

Simulink[®]

Simulation and Model-Based Design

- Modeling
- Simulation
- Implementation

Getting Started

Version 6



How to Contact The MathWorks:



www.mathworks.com
comp.soft-sys.matlab
www.mathworks.com/contact_TS.html

Web
Newsgroup
Technical Support



suggest@mathworks.com
bugs@mathworks.com
doc@mathworks.com
service@mathworks.com
info@mathworks.com

Product enhancement suggestions
Bug reports
Documentation error reports
Order status, license renewals, passcodes
Sales, pricing, and general information



508-647-7000 (Phone)



508-647-7001 (Fax)



The MathWorks, Inc.
3 Apple Hill Drive
Natick, MA 01760-2098

For contact information about worldwide offices, see the MathWorks Web site.

Getting Started with Simulink

© COPYRIGHT 2005–2006 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

FEDERAL ACQUISITION: This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

Trademarks

MATLAB, Simulink, Stateflow, Handle Graphics, Real-Time Workshop, and xPC TargetBox are registered trademarks of The MathWorks, Inc.

Other product or brand names are trademarks or registered trademarks of their respective holders.

Patents

The MathWorks products are protected by one or more U.S. patents. Please see www.mathworks.com/patents for more information.

Revision History

September 2005	Online only	New for Version 6.3 (Release 14SP3)
March 2006	Online only	Revised for Simulink 6.4 (Release 2006a)

Introduction

1

What Is Simulink?	1-2
Tool for Simulation	1-2
Tool for Model-Based Design	1-2
Related Products	1-4

Running a Model

2

Running a Demo Model	2-2
About the Demo Model	2-4
Some Things to Try	2-7
What This Demo Illustrates	2-7
Other Useful Demos	2-8

Building a Model

3

Model Example	3-2
Creating an Empty Model	3-3

Adding Blocks	3-5
Connecting the Blocks	3-9
Configuring the Model	3-12
Running the Model	3-13

Setting Simulink Preferences

4

Setting General Preferences	4-2
Window reuse	4-2
Model Browser	4-3
Display	4-3
Callback tracing	4-3
Setting Font Preferences	4-4
Simulation Preferences	4-5

Examples

A

Working with Demos	A-2
Building a Model	A-2

Index

Introduction

The following sections give you a brief overview of Simulink® and related products.

What Is Simulink? (p. 1-2)

Introduces Simulink.

Related Products (p. 1-4)

Describes products that extend the capabilities of Simulink.

What Is Simulink?

Simulink is a software package for modeling, simulating, and analyzing dynamic systems. It supports linear and nonlinear systems, modeled in continuous time, sampled time, or a hybrid of the two. Systems can also be multirate, i.e., have different parts that are sampled or updated at different rates.

Tool for Simulation

Simulink encourages you to try things out. You can easily build models from scratch, or take an existing model and add to it. You have instant access to all the analysis tools in MATLAB®, so you can take the results and analyze and visualize them. A goal of Simulink is to give you a sense of the *fun* of modeling and simulation, through an environment that encourages you to pose a question, model it, and see what happens.

Simulink is also practical. With thousands of engineers around the world using it to model and solve real problems, knowledge of this tool will serve you well throughout your professional career.

Tool for Model-Based Design

With Simulink, you can move beyond idealized linear models to explore more realistic nonlinear models, factoring in friction, air resistance, gear slippage, hard stops, and the other things that describe real-world phenomena. Simulink turns your computer into a lab for modeling and analyzing systems that simply wouldn't be possible or practical otherwise, whether the behavior of an automotive clutch system, the flutter of an airplane wing, the dynamics of a predator-prey model, or the effect of the monetary supply on the economy. Simulink provides numerous demos that model a wide variety of such real-world phenomena. For more information about accessing and executing these demos, see Chapter 2, “Running a Model”.

For modeling, Simulink provides a graphical user interface (GUI) for building models as block diagrams, using click-and-drag mouse operations. With this interface, you can draw the models just as you would with pencil and paper (or as most textbooks depict them). This is a far cry from previous simulation packages that require you to formulate differential equations and difference equations in a language or program. Simulink includes

a comprehensive block library of sinks, sources, linear and nonlinear components, and connectors. You can also customize and create your own blocks. For information on creating your own blocks, see [Writing S-Functions](#) in the online documentation.

Models are hierarchical, so you can build models using both top-down and bottom-up approaches. You can view the system at a high level, then double-click blocks to go down through the levels to see increasing levels of model detail. This approach provides insight into how a model is organized and how its parts interact.

After you define a model, you can simulate it, using a choice of integration methods, either from the Simulink menus or by entering commands in the MATLAB Command Window. The menus are particularly convenient for interactive work, while the command-line approach is very useful for running a batch of simulations (for example, if you are doing Monte Carlo simulations or want to sweep a parameter across a range of values). Using scopes and other display blocks, you can see the simulation results while the simulation is running. In addition, you can change many parameters and see what happens for “what if” exploration. The simulation results can be put in the MATLAB workspace for postprocessing and visualization.

