file during execution of the model.
- As a module within a stand-alone real-time program that is generated from a model by Real-Time Workshop. The driver routines are called from within the application in essentially the same way that Simulink calls them.

  In many cases, the code generated from driver blocks for real-time execution must run differently from the code used by the blocks in simulation. For example, an output driver may write to hard device addresses in real time; but these write operations could cause errors in simulation.

  Real-Time Workshop provides standard compilation conditionals and include files to let you build the drivers for both cases. (See “Conditional Compilation for Simulink and Real-Time” on page 17-40.)
- As *inlined* code. The Target Language Compiler enables you to generate the explicit code from your routines (instead of calls to these routines) in the body of the application. Inlined code eliminates calling overhead, and reduces memory usage.

Inlining an S-function can improve its performance significantly. However, there is a tradeoff in increased development and maintenance effort. To inline a device driver block, you must implement the block twice: first, as a C MEX-file, and second, as a TLC program.

The C MEX-file version is for use in simulation. Since a simulation normally does not have access to I/O boards or other target hardware, the C MEX-file version often acts as a “dummy” block within a model. For example, a digital-to-analog converter (DAC) device driver block is often implemented as a stub for simulation.

Alternatively, the C MEX-file version can simulate the behavior of the hardware. For example, an analog-to-digital converter (ADC) device driver block might read sample values from a data file or from the MATLAB workspace.

The TLC version generates actual working code that accesses the target hardware in a production system.

Inlined device drivers are an appropriate design choice when:

- You are using the Real-Time Workshop Embedded Coder target. Inlined S-functions are *required* when building code from the Real-Time Workshop Embedded Coder target. S-functions for other targets can be either inlined or noninlined.
- You need production code generated from the S-function to behave differently than code used during simulation. For example, an output device block may write to an actual hardware address in generated code, but perform no output during simulation.
- You want to avoid overhead associated with calling the S-function API.
- You want to reduce memory usage. Note that each noninlined S-function creates its own `Simstruct`. Each `Simstruct` uses over 1K of memory. Inlined S-functions do not allocate any `Simstruct`. For optimal memory usage, consider using inlined S-functions with the Real-Time Workshop Embedded Coder target.
- You want to avoid making calls to routines that are required by Simulink, but which are empty, in your generated code.

Device Driver Requirements and Limitations

In order to create a device driver block, the following components are required:

- Hardware-specific driver code, which handles communication between a real-time program and an I/O device. See your I/O device documentation for information on hardware requirements.
- S-function code, which implements the model initialization, output, and other functions required by the S-function API. The S-function code calls your driver code.

  Your S-function code and the hardware-specific driver code are compiled and linked into a component that is bound to an S-Function block in your Simulink model. The MATLAB `mex` utility builds this component (a DLL under Windows, or a shared library under UNIX).

We recommend that you use the S-function template provided by the Real-Time Workshop as a starting point for developing your driver S-functions. The template file is

matlabroot`/simulink/src/sfuntmpl.c`

An extensively commented version of the S-function template is also available. See *matlabroot*`/simulink/src/sfuntmpl.doc`.

The following components are optional:

- A TLC file that generates inline code for the S-function.
- A mask for the device driver block to create a customized user interface.

Limitations of Device Driver Blocks

The following limitations apply to *noninlined* driver blocks:

- Only a subset of MATLAB API functions are supported. See the “Noninlined S-functions” section of Chapter 4 of *Writing S-Functions* for a complete list of supported calls.
- Parameters must be doubles or characters contained in scalars, vectors, or 2-D matrices.

The following applies to *inlined* driver blocks:

- If the driver does not have a `mdlRTW` function, parameter restrictions are the same as for noninlined drivers.
- If the driver has a `mdlRTW` function, any parameter type is supported.

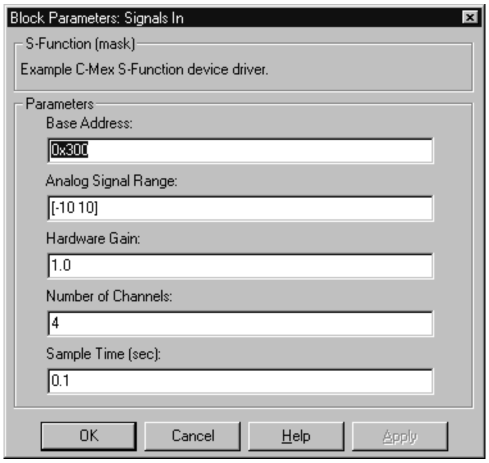
Preemption

Consider preemption issues in the design of your drivers. In a typical real-time program, a timer interrupt invokes `rtOneStep`, which in turn calls `MdlOutputs`, which in turn calls your input (ADC) and /or output (DAC) drivers. In this situation, your drivers are interruptible.

Parameterizing Your Driver

You can add a custom icon, dialog box, and initialization commands to an S-Function block by masking it. This provides an easy-to-use graphical user interface for your device driver in the Simulink environment.

You can parameterize your driver by letting the user enter hardware-related variables. Figure 17-7 shows the dialog box of a masked device driver block for an input (ADC) device. The Simulink user can enter the device address, the number of channels, and other operational parameters.


Block Parameters: Signals In

S-Function (mask)

Example C-Mex S-Function device driver.

Parameters

Base Address: 0x300

Analog Signal Range: [-10 10]

Hardware Gain: 1.0

Number of Channels: 4

Sample Time (sec): 0.1

Figure 17-7: Dialog Box for a Masked ADC Driver Block

A masked S-Function block obtains parameter data from its dialog box using macros and functions provided for the purpose.

To obtain a parameter value from the dialog:

1 Access the parameter from the dialog box using the `ssGetSFcnParam` macro. The arguments to `ssGetSFcnParam` are a pointer to the block's `Simstruct`, and the index (0-based) to the desired parameter. For example, use the following call to access the **Number of Channels** parameter from the dialog above.

   ```
   ssGetSFcnParam(S,3); /* S points to block's Simstruct */
   ```

2 Parameters are stored in arrays of type `mxArray`, even if there is only a single value. Get a particular value from the input `mxArray` using the `mxGetPr` function. The following code fragment extracts the first (and only) element in the **Number of Channels** parameter.

   ```
   #define  NUM_CHANNELS_PARAM        (ssGetSFcnParam(S,3))
   #define NUM_CHANNELS ((uint_T) mxGetPr(NUM_CHANNELS_PARAM)[0])
   uint_T num_channels;
   num_channels = NUM_CHANNELS;
   ```

It is typical for a device driver block to read and validate input parameters in its `mdlInitializeSizes` function. See the listing "adc.c" on page 17-55 for an example.

By default, S-function parameters are tunable. To make a parameter nontunable, use the `ssSetSFcParamNotTunable` macro in the `mdlInitializeSizes` routine. Nontunable S-function parameters become constants in the generated code, improving performance.

For further information on creation and use of masked blocks, see the *Using Simulink* and *Writing S-Functions* manuals.

Writing a Noninlined S-Function Device Driver

Overview

Device driver S-functions are relatively simple to implement because they perform only a few operations. These operations include:

- Initializing the `SimStruct`.
- Initializing the I/O device.
- Calculating the block outputs. How this is done depends upon the type of driver being implemented:

- An input driver for a device such as an ADC reads values from an I/O device and assigns these values to the block's output vector `y`.
- An output driver for a device such as a DAC writes values from the block's input vector `u` to an I/O device.

• Terminating the program. This may require setting hardware to a "neutral" state; for example, zeroing DAC outputs.

Your driver performs these operations by implementing certain specific functions required by the S-function API.

Since these functions are private to the source file, you can incorporate multiple instances of the same S-function into a model. Note that each such noninlined S-function also instantiates a `SimStruct`.

Conditional Compilation for Simulink and Real-Time

Noninlined S-functions must function in both Simulink and in real-time environments. The Real-Time Workshop defines the preprocessor symbols `MATLAB_MEX_FILE`, `RT`, and `NRT` to distinguish simulation code from real-time code. Use these symbols as follows:

• `MATLAB_MEX_FILE`

  Conditionally include code that is intended only for use in simulation under this symbol. When you build your S-function as a MEX-file via the `mex` command, `MATLAB_MEX_FILE` is automatically defined.

• `RT`

  Conditionally include code that is intended to run only in a real-time program under this symbol. When you generate code via the Real-Time Workshop build command, `RT` is automatically defined.

• `NRT`

  Conditionally include code that is intended only for use with a variable-step solver, in a non-real-time standalone simulation or in a MEX-file for use with Simulink, under this symbol.

Real-Time Workshop provides these conditionals to help ensure that your driver S-functions access hardware only when it is appropriate to do so. Since your target I/O hardware is not present during simulation, writing to addresses in the target environment can result in illegal memory references, overwriting system memory, and other severe errors. Similarly, read operations from nonexistent hardware registers can cause model execution errors.

In the following code fragment, a hardware initialization call is compiled in generated real-time code. During simulation, a message is printed to the MATLAB command window.

```
#if defined(RT)
  /* generated code calls function to initialize an A/D device */
  INIT_AD();
#elif defined(MATLAB_MEX_FILE)
  /* during simulation, just print a message */
  if (ssGetSimMode(S) == SS_SIMMODE_NORMAL) {
    mexPrintf("\n adc.c: Simulating initialization\n”);
  }
#endif
```

The `MATLAB_MEX_FILE` and `RT` conditionals also control the use of certain required include files. See “Required Defines and Include Files” below.

You may prefer to control execution of real-time and simulation code by some other means. For an example, see the use of the variable `ACCESS_HW` in *matlabroot*`/rtw/c/dos/devices/das16ad.c`

Required Defines and Include Files

Your driver S-function must begin with the following three statements, in the following order:

1 `#define S_FUNCTION_NAME` *name*

This defines the name of the entry point for the S-function code. *name* must be the name of the S-function source file, without the `.c` extension. For example, if the S-function source file is `das16ad.c`:

```
#define S_FUNCTION_NAME das16ad
```

2 `#define S_FUNCTION_LEVEL 2`

This statement defines the file as a level 2 S-function. This allows you to take advantage of the full feature set included with S-functions. Level-1 S-functions are currently used only to maintain backwards compatibility.

3 #include “simstruc.h”

The file simstruc.h defines the SimStruct (the Simulink data structure) and associated accessor macros. It also defines access methods for the mx* functions from the MATLAB MEX API.

Depending upon whether you intend to build your S-function as a MEX file or as real-time code, you must include one of the following files at the end of your S-function:

- simulink.c provides required functions interfacing to Simulink.
- cg_sfun.h provides the required S-function entry point for generated code.

A noninlined S-function should conditionally include both these files, as in the following code from sfuntmpl.c:

```
#ifdef  MATLAB_MEX_FILE   /* File being compiled as a MEX-file? */
#include "simulink.c"      /* MEX-file interface mechanism */
#else
#include "cg_sfun.h" /* Code generation registration function */
#endif
```

Required Functions

The S-function API requires you to implement several functions in your driver:

- mdlInitializeSizes specifies the sizes of various parameters in the SimStruct, such as the number of output ports for the block.
- mdlInitializeSampleTimes specifies the sample time(s) of the block.

  If your device driver block is masked, your initialization functions can obtain the sample time and other parameters entered by the user in the block’s dialog box.
- mdlOutputs: for an input device, reads values from the hardware and sets these values in the output vector y. For an output device, reads the input u from the upstream block and outputs the value(s) to the hardware.
- mdlTerminate resets hardware devices to a desired state, if any. This function may be implemented as a stub.

In addition to the above, you may want to implement the mdlStart function. mdlStart, which is called once at the start of model execution, is useful for operations such as setting I/O hardware to some desired initial state.

This following sections provide guidelines for implementing these functions.

mdlInitializeSizes

In this function you specify the sizes of various parameters in the SimStruct. This information may depend upon the parameters passed to the S-function. "Parameterizing Your Driver" on page 17-38 describes how to access parameter values specified in S-function dialog boxes.

Initializing Sizes - Input Devices. The mdlInitializeSizes function sets size information in the SimStruct. The following implementation of mdlInitializeSizes initializes a typical ADC driver block.

```
static void mdlInitializeSizes(SimStruct *S)
{
uint_T num_channels;

ssSetNumSFcnParams(S, 3); /* Number of expected parameters */
if (ssGetNumSFcnParams(S) != ssGetSFcnParamsCount(S)){
  /*Return if number of expected != number of actual params */
  return;
  }
num_channels = mxGetPr(NUM_CHANNELS_PARAM)[0];

ssSetNumInputPorts(S, 0);
ssSetNumOutputPorts(S, num_channels);
ssSetNumSampleTimes(S,1);
}
```

This routine first validates that the number of input parameters is equal to the number of parameters in the block's dialog box. Next, it obtains the **Number of Channels** parameter from the dialog.

ssSetNumInputPorts sets the number of input ports to 0 because an ADC is a source block, having only outputs.

ssSetNumOutputPorts sets the number of output ports equal to the number of I/O channels obtained from the dialog box.

ssSetNumSampleTimes sets the number of sample times to 1. This would be the case where all ADC channels run at the same rate. Note that the actual sample period is set in mdlInitializeSampleTimes.

Note that by default, the ADC block has no direct feedthrough. The ADC output is calculated based on values read from hardware, not from data obtained from another block.

Initializing Sizes - Output Devices. Initializing size information for an output device, such as a DAC, differs in several important ways from initializing sizes for an ADC:

- Since the DAC obtains its inputs from other blocks, the number of channels is equal to the number of inputs.
- The DAC is a sink block. That is, it has input ports but no output ports. Its output is written to a hardware device.
- The DAC block has direct feedthrough. The DAC block cannot execute until the block feeding it updates its outputs.

The following example is an implementation of `mdlInitializeSizes` for a DAC driver block.

```
static void mdlInitializeSizes(SimStruct *S)
{
uint_T num_channels;

ssSetNumSFcnParams(S, 3); /* Number of expected parameters */
if (ssGetNumSFcnParams(S) != ssGetSFcnParamsCount(S)){
  /* Return if number of expected != number of actual params */
  return;
  }
num_channels = mxGetPr(NUM_CHANNELS_PARAM)[0];
ssSetNumInputPorts(S, num_channels);
/* Number of inputs is now the number of channels. */
ssSetNumOutputPorts(S, 0);
/* Set direct feedthrough for all ports */
  {
  uint_T i;
  for(i=0, i < num_channels, i++) {
    ssSetInputPortDirectFeedThrough(S,i,1);
    }
  }
ssSetNumSampleTimes(S, 1);
}
```

mdlInitializeSampleTimes

Device driver blocks are discrete blocks, requiring you to set a sample time. The procedure for setting sample times is the same for both input and output device drivers. Assuming that all channels of the device run at the same rate, the S-function has only one sample time.

The following implementation of `mdlInitializeSampleTimes` reads the sample time from a block's dialog box. In this case, sample time is the fifth parameter in the dialog box. The sample time offset is set to 0.

```
static void mdlInitializeSampleTimes(SimStruct *S)
{
ssSetSampleTime(S, 0, mxGetPr(ssGetSFcnParams(S,4))[0]);
ssSetOffsetTime(S, 0, 0.0);
}
```

mdlStart

`mdlStart` is an optional function. It is called once at the start of model execution, and is often used to initialize hardware. Since it accesses hardware, you should compile it conditionally for use in real-time code or simulation, as in this example:

```
static void mdlStart(SimStruct *S)
{
#if defined(RT)
  /* Generated code calls function to initialize an A/D device */
  INIT_AD(); /* This call accesses hardware */
#elif defined(MATLAB_MEX_FILE)
  /* During simulation, just print a message */
  if (ssGetSimMode(S) == SS_SIMMODE_NORMAL) {
    mexPrintf("\n adc.c: Simulating initialization\n");
    }
#endif
}
```

mdlOutputs

The basic purpose of a device driver block is to allow your program to communicate with I/O hardware. Typically, you accomplish this by using low level hardware calls that are part of your compiler's C library, or by using C-callable functions provided with your I/O hardware.

All S-functions implement a mdlOutputs function to calculate block outputs. For a device driver block, mdlOutputs contains the code that reads from or writes to the hardware.

mdlOutputs - Input Devices. In a driver for an input device (such as an ADC), mdlOutputs must:

- Initiate a conversion for each channel.
- Read the board's ADC output for each channel (and perhaps apply scaling to the values read).
- Set these values in the output vector y for use by the model.

The following code is the mdlOutputs function from the ADC driver *matlabroot*/rtw/c/dos/devices/das16ad.c. The function uses macros defined in *matlabroot*/rtw/c/dos/devices/das16ad.h to perform low-level hardware access. Note that the Boolean variable ACCESS_HW (rather than conditional compilation) controls execution of simulation and real-time code. The real-time code reads values from the hardware and stores them to the output vector. The simulation code simply outputs 0 on all channels.

```
static void mdlOutputs(SimStruct *S, int_T tid)
{
real_T *y = ssGetOutputPortRealSignal(S,0);
uint_T  i;
if (ACCESS_HW) {
    /* Real-time code reads hardware*/
    ADCInfo   *adcInfo    = ssGetUserData(S);
    uint_T    baseAddr    = adcInfo->baseAddr;
    real_T    offset      = adcInfo->offset;
    real_T    resolution  = adcInfo->resolution;
    /* For each ADC channel initiate conversion,*/
    /* then read channel value, scale and offset it and store */
    /* it to output y */
    for (i = 0; i < NUM_CHANNELS; i++) {
      uint_T adcValue;
      adcStartConversion(baseAddr);
      for ( ;  ; ){
        if (!adcIsBusy(baseAddr)) break;
        }
      adcValue = adcGetValue(baseAddr);
```

```
        y[i] = offset + resolution*adcValue;
        }
      }
  else {
    /* simulation code just zeroes the output for all channels*/
    for (i = 0; i < NUM_CHANNELS; i++){
      y[i] = 0.0;
      }
    }
  }
```

mdlOutputs - Output Devices. In a driver for an output device (such as a DAC), `mdlOutputs` must:

- Read the input `u` from the upstream block.
- Set the board's DAC output for each channel (and apply scaling to the input values if necessary).
- Initiate a conversion for each channel.

The following code is the `mdlOutputs` function from the DAC driver *matlabroot*`/rtw/c/dos/devices/das16da.c`. The function uses macros defined in *matlabroot*`/rtw/c/dos/devices/das16ad.h` to perform low-level hardware access. This function iterates over all channels, obtaining and scaling a block input value. It then range-checks and (if necessary) trims each value. Finally it writes the value to the hardware.

In simulation, this function is a stub.

```
static void mdlOutputs(SimStruct *S, int_T tid)
{
if (ACCESS_HW) {
  int_T             i;
  DACInfo           *dacInfo   = ssGetUserData(S);
  uint_T            baseAddr   = dacInfo->baseAddr;
  real_T            resolution = dacInfo->resolution;
  InputRealPtrsType uPtrs  = ssGetInputPortRealSignalPtrs(S,0);
  for (i = 0; i < NUM_CHANNELS; i++) {
    uint_T codeValue;
    /* Get and scale input for channel i. */
    real_T value = (*uPtrs[i] - MIN_OUTPUT)*resolution;
    /* Range check value */
```

```
        value = (value < DAC_MIN_OUTPUT) ? DAC_MIN_OUTPUT : value;
        value = (value > DAC_MAX_OUTPUT) ? DAC_MAX_OUTPUT : value;
        codeValue = (uint_T) value;
        /* Output to hardware */
        switch (i) {
    case 0:
            dac0SetValue(baseAddr, codeValue);
            break;
          case 1:
            dac1SetValue(baseAddr, codeValue);
            break; }
        }
      }
    }
```

mdlTerminate

This final required function is typically needed only in DAC drivers. The following routine sets the output of each DAC channel to zero:

```
static void mdlTerminate(SimStruct *S)
{
uint_T num_channels;
uint_T i;

num_channels = (uint_t)mxGetPr(ssGetSFcnParams(S,0)[0]);
for (i = 0; i < num_channels; i++){
  ds1102_da(i + 1, 0.0); /* Hardware-specific DAC output */
  }
}
```

ADC drivers usually implement `mdlTerminate` as an empty stub.

Writing an Inlined S-Function Device Driver

Overview

To inline a device driver, you must provide:

- *driver*`.c`: C MEX S-function source code, implementing the functions required by the S-function API. These are the same functions required for noninlined drivers, as described in “Required Functions” on page 17-42. For these functions, only the code for simulation in Simulink simulation is required.