Model analysis tools include linearization and trimming tools, which can be accessed from the MATLAB command line, plus the many tools in MATLAB and its application toolboxes. And because MATLAB and Simulink are integrated, you can simulate, analyze, and revise your models in either environment at any point.

Related Products

The MathWorks provides several products that are especially relevant to the kinds of tasks you can perform with Simulink and that extend the capabilities of Simulink. For information about these related products, see <http://www.mathworks.com/products/simulink/related.html>.

Running a Model

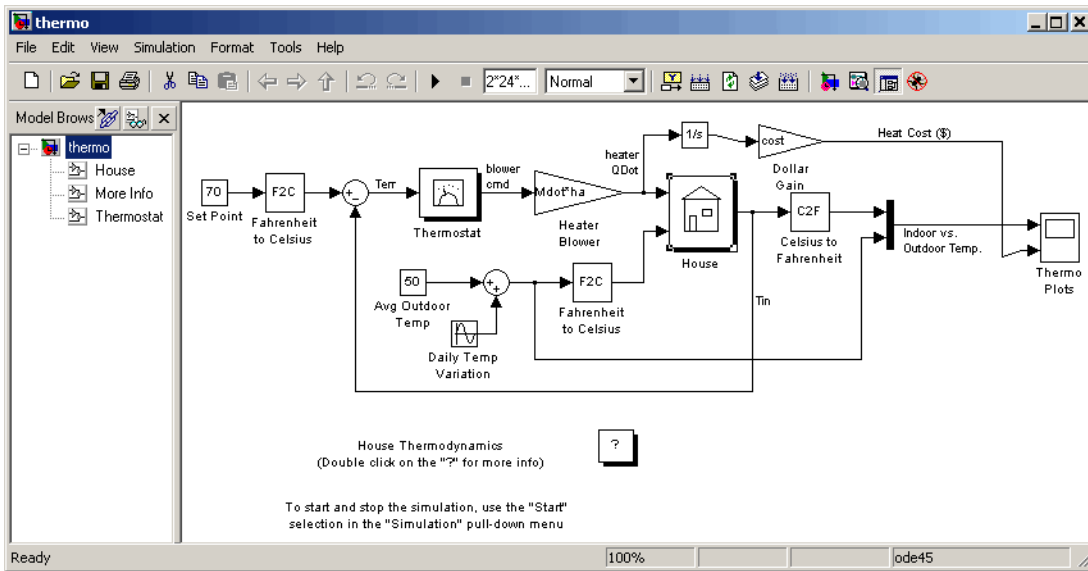
The following section gives you a quick introduction to running a Simulink® model.

Running a Demo Model (p. 2-2)	Example of how to run a Simulink model.
About the Demo Model (p. 2-4)	Explains how the model works.
Some Things to Try (p. 2-7)	Suggests some model parameters to change to see how they affect the simulation.
What This Demo Illustrates (p. 2-7)	Describes the common modeling and simulation tasks that this demo illustrates.
Other Useful Demos (p. 2-8)	Shows how to find other demos that illustrate key Simulink concepts and features.

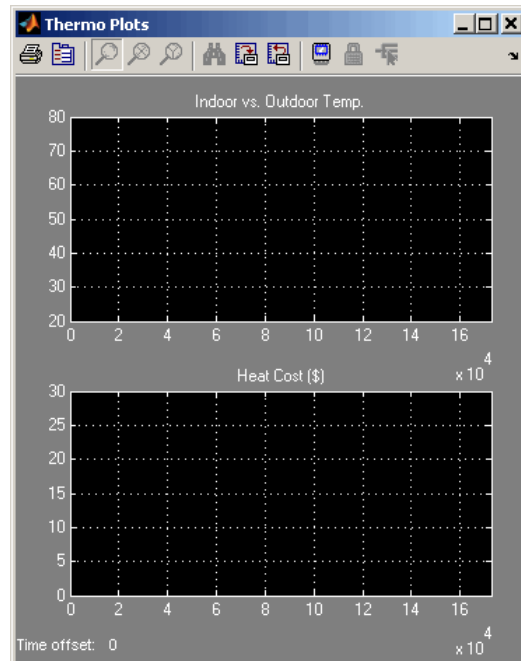
Running a Demo Model

An interesting demo program provided with Simulink models the thermodynamics of a house. To run this demo, follow these steps:

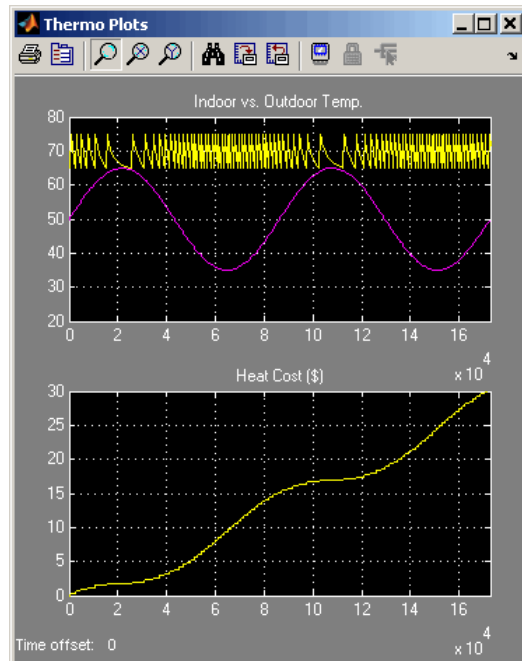
- 1 Start MATLAB. See your MATLAB documentation if you're not sure how to do this.
- 2 Run the demo model by typing thermo in the MATLAB Command Window. This command starts up Simulink and creates a model window that contains this model.