It is important to ensure that *driver*`.c` does not attempt to read or write memory locations that are intended to be used in the target hardware environment. The real-time driver implementation, generated via a *driver*`.tlc` file, should access the target hardware.

- Any hardware support files such as header files, macro definitions, or code libraries that are provided with your I/O devices.
- Optionally, a `mdlRTW` function within *driver*`.c`. The sole purpose of this function is to evaluate and format parameter data during code generation. The parameter data is output to the *model*`.rtw` file. If your driver block does not need to pass information to the code generation process, you do not need to write a `mdlRTW` function. See “mdlRTW and Code Generation” on page 17-52 .
- *driver*`.dll` (PC) or *driver* (UNIX): MEX-file built from your C MEX S-function source code. This component is used:
  - In simulation: Simulink calls the simulation versions of the required functions
  - During code generation: if a `mdlRTW` function exists in the MEX-file, the code generator executes it to write parameter data to the *model*`.rtw` file.
- *driver*`.tlc`: TLC functions that generate real-time implementations of the functions required by the S-function API.

Example: An Inlined ADC Driver

As an aid to understanding the process of inlining a device driver, this section describes an example driver block for an ADC device. “Source Code for Inlined ADC Driver” on page 17-55 lists code for:

- `adc.c`, the C MEX S-function
- `adc.tlc`, the corresponding TLC file
- `device.h`, a hardware-specific header file included in both the simulation and real-time generated code

The `driver` S-Function block is masked and has an icon. Figure 17-8 shows a model using the `driver` S-Function block. Figure 17-9 shows the block’s dialog box.

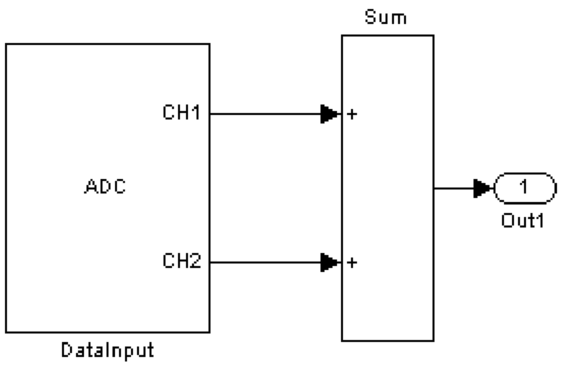
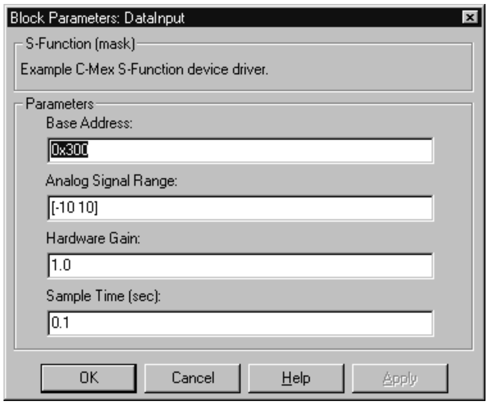


Figure 17-8: ADC S-function Driver Block in a Model

The dialog box lets the user enter:

- The ADC base address
- An array defining its signal range
- Its gain factor
- The block's sample time


Block Parameters: DataInput

S-Function (mask)

Example C-Mex S-Function device driver.

Parameters

Base Address: 0x300

Analog Signal Range: [-10 10]

Hardware Gain: 1.0

Sample Time (sec): 0.1

Figure 17-9: ADC Driver Dialog Box

Simulation Code. `adc.c` consists almost entirely of functions to be executed during simulation. (The sole exception is `mdlRTW`, which executes during code generation.) Most of these functions are similar to the examples of non-real-time code given in "Writing a Noninlined S-Function Device Driver" on page 17-39. The S-function implements the following functions:

- `mdlInitializeSizes` validates input parameters (via `mdlCheckParameters`) and declares all parameters nontunable. This function also initializes ports and sets the number of sample times.
- `mdlInitializeSampleTimes` sets the sample time using the user-entered value.
- `mdlStart` prints a message to the MATLAB command window.
- `mdlOutputs` outputs zero on all channels.
- `mdlTerminate` is a stub routine.

Since `adc.c` contains only simulation code, it uses a single test of `MATLAB_MEX_FILE` to ensure that it is compiled as a C MEX-file.

```
#ifndef MATLAB_MEX_FILE
#error "Fatal Error: adc.c can only be used to create C-MEX S-Function"
#endif
```

For the same reason, `adc.c` unconditionally includes `simulink.c`.

mdlRTW and Code Generation. mdlRTW is a mechanism by which an S-function can generate and write data structures to the *model*.rtw file. The Target Language Compiler, in turn, uses these data structures when generating code. Unlike the other functions in the driver, mdlRTW executes at code generation time.

In this example, mdlRTW calls the ssWriteRTWParamSettings function to generate a structure that contains both user-entered parameter values (base address, hardware gain) and values computed from user-entered values (resolution, offset).

```
static void mdlRTW(SimStruct *S)
{
    boolean_T polarity   = adcIsUnipolar(MIN_SIGNAL_VALUE, MAX_SIGNAL_VALUE);
    real_T    offset     = polarity ? 0.0 : MIN_SIGNAL_VALUE/HARDWARE_GAIN;
    real_T    resolution = (((MAX_SIGNAL_VALUE-MIN_SIGNAL_VALUE)/HARDWARE_GAIN)/
                            ADC_NUM_LEVELS);
    char_T    baseAddrStr[128];

    if ( mxGetString(BASE_ADDRESS_PARAM, baseAddrStr, 128) ) {
        ssSetErrorStatus(S, "Error reading Base Address parameter, "
                         "need to increase string buffer size.");
        return;
    }

    if ( !ssWriteRTWParamSettings(S, 4,
                       SSWRITE_VALUE_QSTR, "BaseAddress",  baseAddrStr,
                       SSWRITE_VALUE_NUM,  "HardwareGain", HARDWARE_GAIN,
                       SSWRITE_VALUE_NUM,  "Resolution",   resolution,
                       SSWRITE_VALUE_NUM,  "Offset",       offset) ) {

        return; /* An error occured, which will be reported by Simulink. */
    }
} /* end: mdlRTW */
```

The structure defined in *model*.rtw is

```
SFcnParamSettings {
 BaseAddress "0x300"
 HardwareGain 1.0
 Resolution 0.0048828125
 Offset -10.0
}
```

(The actual values of SFcnParamSettings derive from data entered by the user.)

Values stored in the SFcnParamSettings structure are referenced in *driver*.tlc, as in the following assignment statement.

```
%assign baseAddr = SFcnParamSettings.BaseAddress
```

The Target Language Compiler uses variables such as baseAddr to generate parameters in real-time code files such as *model*.c and *model*.h. This is discussed in the next section.

The TLC File. adc.tlc contains three TLC functions. The BlockTypeSetup function generates the statement

```
#include "device.h"
```

in the *model*.h file. The other two functions, Start and Outputs, generate code within the MdlStart and MdlOutputs functions of *model*.c.

Statements in adc.tlc, and in the generated code, employ macros and symbols defined in device.h, and parameter values in the SFcnParamSettings structure. The following code uses the values from the SFcnParamSettings structure above to generate code containing constant values:

```
%assign baseAddr = SFcnParamSettings.BaseAddress
%assign hwGain = SFcnParamSettings.HardwareGain
...
adcSetHardwareGain(%<baseAddr>, adcGetGainMask(%<hwGain>));
```

The TLC code above generates this statement in the MdlOutputs function of *model*.c.

```
adcSetHardwareGain(0x300, adcGetGainMask(1.0));
```

`adcSetHardwareGain` and `adcGetGainMask` are macros that expand to low-level hardware calls.

S-Function Wrappers

Another technique for integrating driver code into your target system is to use S-function wrappers. In this approach, you write:

- An S-function (the wrapper) that calls your driver code as an external module
- A TLC file that generates a call to the same driver code that was called in the wrapper

See *Writing S-Functions* for a full description of how to use wrapper S-functions.

Building the MEX-File and the Driver Block

This section outlines how to build a MEX-file from your driver source code for use in Simulink. For full details on how to use `mex` to compile the device driver S-function into an executable MEX-file, see "External Interfaces" in the MATLAB online documentation. For details on masking the device driver block, see "Using Masks to Customize Blocks" in *Using Simulink*.

1 Your C S-function source code should be in your working directory. To build a MEX-file from *mydriver*`.c` type

   ```
   mex mydriver.c
   ```

   `mex` builds *mydriver*`.dll` (PC) or *mydriver* (UNIX).

2 Add an S-Function block (from the Simulink Functions & Tables library in the Library Browser) to your model.

3 Double-click the S-Function block to open the **Block Parameters** dialog. Enter the S-function name *mydriver*. The block is now bound to the *mydriver* MEX-file.

4 Create a mask for the block if you want to use a custom icon or dialog.

Source Code for Inlined ADC Driver

These files are described in “Example: An Inlined ADC Driver” on page 17-49.

adc.c

```
/*
 * File    : adc.c
 * Abstract:
 * Example S-function device driver (analog to digital convertor) for use
 * with Simulink and Real-Time Workshop.
 * This S-function contains simulation code only (except mdlRTW, used
 * only during code generation.) An error will be generated if
 * this code is compiled without MATLAB_MEX_FILE defined. That
 * is,it must be compiled via the MATLAB mex utility.
 *
 * DEPENDENCIES:
 * (1) This S-function is intended for use in conjunction with adc.tlc,
 * a Target Language Compiler program that generates inlined, real-time code that
 * implements the real-time I/O functions required by mdlOutputs, etc.
 *
 * (2) device.h defines hardware-specific macros, etc. that implement
 * actual I/O to the board
 *
 * (3) This file contains a mdlRTW function that writes parameters to
 * the model.rtw file during code generation.
 *
 * Copyright (c) 1994-2000 by The MathWorks, Inc. All Rights Reserved.
 *
 */

/*********************
 * Required  defines *
 *********************/

#define S_FUNCTION_NAME adc
#define S_FUNCTION_LEVEL 2

/*********************
 * Required includes *
 *********************/

#include "simstruc.h"    /* The Simstruct API, definitions and macros */

/*
 * Generate a fatal error if this file is (by mistake) used by Real-Time
 * Workshop. There is a  target file corresponding to this S-function: adc.tlc,
 * which should be used to generate inlined code for this S-funciton.
 */
#ifndef MATLAB_MEX_FILE
# error "Fatal Error: adc.c can only be used to create C-MEX S-Function"
#endif
```

```
/*
 * Define the number of S-function parameters and set up convenient macros to
 * access the parameter values.
 */
#define  NUM_S_FUNCTION_PARAMS      (4)
#define N_CHANNELS (2) /* For this example, num. of channels is fixed */

/* 1. Base Address */
#define  BASE_ADDRESS_PARAM         (ssGetSFcnParam(S,0))

/* 2. Analog Signal Range */
#define  SIGNAL_RANGE_PARAM         (ssGetSFcnParam(S,1))
#define  MIN_SIGNAL_VALUE           ((real_T) (mxGetPr(SIGNAL_RANGE_PARAM)[0]))
#define  MAX_SIGNAL_VALUE           ((real_T) (mxGetPr(SIGNAL_RANGE_PARAM)[1]))

/* 3. Hardware Gain */
#define  HARDWARE_GAIN_PARAM        (ssGetSFcnParam(S,2))
#define  HARDWARE_GAIN              ((real_T) (mxGetPr(HARDWARE_GAIN_PARAM)[0]))

/* 4. Sample Time */
#define  SAMPLE_TIME_PARAM          (ssGetSFcnParam(S,3))
#define  SAMPLE_TIME                ((real_T) (mxGetPr(SAMPLE_TIME_PARAM)[0]))


/*
 * Hardware specific information pertaining to the A/D board. This information
 * should be provided with the documentation that comes with the board.
 */
#include "device.h"

/*====================*
 * S-function methods *
 *====================*/


/* Function: mdlCheckParameters ================================================
 * Abstract:
 *      Check that the parameters passed to this S-function are valid.
 */
#define MDL_CHECK_PARAMETERS
static void mdlCheckParameters(SimStruct *S)
{
    static char_T errMsg[256];
    boolean_T allParamsOK = 1;

    /*
     * Base I/O Address
     */
    if (!mxIsChar(BASE_ADDRESS_PARAM)) {
        sprintf(errMsg, "Base address parameter must be a string.\n");
        allParamsOK = 0;
        goto EXIT_POINT;
    }
    /*
     * Signal Range
     */
    if (mxGetNumberOfElements(SIGNAL_RANGE_PARAM) != 2) {
        sprintf(errMsg,
               "Signal Range must be a two element vector [minInp maxInp]\n");
        allParamsOK = 0;
        goto EXIT_POINT;
    }
    if ( !adcIsSignalRangeParamOK(MIN_SIGNAL_VALUE, MAX_SIGNAL_VALUE) ) {
        sprintf(errMsg,
               "The specified Signal Range is not supported by I/O board.\n");
        allParamsOK = 0;
        goto EXIT_POINT;
    }
    /*
     * Hardware Gain
     */
    if (mxGetNumberOfElements(HARDWARE_GAIN_PARAM) != 1) {
        sprintf(errMsg, "Hardware Gain must be a scalar valued real number\n");
        allParamsOK = 0;
        goto EXIT_POINT;
    }
    if (!adcIsHardwareGainParamOK(HARDWARE_GAIN)) {
        sprintf(errMsg, "The specified hardware gain is not supported.\n");
        allParamsOK = 0;
        goto EXIT_POINT;
    }

    /*
     * Sample Time
     */
    if (mxGetNumberOfElements(SAMPLE_TIME_PARAM) != 1) {
        sprintf(errMsg, "Sample Time must be a positive scalar.\n");
        allParamsOK = 0;
        goto EXIT_POINT;
    }
EXIT_POINT:
    if ( !allParamsOK ) {
        ssSetErrorStatus(S, errMsg);
    }

} /* end: mdlCheckParameters */



/* Function: mdlInitializeSizes ===============================================
 * Abstract:
 * Validate parameters,set number and width of ports.
 */
static void mdlInitializeSizes(SimStruct *S)
{
    /* Set the number of parameters expected. */
    ssSetNumSFcnParams(S, NUM_S_FUNCTION_PARAMS);
    if ( ssGetNumSFcnParams(S) == ssGetSFcnParamsCount(S) ) {
        /*
         * If the number of parameter passed in is equal to the number of
         * parameters expected, then check that the specified parameters
         * are valid.
         */
        mdlCheckParameters(S);
        if ( ssGetErrorStatus(S) != NULL ) {
            return; /* Error was reported in mdlCheckParameters. */
        }
    } else {
        return; /* Parameter mismatch. Error will be reported by Simulink. */
    }

    /*
     * This S-functions's parameters cannot be changed in the middle of a
     * simulation, hence set them to be nontunable.
     */
    {
        int_T i;
        for (i=0; i < NUM_S_FUNCTION_PARAMS; i++) {
            ssSetSFcnParamNotTunable(S, i);
        }
    }

    /* Has no input ports */
    if ( !ssSetNumInputPorts(S, 0) ) return;

    /* Number of output ports = number of channels specified */
    if ( !ssSetNumOutputPorts(S, N_CHANNELS) ) return;

    /* Set the width of each output ports to be one. */
    {
        int_T oPort;
        for (oPort = 0; oPort < ssGetNumOutputPorts(S); oPort++) {
            ssSetOutputPortWidth(S, oPort, 1);
        }
    }

    ssSetNumSampleTimes( S, 1);

} /* end: mdlInitializeSizes */



/* Function: mdlInitializeSampleTimes ==========================================
 * Abstract:
 *      Set the sample time of this block as specified via the sample time
 *      parameter.
 */
static void mdlInitializeSampleTimes(SimStruct *S)
```

```
{
    ssSetSampleTime(S, 0, SAMPLE_TIME);
    ssSetOffsetTime(S, 0, 0.0);

} /* end: mdlInitializeSampleTimes */


/* Function: mdlStart ==========================================================
 * Abstract:
 *      At the start of simulation in Simulink, print a message to the MATLAB
 *      command window indicating that output of this block will be zero during
 *      simulation.
 */
#define MDL_START
static void mdlStart(SimStruct *S)
{
    if (ssGetSimMode(S) == SS_SIMMODE_NORMAL) {
        mexPrintf("\n adc.c: The output of the A/D block '%s' will be set "
                  "to zero during simulation in Simulink.\n", ssGetPath(S));
    }
} /* end: mdlStart */


/* Function: mdlOutputs ========================================================
 * Abstract:
 *      Set the output to zero.
 */
static void mdlOutputs(SimStruct *S, int_T tid)
{
    int oPort;

    for (oPort = 0; oPort < ssGetNumOutputPorts(S); oPort++) {
        real_T *y = ssGetOutputPortRealSignal(S, oPort);
        y[0] = 0.0;

    }
} /* end: mdlOutputs */


/* Function: mdlTerminate ======================================================
 * Abstract:
 *      Required S-function method that gets called at the end of simulation
 *      and code generation. Nothing to do in simulation.
 */
static void mdlTerminate(SimStruct *S)
{
} /* end: mdlTerminate */



/* Function: mdlRTW ============================================================
 * Abstract:
 *      Evaluate parameter data and write it to the model.rtw file.
 */
#define MDL_RTW
static void mdlRTW(SimStruct *S)
{
    boolean_T polarity   = adcIsUnipolar(MIN_SIGNAL_VALUE, MAX_SIGNAL_VALUE);
    real_T    offset     = polarity ? 0.0 : MIN_SIGNAL_VALUE/HARDWARE_GAIN;
    real_T    resolution = (((MAX_SIGNAL_VALUE-MIN_SIGNAL_VALUE)/HARDWARE_GAIN)/
                             ADC_NUM_LEVELS);
    char_T    baseAddrStr[128];

    if ( mxGetString(BASE_ADDRESS_PARAM, baseAddrStr, 128) ) {
        ssSetErrorStatus(S, "Error reading Base Address parameter, "
                         "need to increase string buffer size.");
        return;
    }

    if ( !ssWriteRTWParamSettings(S, 4,
                       SSWRITE_VALUE_QSTR, "BaseAddress",  baseAddrStr,
                       SSWRITE_VALUE_NUM,  "HardwareGain", HARDWARE_GAIN,
                       SSWRITE_VALUE_NUM,  "Resolution",   resolution,
                       SSWRITE_VALUE_NUM,  "Offset",       offset) ) {

        return; /* An error occured, which will be reported by Simulink. */
    }
} /* end: mdlRTW */


/*
 * Required include for Simulink-MEX interface mechanism
 */
#include "simulink.c"
/* EOF: adc.c */
```

adc.tlc

```
%% File    : adc.tlc
%% Abstract:
%%     Target file for the C-Mex S-function adc.c
%%
%% Copyright (c) 1994-2000 by The MathWorks, Inc. All Rights Reserved.
%%

%implements "adc" "C"

%% Function: BlockTypeSetup ==========================================
%% Abstract:
%%     This function is called once for all instance of the S-function
%% "dac" in the model. Since this block requires hardware specific
%% information about the I/O board, we generate code to include
%% "device.h" in the generated model.h file.
%%
%function BlockTypeSetup(block, system) void
```

```
  %%
  %% Use the Target Language Ccompiler global variable INCLUDE_DEVICE_H to make sure that
  %% the line "#include device.h" gets generated into the model.h
  %%file only once.
  %%
  %if !EXISTS("INCLUDE_DEVICE_H")
    %assign ::INCLUDE_DEVICE_H = 1
    %openfile buffer
    /* Include information about the I/O board */
    #include "device.h"
    %closefile buffer
    %<LibCacheIncludes(buffer)>
  %endif

%endfunction %% BlockTypeSetup


%% Function: Start ====================================================
%% Abstract:
%%      Generate code to set the number of channels and the hardware gain
%% mask in the start function.
%%
%function Start(block, system) Output
  /* %<Type> Block: %<Name> (%<ParamSettings.FunctionName>) */
  %%
  %assign numChannels = block.NumDataOutputPorts
  %assign baseAddr    = SFcnParamSettings.BaseAddress
  %assign hwGain      = SFcnParamSettings.HardwareGain
  %%
  %% Initialize the Mux Scan Register to scan from 0 to NumChannels-1.
  %% Also set the Gain Select Register to the appropriate value.
  %%
  adcSetLastChannel(%<baseAddr>, %<numChannels-1>);
  adcSetHardwareGain(%<baseAddr>, adcGetGainMask(%<hwGain>));

%endfunction %% Start


%% Function: Outputs =================================================
%% Abstract:
%%      Generate inlined code to perform one A/D conversion on the enabled
%%      channels.
%%
%function Outputs(block, system) Output
  %%
  %assign offset     = SFcnParamSettings.Offset
  %assign resolution = SFcnParamSettings.Resolution
  %assign baseAddr   = SFcnParamSettings.BaseAddress
  %%
  /* %<Type> Block: %<Name> (%<ParamSettings.FunctionName>) */
  {
    int_T  chIdx;
    uint_T adcValues[%<NumDataOutputPorts>];
    for (chIdx = 0; chIdx < %<NumDataOutputPorts>; chIdx++) {
      adcStartConversion(%<baseAddr>);
      while (adcIsBusy(%<baseAddr>)) {
      /* wait for conversion */
      }
      adcValues[chIdx] = adcGetValue(%<baseAddr>);
    }

    %foreach oPort = NumDataOutputPorts
      %assign y = LibBlockOutputSignal(oPort, "", "", 0)
      %<y> = %<offset> + %<resolution>*adcValues[%<oPort>];
    %endforeach
  }

%endfunction %% Outputs
%% EOF: adc.tlc
```