It also creates a scope window for the model. The Scope block displays two plots labeled Indoor vs. Outdoor Temp and Heat Cost (\$), respectively.



- 3 To start the simulation, pull down the **Simulation** menu and choose the **Start** command (or, on Microsoft Windows, click the **Start** button on the Simulink toolbar). As the simulation runs, the indoor and outdoor temperatures appear in the Indoor vs. Outdoor Temp plot as yellow and magenta signals, respectively. The cumulative heating cost appears in the Heat Cost (\$) plot.



See Scope for further details about signal element colors.

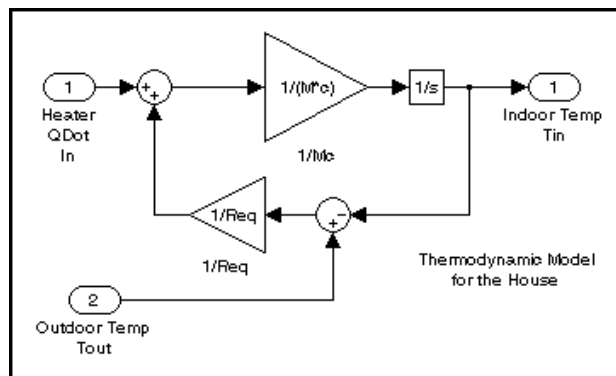
- 4 To stop the simulation, choose the **Stop** command from the **Simulation** menu (or click the **Pause** button on the toolbar). If you want to explore other parts of the model, look over the suggestions in “Some Things to Try” on page 2-7.
- 5 When you’re finished running the simulation, close the model by choosing **Close** from the **File** menu.

About the Demo Model

The demo models the thermodynamics of a house. The thermostat is set to 70 degrees Fahrenheit and is affected by the outside temperature, which varies by applying a sine wave with amplitude of 15 degrees to a base temperature of 50 degrees. This simulates daily temperature fluctuations.

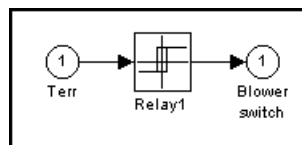
The model uses subsystems to simplify the model diagram and create reusable systems. A subsystem is a group of blocks that is represented by a Subsystem block. This model contains five subsystems: one named Thermostat, one named House, and three Temp Convert subsystems (two convert Fahrenheit to Celsius, one converts Celsius to Fahrenheit).

The internal and external temperatures are fed into the House subsystem, which updates the internal temperature. Double-click the House block to see the underlying blocks in that subsystem.



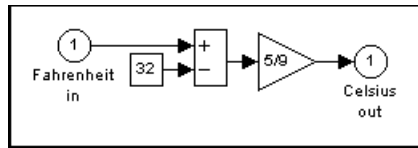
House subsystem

The Thermostat subsystem models the operation of a thermostat, determining when the heating system is turned on and off. Double-click the block to see the underlying blocks in that subsystem.



Thermostat subsystem

Both the outside and inside temperatures are converted from Fahrenheit to Celsius by identical subsystems. Note that the conversion subsystems are masks. Right-click the subsystem and select **Look Under Mask** to view the subsystem (see "Creating Masked Subsystems" for further details).



Fahrenheit to Celsius conversion (F2C)

When the heat is on, the heating costs are computed and displayed on the Heat Cost (\$) plot on the Thermo Plots Scope. The internal temperature is displayed on the Indoor Temp Scope.

Some Things to Try

Here are several things to try to see how the model responds to different parameters:

- Each Scope block contains one or more signal display areas and controls that enable you to select the range of the signal displayed, zoom in on a portion of the signal, and perform other useful tasks. The horizontal axis represents time and the vertical axis represents the signal value.
- The Constant block labeled Set Point (at the top left of the model) sets the desired internal temperature. Open this block and reset the value to 80 degrees. Rerun the simulation to see how the indoor temperature and heating costs change. Also, adjust the outside temperature (the Avg Outdoor Temp block) and rerun the simulation to see how it affects the indoor temperature.
- Adjust the daily temperature variation by opening the Sine Wave block labeled Daily Temp Variation and changing the **Amplitude** parameter and rerun the simulation.