device.h

```
/*
* File    : device.h
*
* Copyright (c) 1994-2000 by The MathWorks, Inc. All Rights
* Reserved.
*
*/

/*
* Operating system utilities to read and write to hardware
* registers.
*/
#define  ReadByte(addr)        inp(addr)
#define  WriteByte(addr,val)   outp(addr,val)


/*============================================================*
* Specification of the Analog  Input Section of the I/O board
* (used in the ADC device driver S-function, adc.c and  *adc.tlc)
*========================================================*/

/*
* Define macros for the attributes of the A/D board, such as the
* number of A/D channels and bits per channel.
*/
#define  ADC_MAX_CHANNELS              (16)
#define  ADC_BITS_PER_CHANNEL          (12)
#define  ADC_NUM_LEVELS ((uint_T) (1 << ADC_BITS_PER_CHANNEL))

/*
* Macros to check if the specified parameters are valid.
* These macros are used by the C-Mex S-function, adc.c
```

```
*/
#define  adcIsUnipolar(lo,hi)          (lo      == 0.0 && 0.0 < hi)
#define  adcIsBipolar(lo,hi)           (lo + hi == 0.0 && 0.0 < hi)
#define  adcIsSignalRangeParamOK(l,h)  (adcIsUnipolar(l,h) || adcIsBipolar(l,h))

#define  adcGetGainMask(g) ( (g==1.0) ? 0x0 : \
                            ( (g==10.0) ? 0x1 : \
                             ( (g==100.0) ? 0x2 : \
                               ( (g==500.0) ? 0x3 : 0x4 ) ) ) )
#define  adcIsHardwareGainParamOK(g)   (adcGetGainMask(g) != 0x4)
#define  adcIsNumChannelsParamOK(n)    (1 <= n && n <= ADC_MAX_CHANNELS)

/* Hardware registers used by the A/D section of the I/O board */

#define  ADC_START_CONV_REG(bA)     (bA)
#define  ADC_LO_BYTE_REG(bA)        (bA)
#define  ADC_HI_BYTE_REG(bA)        (bA + 0x1)
#define  ADC_MUX_SCAN_REG(bA)       (bA + 0x2)
#define  ADC_STATUS_REG(bA)         (bA + 0x8)
#define  ADC_GAIN_SELECT_REG(bA)    (bA + 0xB)

/*
* Macros for the A/D section of the I/O board
*/
#define  adcSetLastChannel(bA,n) WriteByte(ADC_MUX_SCAN_REG(bA), n<<4)
#define  adcSetHardwareGain(bA,gM) WriteByte(ADC_GAIN_SELECT_REG(bA), gM)
#define  adcStartConversion(bA) WriteByte(ADC_START_CONV_REG(bA), 0x00)
#define  adcIsBusy(bA) (ReadByte(ADC_STATUS_REG(bA)) & 0x80)
#define  adcGetLoByte(bA) ReadByte(ADC_LO_BYTE_REG(bA))
#define  adcGetHiByte(bA) ReadByte(ADC_HI_BYTE_REG(bA))
#define  adcGetValue(bA) ((adcGetLoByte(bA)>>4) | (adcGetHiByte(bA)<<4))

/*=============================================================*
* Specification of the Analog Output Section of the I/O board
* (used in the DAC device driver S-function, adc.c and adc.tlc)
*=============================================================*/

#define DAC_BITS_PER_CHANNEL  (12)
#define DAC_UNIPOLAR_ZERO     ( 0)
#define DAC_BIPOLAR_ZERO      (1 << (DAC_BITS_PER_CHANNEL-1))
#define DAC_MIN_OUTPUT        (0.0)
#define DAC_MAX_OUTPUT        ((real_T) ((1 << DAC_BITS_PER_CHANNEL)-1))
#define DAC_NUM_LEVELS        ((uint_T) (1 << DAC_BITS_PER_CHANNEL))

/*
* Macros to check if the specified parameters are valid.
* These macros are used by the C-Mex S-function,dac.c.
*/
#define dacIsUnipolar(lo,hi)         (lo    == 0.0 && 0.0 < hi)
#define dacIsBipolar(lo,hi)          (lo+hi == 0.0 && 0.0 < hi)
#define dacIsSignalRangeParamOK(l,h) (dacIsUnipolar(l,h) || dacIsBipolar(l,h))

/* Hardware registers */
```

```
#define DAC_LO_BYTE_REG(bA)     (bA + 0x4)
#define DAC_HI_BYTE_REG(bA)     (bA + 0x5)

#define dacSetLoByte(bA,c) WriteByte(DAC_LO_BYTE_REG(bA),(c & 0x00f)<<4)
#define dacSetHiByte(bA,c) WriteByte(DAC_HI_BYTE_REG(bA),(c & 0xff0)>>4)
#define dacSetValue(bA,c) dacSetLoByte(bA,c); dacSetHiByte(bA,c)

/* EOF: device.h */
```

Interfacing Parameters and Signals

Simulink external mode (see Chapter 5, "External Mode") offers a quick and easy way to monitor signals and modify parameter values while generated model code executes. However, external mode may not be appropriate for your application. Some targets (such as the Real-Time Workshop Embedded Coder) do not support external mode, due to optimizations. In other cases, you may want existing code to access parameters and signals of a model directly, rather than using the external mode mechanism.

The Real-Time Workshop supports several approaches to the task of interfacing block parameters and signals to your hand-written code.

The **Workspace Parameter Attributes** dialog enables you to declare how to the generated code allocates memory for variables used in your model. This allows your supervisory software to read or write block parameter variables as your model executes. Similarly, the **Signal Properties** dialog gives your code access to selected signals within your model. Operation of these dialogs is described in "Parameters: Storage, Interfacing, and Tuning" on page 3-51 and "Signals: Storage, Optimization, and Interfacing" on page 3-65.

In addition, the MathWorks provides C and Target Language Compiler APIs that give your code additional access to block outputs, and parameters that are stored in global data structures created by the Real-Time Workshop. This section is an overview of these APIs. This section also includes pointers to additional detailed API documents shipped with the Real-Time Workshop.

Signal Monitoring via Block Outputs

All block output data is written to the block outputs structure with each time step in the model code. To access the output of a given block in the block outputs structure, your code must have the following information, per port:

- The address of the `rtB` structure where the data is stored
- The number of output ports of the block
- The width of each output
- The data type of the signal

This information is contained in the `BlockIOSignals` data structure. The TLC code generation variable, `BlockIOSignals`, determines whether `BlockIOSignals` data is generated. If `BlockIOSignals` is enabled, a file containing an array of `BlockIOSignals` structures is written during code generation. This file is named *model*`_bio.c`.

`BlockIOSignals` is disabled by default. To enable generation of *model*`_bio.c`, use a `%assign` statement in the **Configure RTW code generation settings** section of your system target file:

```
%assign BlockIOSignals = 1
```

Alternatively, you can append the following command to the **System target file** field on the **Target configuration** section of the Real-Time Workshop page.

```
-aBlockIOSignals=1
```

Note that, depending on the size of your model, the `BlockIOSignals` array can consume a considerable amount of memory.

BlockIOSignals and the Local Block Outputs Option

When the **Local block outputs** code generation option is selected, block outputs are declared locally in functions instead of being declared globally in the `rtB` structure (when possible). The `BlockIOSignals` array in *model*`_bio.c` will not contain information about such locally declared signals. (Note that even when all outputs in the system are declared locally, enabling `BlockIOSignals` will generate *model*`_bio.c`. In such a case the `BlockIOSignals` array will contain only a null entry.)

Signals that are designated as test points via the **Signal Properties** dialog are declared globally in the `rtB` structure, even when the **Local block outputs** option is selected. Information about test-pointed signals is therefore written to the `BlockIOSignals` array in *model*`_bio.c`.

Therefore, you can interface your code to selected signals by test-pointing them and enabling `BlockIOSignals`, without losing the benefits of the **Local block outputs** optimization for the other signals in your model.

model_bio.c and the BlockIO Data Structure

The `BlockIOSignals` data structure is declared as follows.

```
typedef struct BlockIOSignals_tag {
  char_T *blockName;  /* Block's full pathname
                         (mangled by the Real-Time Workshop) */
  char_T *signalName; /* Signal label (unmangled) */
  uint_T portNumber;  /* Block output port number (start at 0) */
  uint_T signalWidth; /* Signal's width */
  void *signalAddr;   /* Signal's address in the rtB vector */
  char_T *dtName;     /* The C language data type name */
  uint_T dtSize;      /* The size (# of bytes) for the data type*/
} BlockIOSignals;
```

The structure definition is in *matlabroot*`/rtw/c/src/bio_sig.h`. The *model*`_bio.c` file includes `bio_sig.h`. Any source file that references the array should also include `bio_sig.h`.

model`_bio.c` defines an array of `BlockIOSignals` structures. Each array element, except the last, describes one output port for a block. The final element is a sentinel, with all fields set to null values.