What This Demo Illustrates

This demo illustrates several tasks commonly used when you are building models:

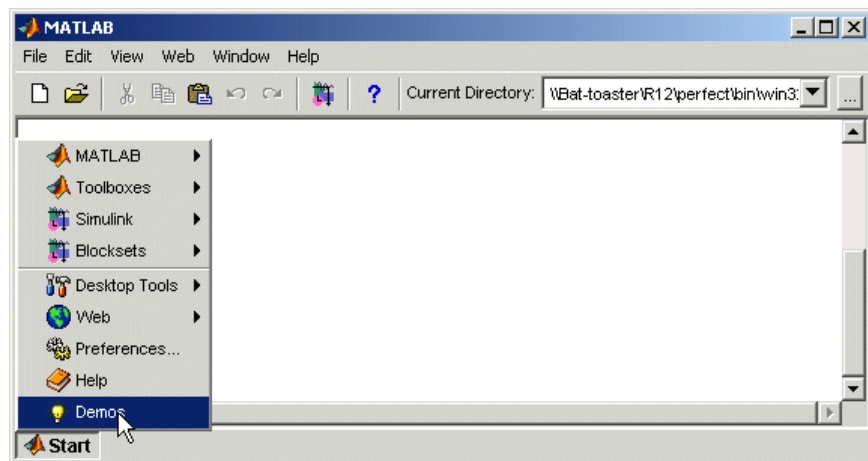
- Running the simulation involves specifying parameters and starting the simulation with the **Start** command, described in “Running Simulations”.
- You can encapsulate complex groups of related blocks in a single block, called a subsystem. See “Creating Subsystems” for more information.
- You can customize the appearance of and design a dialog box for a block by using the masking feature, described in detail in “Creating Masked Subsystems”. The thermo model uses the masking feature to customize the appearance of all the Subsystem blocks that it contains.
- The Scope blocks display graphic output much as an actual oscilloscope does.

Other Useful Demos

Other demos illustrate useful modeling concepts. You can access these demos from the MATLAB Command Window:

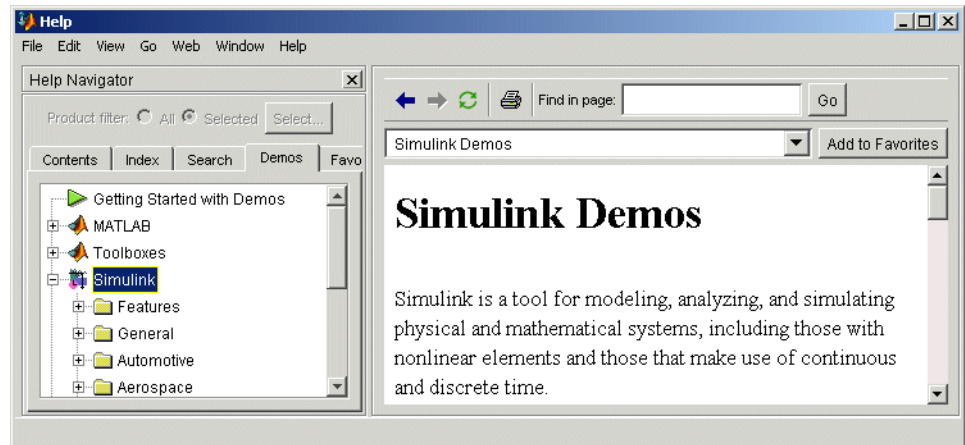
- 1 Click the **Start** button on the bottom left corner of the MATLAB Command Window.

The **Start** menu appears.



- 2 Select **Demos** from the menu.

The MATLAB Help browser appears with the **Demos** pane selected.



3 Click the **Simulink** entry in the **Demos** pane.

The entry expands to show groups of Simulink demos. Use the browser to navigate to demos of interest. The browser displays explanations of each demo and includes a link to the demo itself. Click on a demo link to start the demo.

Building a Model

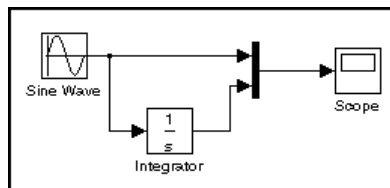
The following sections show you to build a model of a simple dynamic system, using Simulink.

Model Example (p. 3-2)	Example of how to build a Simulink model.
Creating an Empty Model (p. 3-3)	How to create an empty model.
Adding Blocks (p. 3-5)	How to add blocks to a model.
Connecting the Blocks (p. 3-9)	How to connect block outputs to block inputs.
Configuring the Model (p. 3-12)	How to select model configuration options.
Running the Model (p. 3-13)	How to run a Simulink model.

Model Example

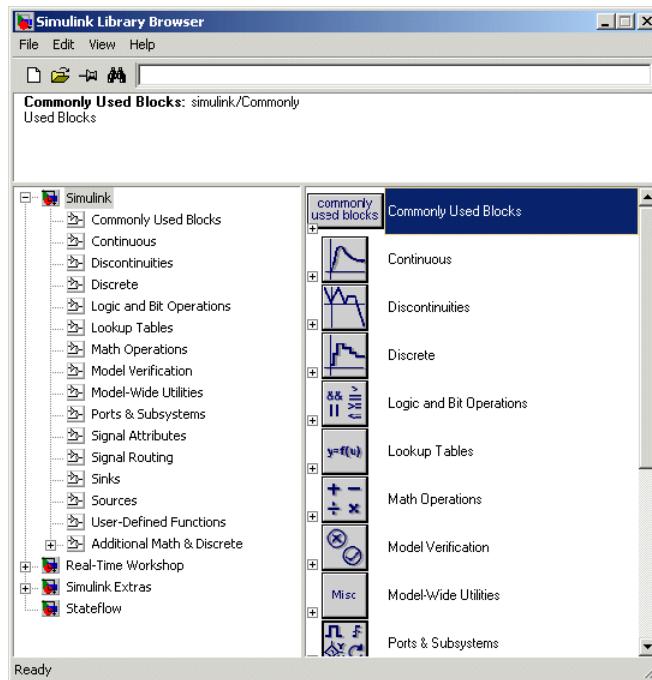
This example shows you how to build a model using many of the model-building commands and actions you will use to build your own models. The instructions for building this model in this section are brief. All the tasks are described in more detail in the next chapter.

The model integrates a sine wave and displays the result along with the sine wave. The block diagram of the model looks like this.

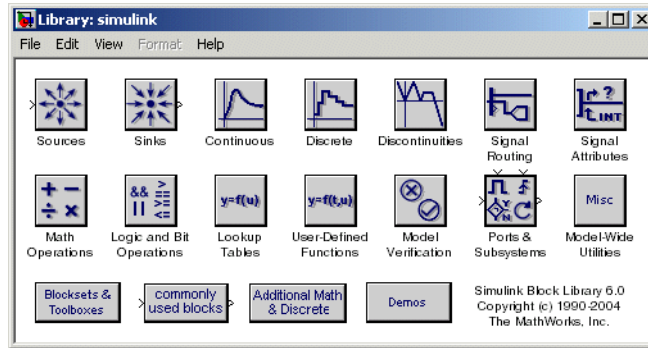


Creating an Empty Model

To create the model, first enter `simulink` in the MATLAB Command Window. On Microsoft Windows, the Simulink Library Browser appears.



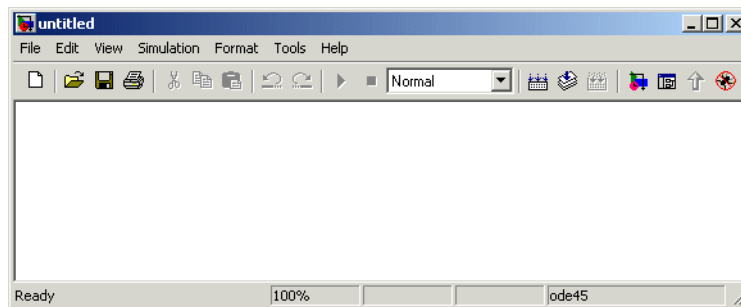
On UNIX,, the Simulink library window appears.



To create a new model on UNIX , select **Model** from the **New** submenu of the Simulink library window's **File** menu. To create a new model on Windows, click the **New Model** button on the Library Browser's toolbar.



Simulink opens a new model window.



Adding Blocks

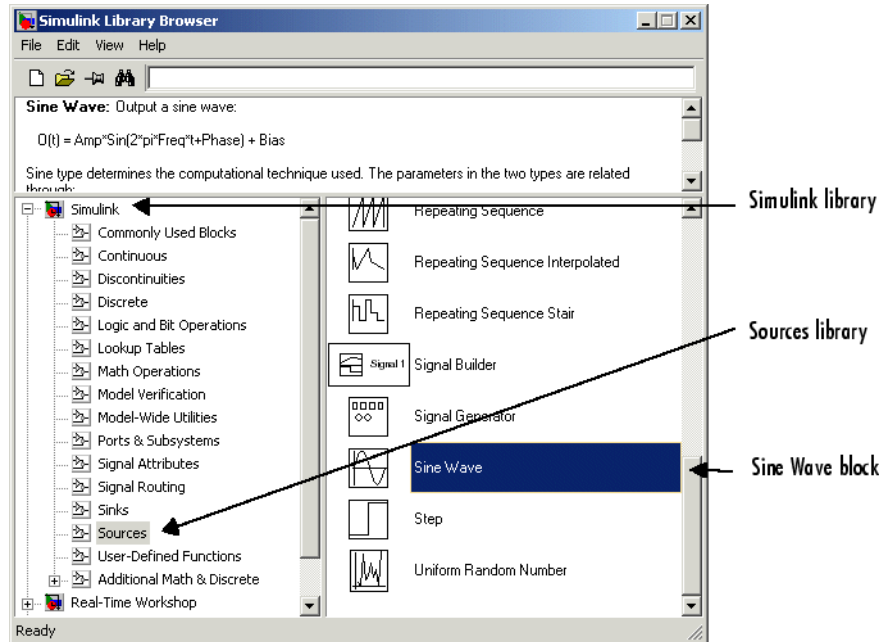
To create this model, you need to copy blocks into the model from the following Simulink block libraries:

- Sources library (the Sine Wave block)
- Sinks library (the Scope block)
- Continuous library (the Integrator block)
- Signal Routing library (the Mux block)

You can copy a Sine Wave block from the Sources library, using the Library Browser (Windows only) or the Sources library window (UNIX, and Windows).