The code fragment below is an example of an array of `BlockIOSignals` structures from a *model*`_bio.c` file.

```
#include "bio_sig.h"
/* Block output signal information */
const BlockIOSignals rtBIOSignals[] =
  {
  /* blockName,
     signalName, portNumber, signalWidth, signalAddr,
     dtName, dtSize */
  {
    "simple/Constant",
    NULL, 0, 1, &rtB.Constant,
    "double", sizeof(real_T)
  },
  {
    "simple/Constant1",
    NULL, 0, 1, &rtB.Constant1,
    "double", sizeof(real_T)
  },
  {
    "simple/Gain",
    "accel", 0, 2, &rtB.accel[0],
    "double", sizeof(real_T)
  },
  {
    NULL, NULL, 0, 0, 0, NULL, 0
  }
};
```

Thus, a given block will have as many entries as it has output ports. In the above example, the `simple/Gain` structure has a signal named `accel` on block output port 0. The width of the signal is 2.

Using BlockIOSignals to Obtain Block Outputs

The *model*`_bio.c` array is accessed via the name `rtBIOSignals`. To avoid overstepping array bounds, you can do either of the following:

- Use the SimStruct access macro ssGetNumBlockIO to determine the number of elements in the array.
- Test for a null blockName to identify the last element.

You must then write code that iterates over the rtBIOSignals array and chooses the signals to be monitored based on the blockName and signalName or portNumber. How the signals are monitored is up to you. For example, you could collect the signals at every time step. Alternatively, you could sample signals asynchronously in a separate, lower priority task.

The following code example is drawn from the main program (rt_main.c) of the Tornado target. The code illustrates how the StethoScope Graphical Monitoring/Data Analysis Tool uses BlockIOSignals to collect signal information in Tornado targets. The following function, rtInstallRemoveSignals, selectively installs signals from the BlockIOSignals array into the StethoScope Tool by calling ScopeInstallSignal. The main simulation task then collects signals by calling ScopeCollectSignals.

```
static int_T rtInstallRemoveSignals(SimStruct *S,
char_T *installStr, int_T fullNames, int_T install)
{
  uint_T                  i, w;
  char_T                 *blockName;
  char_T                 name[1024];
  extern BlockIOSignals  rtBIOSignals[];
  int_T                  ret = -1;

  if (installStr == NULL) {
    return -1;
  }

  i = 0;
  while(rtBIOSignals[i].blockName != NULL) {
    BlockIOSignals *blockInfo = &rtBIOSignals[i++];

    if (fullNames) {
      blockName = blockInfo->blockName;
    } else {
      blockName = strrchr(blockInfo->blockName, '/');
      if (blockName == NULL) {
        blockName = blockInfo->blockName;
      } else {
        blockName++;
      }
    }

    if ((*installStr) == '*') {
    } else if (strcmp("[A-Z]*", installStr) == 0) {
      if (!isupper(*blockName)) {
        continue;
      }
    } else {
      if (strncmp(blockName, installStr, strlen(installStr)) != 0) {
        continue;
      }
    }
    /*install/remove the signals*/
    for (w = 0; w < blockInfo->signalWidth; w++) {
      sprintf(name, "%s_%d_%s_%d", blockName, blockInfo->portNumber,
          (blockInfo->signalName==NULL)?"":blockInfo->signalName, w);
      if (install) { /*install*/
        if (!ScopeInstallSignal(name, "units",
                               (void *)((int)blockInfo->signalAddr +
                                           w*blockInfo->dtSize),
                                  blockInfo->dtName, 0)) {
              fprintf(stderr,"rtInstallRemoveSignals: ScopeInstallSignal "
                      "possible error: over 256 signals.\n");
              return -1;
          } else {
              ret =0;
          }
      } else { /*remove*/
        if (!ScopeRemoveSignal(name, 0)) {
          ifprintf(stderr,"rtInstallRemoveSignals: ScopeRemoveSignal\n"
                          "%s not found.\n",name);
            return -1;
          } else {
            ret =0;
        }
      }
    }
  }
  return ret;
}
```

Below is an excerpt from an example routine that collects signals taken from the main simulation loop.

```
/*******************************************
 * Step the model for the base sample time *
 *******************************************/
MdlOutputs(FIRST_TID);

#ifdef MAT_FILE
  if (rt_UpdateTXYLogVars(S) != NULL) {
    fprintf(stderr,"rt_UpdateTXYLogVars() failed\n");
    return(1);
  }
#endif

#ifdef STETHOSCOPE
  ScopeCollectSignals(0);
#endif

MdlUpdate(FIRST_TID);
<code continues ...>
```

See Chapter 12, “Targeting Tornado for Real-Time Applications” for more information on using StethoScope.

Parameter Tuning via model_pt.c

By enabling the TLC variable `ParameterTuning`, you can generate a file containing data structures that enable a running program to access model parameters without use of external mode. The file is named *model*`_pt.c`.

`ParameterTuning` is disabled by default. To enable generation of *model*`_pt.c`, use a `%assign` statement in your system target file.

```
%assign ParameterTuning = 1
```

Alternatively, you can append the following command to the **System target file** field on the **Target configuration** section of the Real-Time Workshop page.

```
-aParameterTuning=1
```

model`_pt.c` contains two parameter mapping structures containing information required for parameter tuning:

- The `BlockTuning` structure contains all the modifiable block parameters by block name and parameter name.
- The `VariableTuning` structure contains all the modifiable workspace variables that were specified in a Simulink parameter dialog box.

The structure and content of the *model*`_pt.c` file are documented in *matlabroot*`/rtw/c/src/pt_readme.txt`. For example source code using the parameter tuning API, see *matlabroot*`/rtw/c/src/pt_print.c`.

Target Language Compiler API for Signals and Parameters

The Real-Time Workshop provides a TLC function library that lets you create a *global data map record*. The global data map record, when generated, is added to the `CompiledModel` structure in the *model*`.rtw` file. The global data map record is a database containing all information required for accessing memory in the generated code, including:

- Signals (Block I/O)
- Parameters
- Data type work vectors (`DWork`)
- External inputs
- External outputs

Use of the global data map requires knowledge of the Target Language Compiler and of the structure of the *model*`.rtw` file. See the *Target Language Compiler Reference Guide* for information on these topics.

The TLC functions that are required to generate and access the global data map record are contained in *matlabroot*`/rtw/c/tlc/globalmaplib.tlc`. The comments in the source code fully document the global data map structures and the library functions.

Note The global data map structures and functions maybe modified and/or enhanced in future releases.

Creating an External Mode Communication Channel

This section provides information you will need in order to support external mode on your custom target, using your own low-level communications layer. This information includes:

- An overview of the design and operation of external mode
- A description of external mode source files
- Guidelines for modifying the external mode source files and rebuilding the `ext_comm` MEX-file

This section assumes that you are familiar with the execution of Real-Time Workshop programs, and with the basic operation of external mode. These topics are described in Chapter 6, “Program Architecture” and Chapter 5, “External Mode.”

The Design of External Mode

External mode communication between Simulink and a target system is based on a client/server architecture. The client (Simulink) transmits messages requesting the server (target) to accept parameter changes or to upload signal data. The server responds by executing the request.

A low-level *transport layer* handles physical transmission of messages. Both Simulink and the model code are independent of this layer. Both the transport layer and code directly interfacing to the transport layer are isolated in separate modules that format, transmit and receive messages and data packets.

This design makes it possible for different targets to use different transport layers. For example, the `grt`, `grt_malloc`, and Tornado targets support host/target communication via TCP/IP, whereas the xPC Target supports both RS232 (serial) and TCP/IP communication.

The Real-Time Workshop provides full source code for both the client and server-side external mode modules, as used by the `grt`, `grt_malloc`, and Tornado targets. The main client-side module is `ext_comm.c`. The main server-side module is `ext_svr.c`.

These two modules call the TCP/IP transport layer. `ext_transport.c` implements the client-side transport functions. `ext_svr_transport.c` contains the corresponding server-side functions. You can modify these files to support external mode via your own low-level communications layer.

You need only modify those parts of the code that implement low-level communications. You need not be concerned with issues such as data conversions between host and target, or with the formatting of messages. Code provided by the Real-Time Workshop handles these functions.

On the client (Simulink) side, communications are handled by `ext_comm`, a C MEX-file. This component is implemented as a DLL on Windows, or as a shared library on UNIX.

On the server (target) side, external mode modules are linked into the target executable. This takes place automatically if the **External mode** code generation option is selected at code generation time. These modules, called from the main program and the model execution engine, are independent of the generated model code.

To implement your own low-level protocol:

- On the client side, you must replace low-level TCP/IP calls in `ext_transport.c` with your own communication calls, and rebuild `ext_comm` using the `mex` command. You should then designate your custom `ext_comm` component as the **MEX-file for external interface** in the Simulink **External Target Interface** dialog.
- On the server side, you must replace low-level TCP/IP calls in `ext_svr_transport.c` with your own communication calls. If you are writing your own system target file and/or template makefile, make sure that the `EXT_MODE` code generation option is defined. The generated makefile will then link `ext_svr_transport.c` and other server code into your executable.
- Define symbols and functions common to both the client and server sides in `ext_transport_share.h`.

Overview of External Mode Communications

This section gives a high-level overview of how a Real-Time Workshop generated program communicates with Simulink in external mode. This description is based on the TCP/IP version of external mode that ships with Real-Time Workshop.

For communication to take place:

- The server (target) program must have been built with the conditional `EXT_MODE` defined. `EXT_MODE` is defined in the *model*`.mk` file if the **External mode** code generation option was selected at code generation time.
- Both the server program and Simulink must be executing. Note that this does not mean that the model code in the server system must be executing. The server may be waiting for Simulink to issue a command to start model execution.

The client and server communicate via two sockets. Each socket supports a distinct *channel*. The *message channel* is bidirectional; it carries commands, responses, and parameter downloads. The unidirectional *upload channel* is for uploading signal data to the client. The message channel is given higher priority.

If the target program was invoked with the `-w` command line option, the program enters a wait state until it receives a message from the host. Otherwise, the program begins execution of the model. While the target program is in a wait state, Simulink can download parameters to the target and configure data uploading.

When the user chooses the **Connect to target** option from the **Simulation** menu, the host initiates a handshake by sending an `EXT_CONNECT` message. The server responds with information about itself. This information includes:

- Checksums. The host uses model checksums to determine that the target code is an exact representation of the current Simulink model.
- Data format information. The host uses this information when formatting data to be downloaded, or interpreting data that has been uploaded.