To copy the Sine Wave block from the Library Browser, first expand the Library Browser tree to display the blocks in the Sources library. Do this by clicking the Sources node to display the Sources library blocks. Finally, click the Sine Wave node to select the Sine Wave block.

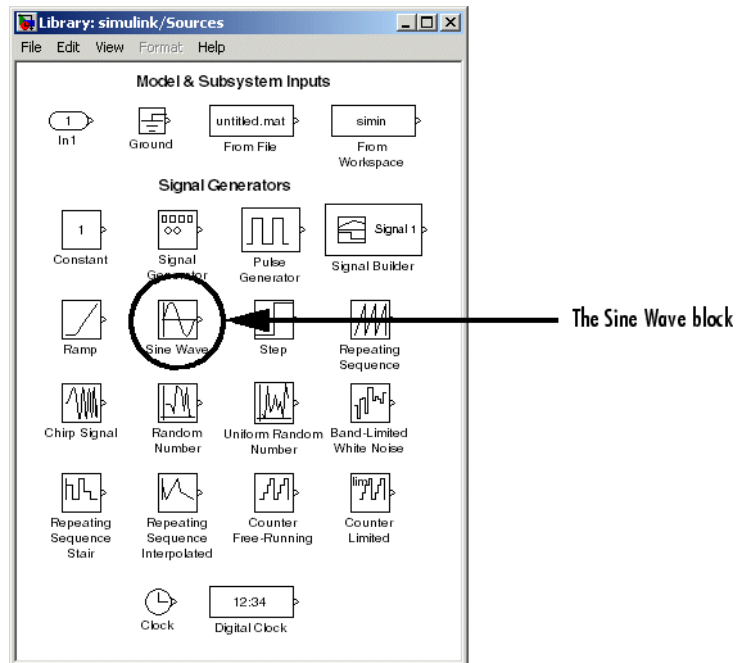
Here is how the Library Browser should look after you have done this.



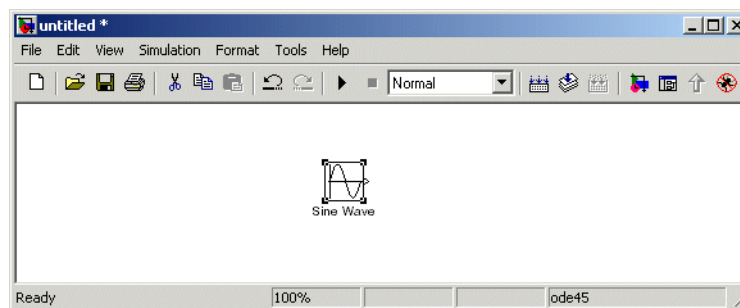
Now drag a copy of the Sine Wave block from the browser and drop it in the model window.

To copy the Sine Wave block from the Sources library window, open the Sources window by double-clicking the Sources icon in the Simulink library window. (On Windows, you can open the Simulink library window by right-clicking the Simulink node in the Library Browser and then clicking the resulting **Open Library** button.)

Simulink displays the Sources library window.



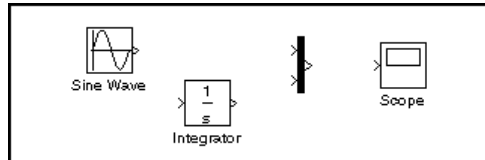
Now drag the Sine Wave block from the Sources window to your model window.



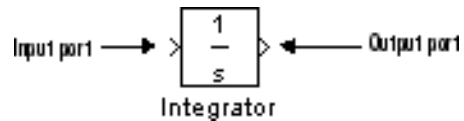
Copy the rest of the blocks in a similar manner from their respective libraries into the model window. You can move a block from one place in the model

window to another by dragging the block. You can move a block a short distance by selecting the block, then pressing the arrow keys.

With all the blocks copied into the model window, the model should look something like this.

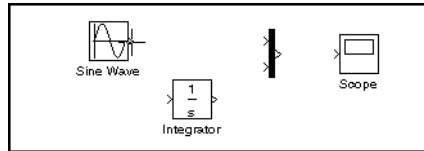


Notice that one or both sides of the blocks have angle brackets. The > symbol pointing out of a block is an *output port*; if the symbol points to a block, it is an *input port*.



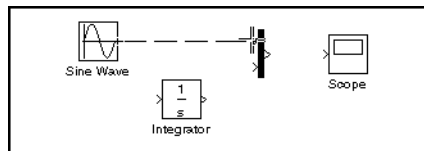
Connecting the Blocks

Now it's time to connect the blocks. Connect the Sine Wave block to the top input port of the Mux block. Position the pointer over the output port on the right side of the Sine Wave block. Notice that the cursor shape changes to crosshairs.

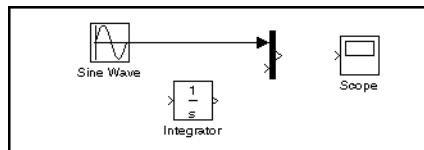


Hold down the mouse button and move the cursor to the top input port of the Mux block.

Notice that the line is dashed while the mouse button is down and that the cursor shape changes to double-lined crosshairs as it approaches the Mux block.



Now release the mouse button. The blocks are connected. You can also connect the line to the block by releasing the mouse button while the pointer is over the block. If you do, the line is connected to the input port closest to the cursor's position.

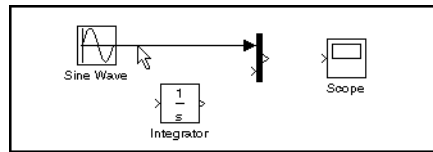


If you look again at the model at the beginning of this section, you'll notice that most of the lines connect output ports of blocks to input ports of other blocks. However, one line connects a *line* to the input port of another block. This line, called a *branch line*, connects the Sine Wave output to the

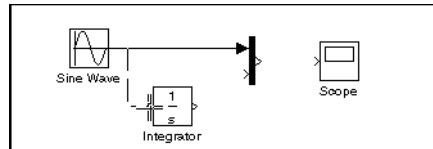
Integrator block, and carries the same signal that passes from the Sine Wave block to the Mux block.

Drawing a branch line is slightly different from drawing the line you just drew. To weld a connection to an existing line, follow these steps:

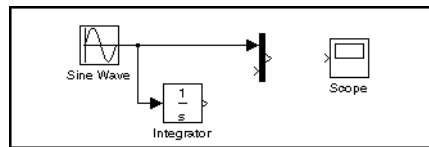
- 1 First, position the pointer *on the line* between the Sine Wave and the Mux block.



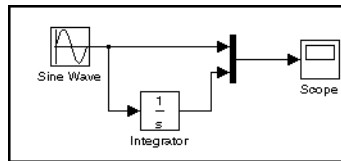
- 2 Press and hold down the **Ctrl** key (or click the right mouse button). Press the mouse button, then drag the pointer to the Integrator block's input port or over the Integrator block itself.



- 3 Release the mouse button. Simulink draws a line between the starting point and the Integrator block's input port.



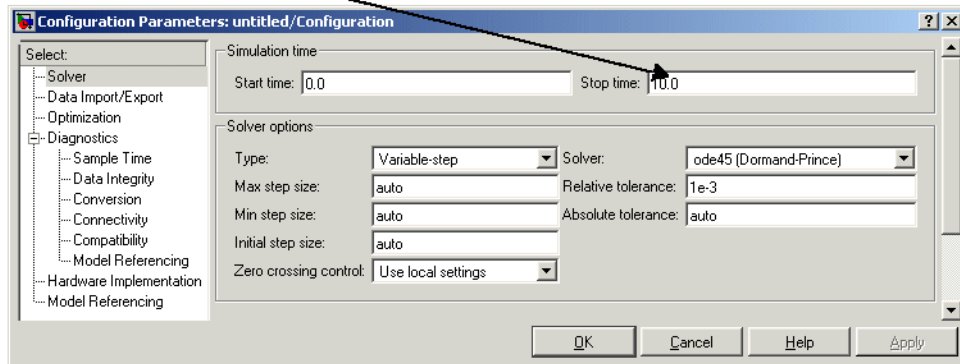
Finish making block connections. When you're done, your model should look something like this.



Configuring the Model

Now set up Simulink to run the simulation for 10 seconds. First, open the **Configuration Parameters** dialog box by choosing **Configuration Parameters** from the **Simulation** menu. On the dialog box that appears, notice that the **Stop time** is set to 10.0 (its default value).

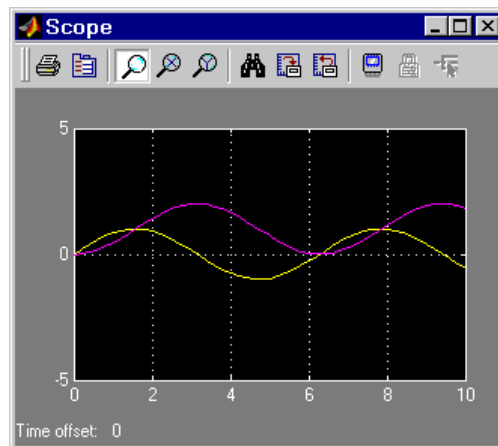
Stop time parameter



Close the **Configuration Parameters** dialog box by clicking the **OK** button. Simulink applies the parameters and closes the dialog box.

Running the Model

Now double-click the Scope block to open its display window. Finally, choose **Start** from the **Simulation** menu and watch the simulation output on the Scope.



The simulation stops when it reaches the stop time specified in the **Configuration Parameters** dialog box or when you choose **Stop** from the **Simulation** menu or click the **Stop** button on the model window's toolbar (Windows only).

To save this model, choose **Save** from the **File** menu and enter a filename and location. That file contains the description of the model.

To terminate Simulink and MATLAB, choose **Exit MATLAB** (on a Microsoft Windows system) or **Quit MATLAB** (on a UNIX system). You can also enter quit in the MATLAB Command Window. If you want to leave Simulink but not terminate MATLAB, just close all Simulink windows.

This exercise shows you how to perform some commonly used model-building tasks. These and other tasks are described in more detail in "Creating a Model".