At this point, host and server are connected. The server is either executing the model or in the wait state. (In the latter case, the user can begin model execution by selecting **Start real-time code** from the **Simulation** menu.)

During model execution, the message server runs as a background task. This task receives and processes messages such as parameter downloads.

Data uploading comprises both foreground execution and background servicing of the upload channel. As the target computes model outputs, it also copies signal values into data upload buffers. This occurs as part of the task associated with each task identifier (`tid`). Therefore, data collection occurs in the foreground. Transmission of the collected data, however, occurs as a background task. The background task sends the data in the collection buffers to Simulink via the upload channel.

The host initiates most exchanges on the message channel. The target usually sends a response confirming that it has received and processed the message. Examples of messages and commands are:

- Connection message / connection response
- Start target simulation / start response
- Parameter download / parameter download response
- Arm trigger for data uploading
- Terminate target simulation / target shutdown response

Model execution terminates when the model reaches its final time, when the host sends a terminate command, or when a Stop Simulation block terminates execution. On termination, the server informs the host that model execution has stopped, and shuts down both sockets. The host also shuts down its sockets, and exits external mode.

External Mode Source Files

Host (ext_comm) Source Files

The source files for the `ext_comm` component are located in the directory *matlabroot*`/rtw/ext_mode`:

- `ext_comm.c`

  This file is the core of external mode communication. It acts as a relay station between the target and Simulink. `ext_comm.c` communicates to Simulink via a shared data structure, `ExternalSim`. It communicates to the target via calls to the transport layer.

  Tasks carried out by `ext_comm` include establishment of a connection with the target, downloading of parameters, and termination of the connection with the target.
- `ext_transport.c`

  This file implements required transport layer functions. (Note that `ext_transport.c` includes `ext_transport_share.h`, which contains functions common to client and server sides.) The version of ext_transport.c shipped with the Real-Time Workshop uses TCP/IP functions including recv(), send(), and socket().

- ext_main.c

  This file is a MEX-file wrapper for external mode. ext_main interfaces to Simulink via the standard mexFunction call. (See "External Interfaces" in the MATLAB online documentation for information on mexFunction.) ext_main contains a function dispatcher, esGetAction, that sends requests from Simulink to ext_comm.

- ext_convert.c

  This file contains functions used for converting data from host to target formats (and vice versa). Functions include byte-swapping (big to little-endian), conversion from non-IEEE floats to IEEE doubles, and other conversions. These functions are called both by ext_comm.c and directly by Simulink (via function pointers).

Note You do not need to customize ext_convert in order to implement a custom transport layer. However, it may be necessary to customize ext_convert for the intended target. For example, if the target represents the float data type in Texas Instruments (TI) format, ext_convert must be modified to perform a TI to IEEE conversion.

- extsim.h

  This file defines the ExternalSim data structure and access macros. This structure is used for communication between Simulink and ext_comm.c.

- extutil.h

  This file contains only conditionals for compilation of the assert macro.

Target (Server) Source Files

These files are part of the run-time interface and are linked into the *model*.exe executable. They are located in the directory *matlabroot*/rtw/c/src.

- ext_svr.c

  ext_svr.c is analogous to ext_comm.c on the host, but generally is responsible for more tasks. It acts as a relay station between the host and the generated code. Like ext_comm.c, ext_svr.c carries out tasks such as establishing and terminating connection with the host. `ext_svr.c` also contains the background task functions that either write downloaded parameters to the target model, or extract data from the target data buffers and send it back to the host.

The version of `ext_svr.c` shipped with the Real-Time Workshop uses TCP/IP functions including `recv()`, `send()`, and `socket()`.

- `ext_svr_transport.c`

  This file implements required transport layer functions. (Note that `ext_svr_transport.c` includes `ext_transport_share.h`, which contains functions common to client and server sides.) The version of `ext_svr_transport.c` shipped with the Real-Time Workshop uses TCP/IP functions including `recv()`, `send()`, and `socket()`.

- `updown.c`

  `updown.c` handles the details of interacting with the target model. During parameter downloads, `updown.c` does the work of installing the new parameters into the model's parameter vector. For data uploading, `updown.c` contains the functions that extract data from the model's `blockio` vector and write the data to the upload buffers. `updown.c` provides services both to `ext_svr.c` and to the model code (e.g., `grt_main.c`). It contains code that is called via the background tasks of `ext_svr.c` as well as code that is called as part of the higher priority model execution.

- `dt_info.h` and *`model`*`.dt`

  These files contain data type transition information that allows access to multi-data type structures across different computer architectures. This information is used in data conversions between host and target formats.

- `updown_util.h`

  This file contains only conditionals for compilation of the `assert` macro.

Other Files

- `ext_share.h`

  Contains message code definitions and other definitions required by both the host and target modules.

- `ext_transport_share.h`

  Contains functions and data structures required by both the host and target modules of the transport layer. The version of `ext_transport_share.h` shipped with the Real-time Workshop is specific to TCP/IP communications.

Guidelines for Implementing the Transport Layer

Requirements

- `ext_svr.c` and `updown.c` use `malloc` to allocate buffers in target memory for messages, data collection, and other purposes. If your target uses some other memory allocation scheme, you must modify these modules appropriately.
- The target is assumed to support both `int32_T` and `uint32_T` data types.

Guidelines for Modifying ext_transport

The function prototypes in `ext_transport.h` define the calling interface for the host (client) side transport layer functions. The implementation is in `ext_transport.c`.

To implement the host side of your transport layer:

- Replace the functions in the "Visible Functions" section of `ext_transport.c` with functions that call your low-level communications primitives. The visible functions are called from other external mode modules such as `ext_comm.c`. You must implement all the functions defined in `ext_transport.h`. Your implementations must conform to the prototypes defined in `ext_transport.h`.
- Supply a definition for the `UserData` structure in `ext_transport.c`. This structure is required. If `UserData` is not necessary for your external mode implementation, define a `UserData` structure with one dummy field.
- Replace the functions in `ext_transport_share.h` with functions that call your low-level communications primitives, or remove these functions. Functions defined in `ext_transport_share.h` are common to the host and target, and are not part of the public interface to the transport layer.
- Rebuild the `ext_comm` MEX-file, using the MATLAB `mex` command. This requires a compiler supported by the MATLAB API. See "External Interfaces" in the MATLAB online documentation for more information on

the mex command. The following table lists the form of the commands to build the standard ext_comm module on PC and UNIX platforms.

Table 17-4: Commands to Rebuild ext_comm MEX-Files

Platform	Commands
PC	cd *matlabroot*\toolbox\rtw mex *matlabroot*\rtw\ext_mode\ext_comm.c *matlabroot*\rtw\ext_mode\ext_convert.c *matlabroot*\rtw\ext_mode\ext_transport.c –I*matlab*\rtw\c\src –DWIN32 compiler_library_path\wsock32.lib
UNIX	cd *matlabroot*/toolbox/rtw mex *matlabroot*/rtw/ext_mode/ext_comm.c *matlabroot*/rtw/ext_mode/ext_convert.c *matlabroot*/rtw/ext_mode/ext_transport.c –I*matlab*/rtw/c/src

The ext_transport and ext_transport_share source code modules are fully commented. See these files for further details.

Guidelines for Modifying ext_svr_transport

The function prototypes in ext_svr_transport.h define the calling interface for the target (server) side transport layer functions. The implementation is in ext_svr_transport.c.

To implement the target side of your transport layer:

- Replace the functions in ext_svr_transport.c with functions that call your low-level communications primitives. These are the functions called from other target modules such as the main program. You must implement all the functions defined in ext_svr_transport.h. Your implementations must conform to the prototypes defined in ext_svr_transport.h.
- Supply a definition for the ExtUserData structure in ext_svr_transport.c. This structure is required. If ExtUserData is not necessary for your external mode implementation, define an ExtUserData structure with one dummy field.
- Define the EXT_BLOCKING conditional in ext_svr_transport.c as needed:

- Define `EXT_BLOCKING` as 0 to poll for a connection to the host (appropriate for single-threaded applications).
- Define `EXT_BLOCKING` as 1 in multi-threaded applications where tasks are able to block for a connection to the host without blocking the entire program.

See also the comments on `EXT_BLOCKING` in `ext_svr_transport.c`.

The `ext_svr_transport` source code modules are fully commented. See these files for further details.

Combining Multiple Models

If you want to combine several models (or several instances of the same model) into a single executable, the Real-Time Workshop offers several options.

One solution is to use the S-function target to combine the models into a single model, and then generate an executable using either the `grt` or `grt_malloc` targets. Simulink and Real-Time workshop implicitly handle connections between models, sequencing of calls to the models, and multiple sample rates. This is the simplest solution in many cases. See Chapter 10, “The S-Function Target” for further information.

A second option, for embedded systems development, is to generate code from your models using the Real-Time Workshop Embedded Coder target. You can interface the model code to a common harness program by directly calling the entry points to each model. The Real-Time Workshop Embedded Coder target has certain restrictions that may not be appropriate for your application. For more information, see “Requirements and Restrictions” in Chapter 9, “Real-Time Workshop Embedded Coder.”

The `grt_malloc` target is a third solution. It is appropriate in situations where you want do any or all of the following:

- Selectively control calls to more than one model.
- Use dynamic memory allocation.
- Include models that employ continuous states.
- Log data to multiple files.
- Run one of the models in external mode.

This section discusses how to use the `grt_malloc` target to combine models into a single program. Before reading this section, you should become familiar with model execution in Real-Time Workshop programs. (See Chapter 6, “Program Architecture” and Chapter 7, “Models with Multiple Sample Rates.”) It will be helpful to refer to `grt_malloc_main.c` while reading these chapters.

The procedure for building a multiple-model executable is fairly straightforward. The first step is to generate and compile code from each of the models that are to be combined. Next, the makefiles for each of the models are combined into one makefile for creating the final multimodel executable. The next step is create a combined simulation engine by modifying `grt_malloc_main.c` to initialize and call the models correctly. Finally, the combination makefile links the object files from the models and the main program into an executable. “Example Mutliple-Model Program” on page 17-83 discusses an example implementation.

Sharing Data Across Models

We recommend using unidirectional signal connections between models. This affects the order in which models are called. For example, if an output signal from `modelA` is used as input to `modelB`, `modelA's` output computation should be called first.

Timing Issues

We recommend that you generate all the models you are combining with the same solver mode (either all singletasking or all multitasking.) In addition, if the models employ continuous states, the same solver should be used for all models.

If all the models to be combined have the same sample rates and the same number of rates, the models can share the same timing engine data structure. (The `TimingData` structure is defined in *matlabroot*`/rtw/c/src/mrt_sim.c`). Alternatively, the models can maintain separate timing engine data structures, as in the example program discussed below.

If the models have the same base rate, but have different sub-rates, each model should maintain a separate timing engine data structure. Each model should use the same base rate timer interrupt.

If the base rates for the models are not the same, the main program (such as `grt_malloc_main`) must set up the timer interrupt to occur at the greatest common divisor rate of the models. The main program is responsible for calling each of the models at the appropriate time interval.

Data Logging and External Mode Support

A multiple-model program can log data to separate MAT-files for each model (as in the example program discussed below.)

Only one of the models in a multiple-model program can use external mode.

Example Mutliple-Model Program

An example multiple-model program, distributed with Real-Time Workshop, is located at *matlabroot*`/rtw/c/grt_malloc/demos`. This example combines two models, `fuelsys1` and `mcolon`. Both models have the same base rate and the same number of sample times. For simplicity, each model creates and maintains a separate timing engine data structure (although it would be possible for them to share a common structure.) Each model logs states, outputs, and simulation time to a separate *model*`.mat` file.

The example files consist of:

- The models, `fuelsys1.mdl` and `mcolon.mdl`
- Run-time interface components: modified main program (`combine_malloc_main.c`) and modified solver (`ode5combine.c`)
- Control files: modified *model*`.bat` and *model*`.mk` files

Runtime Interface Components. The main program, `combine_malloc_main.c`, is a modified version of `grt_malloc_main.c`. `combine_malloc_main` employs two `#defines` for the models. `MODEL1()` and `MODEL2()` are macros that return pointers to the `Simstructs` for `fuelsys1` and `mcolon`, respectively. The code refers to the models through these pointers throughout, as in the following extract.

```
SimStruct  *S1;
SimStruct  *S2;
...

S1 = MODEL1();
S2 = MODEL2();
...

sfcnInitializeSizes(S1);
sfcnInitializeSizes(S2);
sfcnInitializeSampleTimes(S1);
sfcnInitializeSampleTimes(S2);
...
```

`combine_malloc_main.c` calls initialization, execution, and cleanup functions once for each model.

`combine_malloc_main.c` also uses the following `#defines`:

- `NCSTATES1` and `NCSTATES2` represent the number of continuous states in each model.
- `MATFILEA` and `MATFILEB` represent the MAT-files logged by each model.

To see all modifications in the main program, compare grt_malloc_main.c to combine_malloc_main.c.

The solver, ode5combine.c, is also modified to handle multiple models. The UpdateContinuousStates function has been modified to obtain the number of continuous states from each model's Simstruct at runtime. Simstructs are passed in by reference.

Control Files. combine.bat was created from the generated control files fuelsys1.bat and mcolon.bat. These were modified to invoke the makefile, combine.mk.

combine.mk combines parameters from the generated makefiles, fuelsys1.mk and mcolon.mk. Note the following modifications:

- combine.mk contains the model-specific defines MODEL1, MODEL2, NCSTATES1, and NCSTATES2. Note that these defines are included in the CPP_REQ_DEFINES list.
- The SOLVER parameter specifies ode5combine.c.
- The REQ_SRCS list includes combine_malloc_main.c.
- Two rules have been added in order for make to locate source files in the build subdirectories, fuelsys1_grt_malloc_rtw and mcolon_grt_malloc_rtw.

Building the Example Program. To try the example:

1. use the build command to generate code and compile object files for the fuelsys1 and mcolon models.
2. Modify the MATLAB_ROOT and MATLAB_BIN path information in combine.mk as appropriate for your installation.
3. At the command prompt, type

   combine.bat (on PC)

   or

   make -f combine.mk (on UNIX)

   to build the combine executable.

DSP Processor Support

The Real-Time Workshop now supports target processors that have only one register size (e.g., 32-bit). This makes data type emulation of 8 and 16 bits on the TCI_C30/C40 DSP and similar processors possible.

To support these processors:

- Add the command

  ```
  -DDSP32=1
  ```

  to your template makefile.
- Add the statement

  ```
  %assign DSP32=1
  ```

  to your system target file.

Blocks That Depend on Absolute Time

Some Simulink blocks use the value of absolute time (i.e., the time from the beginning of the program to the present time) to calculate their outputs. If you are designing a program that is intended to run indefinitely, then you cannot use blocks that have a dependency on absolute time.

The problem arises when the value of time reaches the largest value that can be represented by a double precision number. At that point, time is no longer incremented and the output of the block is no longer correct.

Note In addition to the blocks listed below, logging **Time** (in the Workspace I/O page of the **Simulation Parameters** dialog box) also requires absolute time.

The following Simulink blocks depend on absolute time:

- Continuous Blocks
  - Derivative
  - Transport Delay
  - Variable Transport Delay
- Discrete Blocks
  - Discrete-Time Integrator (when used in triggered subsystems)
- Nonlinear Blocks
  - Rate Limiter
- Sinks
  - Scope
  - To File
  - To Workspace (only if logging to `StructureWithTime` format)
- Sources
  - Chirp Signal
  - Clock
  - Digital Clock
  - Discrete Pulse Generator
  - From File

- From Workspace
- Pulse Generator
- Ramp
- Repeating Sequence
- Signal Generator
- SineWave
- Step

In addition to the Simulink block above:

- Blocks in other Blocksets may reference absolute time. Please refer to the documentation for the Blocksets that you use.
- Stateflow charts that use time are dependent on absolute time.

B

Glossary

Application Modules – With respect to Real-Time Workshop program architecture, these are collections of programs that implement functions carried out by the system dependent, system independent, and application components.

Block Target File – A file that describes how a specific Simulink block is to be transformed to a language such as C, based on the block's description in the Real-Time Workshop file (*model*`.rtw`). Typically, there is one block target file for each Simulink block.

File Extensions – Below is a table that lists the file extensions associated with Simulink, the Target Language Compiler, and the Real-Time Workshop.

Extension	Created by	Description
`.c`	Target Language Compiler	The generated C code
`.h`	Target Language Compiler	A C include header file used by the `.c` program
`.mdl`	Simulink	Contains structures associated with Simulink block diagrams
`.mk`	Real-Time Workshop	A makefile specific to your model that is derived from the template makefile
`.rtw`	Real-Time Workshop	A translation of the `.mdl` file into an intermediate file prior to generating C code
`.tlc`	Target Language Compiler	Script files that Real-Time Workshop uses to translate
`.tmf`	Supplied with Real-Time Workshop	A template makefile

Generic Real-Time (GRT) target – A target configuration that generates model code for a real-time system, with the resulting code executed on your workstation. (Note that execution is not tied to a real-time clock.) You can use GRT as a starting point for targeting custom hardware.

Host System – The computer system on which you create your real-time application.

Inline – Generally, this means to place something directly in the generated source code. You can inline parameters and S-functions using the Real-Time Workshop.

Inlined Parameters (Target Language Compiler Boolean global variable: `InlineParameters`) – The numerical values of the block parameters are hard coded into the generated code. Advantages include faster execution and less memory use, but you lose the ability to change the block parameter values at run-time.

Inlined S-Function – An S-function can be inlined into the generated code by implementing it as a `.tlc` file. The code for this S-function is placed in the generated model code itself. In contrast, noninlined S-functions require a function call to S-function residing in an external MEX-file.

Interrupt Service Routine (ISR) – A piece of code that your processor executes when an external event, such as a timer, occurs.

Loop Rolling (Target Language Compiler global variable: `RollThreshold`) – Depending on the block's operation and the width of the input/output ports, the generated code uses a `for` statement (rolled code) instead of repeating identical lines of code (flat code) over the block width.

Make – A utility to maintain, update, and regenerate related programs and files. The commands to be executed are placed in a *makefile*.

Makefiles – Files that contain a collection of commands that allow groups of programs, object files, libraries, etc. to interact. Makefiles are executed by your development system’s make utility.

Multitasking – A process by which your microprocessor schedules the handling of multiple tasks. The number of tasks is equal to the number of sample times in your model.

Noninlined S-Function – In the context of the Real-Time Workshop, this is any C MEX S-function that is not implemented using a customized `.tlc` file. If you create an C MEX S-function as part of a Simulink model, it is by default noninlined unless you write your own `.tlc` file that inlines it.

Non-Real-Time – A simulation environment of a block diagram provided for high-speed simulation of your model. Execution is not tied to a real-time clock.

Nonvirtual Block – Any block that performs some algorithm, such as a Gain block.

Pseudomultitasking – In processors that do not offer multitasking support, you can perform pseudomultitasking by scheduling events on a fixed time-sharing basis.

Real-Time System – A system that uses actual hardware to implement algorithms, for example, digital signal processing or control applications.

Run-time Interface – A wrapper around the generated code that can be built into a stand-alone executable. The run-time interface consists of routines to move the time forward, save logged variables at the appropriate time steps, etc. The run-time interface is responsible for managing the execution of the real-time program created from your Simulink block diagram.

S-Function – A customized Simulink block written in C or M-code. C-code S-functions can be inlined in the Real-Time Workshop.

Singletasking – A mode in which a model runs in one task.

System Target File – The entry point to the Target Language Compiler program, used to transform the Real-Time Workshop file into target specific code.

Target Language Compiler – A compiler that compiles and executes system and target files.

Target File – A file that is compiled and executed by the Target Language Compiler. A combination of these files describes how to transform the Real-Time Workshop file (`model.rtw`) into target-specific code.

Target System – The computer system on which you execute your real-time application.

Targeting – The process of creating an executable for your target system.

Template Makefile – A line-for-line makefile used by a make utility. The template makefile is converted to a makefile by copying the contents of the template makefile (usually `system.tmf`) to a makefile (usually `system.mk`) replacing tokens describing your model's configuration.

Target Language Compiler Program – A set of TLC files that describe how to convert a `model.rtw` file into generated code.

Task Identifier (tid) – Each sample time in your model is assigned a task identifier (`tid`). The `tid` is passed to the model output and update routines to decide which portion of your model should be executed at a given time.

Virtual Block – A connection or graphical block, for example, a Mux block.

Index