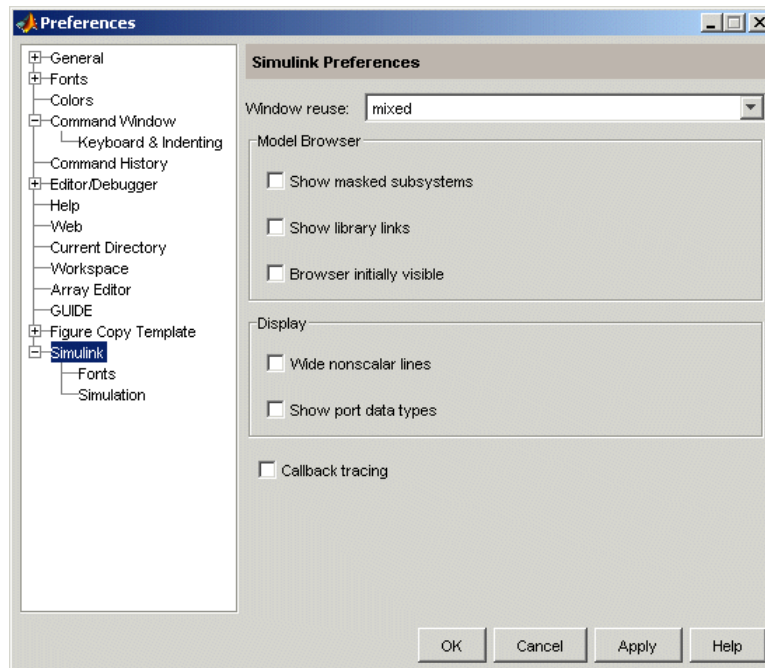
Setting Simulink Preferences

The following sections explain how to specify your own default settings for many Simulink options.

- | | |
|--------------------------------------|---|
| Setting General Preferences (p. 4-2) | How to open the dialog box used to set Simulink preferences and set preferences for miscellaneous Simulink options. |
| Setting Font Preferences (p. 4-4) | How to set the fonts you prefer for block and line labels and model annotations. |
| Simulation Preferences (p. 4-5) | How to specify a preferred simulation start and stop time and other preferred simulation options. |

Setting General Preferences

The Simulink **Preferences** dialog box allows you to specify default settings for some Simulink options. To display the **Preferences** dialog box, select **Preferences** from the Simulink **File** menu. A **Simulink Preferences** pane is displayed on the right side of the dialog box.



Alternatively, you can access the same dialog box from the MATLAB window by selecting **Preferences** from the MATLAB **File** menu, then selecting **Simulink** in the left pane.

The **Simulink Preferences** pane allows you to specify the following Simulink preferences.

Window reuse

Specifies whether Simulink uses existing windows or opens new windows to display a model's subsystems (see “Window Reuse” in “Using Simulink”).

Model Browser

Specifies whether Simulink displays the browser when you open a model and whether the browser shows blocks imported from subsystems and the contents of masked subsystems (see “The Model Browser” in “Using Simulink”).

Display

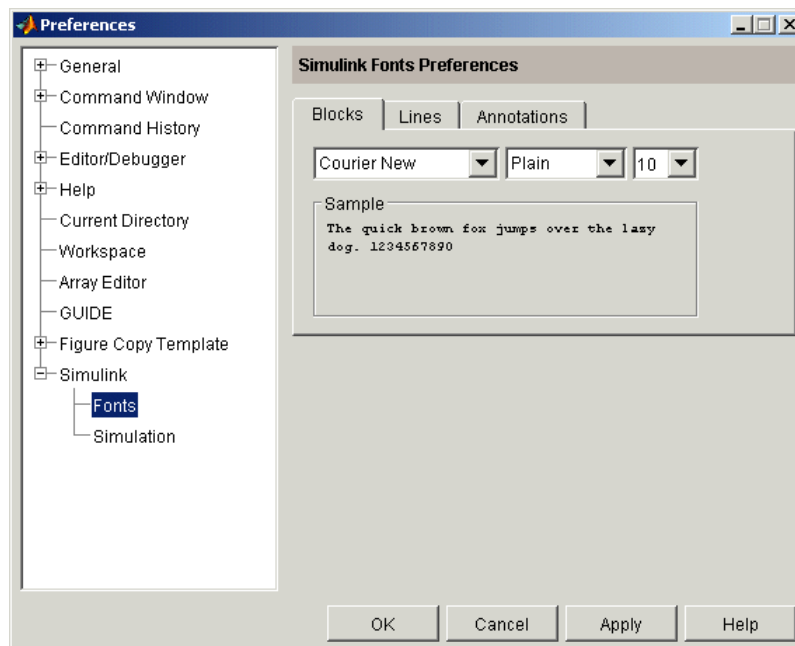
Specifies whether to use thick lines to display nonscalar connections between blocks and whether to display port data types on the block diagram

Callback tracing

Specifies whether to display the model callbacks that Simulink invokes when simulating a model (see “Using Callback Functions in “Using Simulink”).

Setting Font Preferences

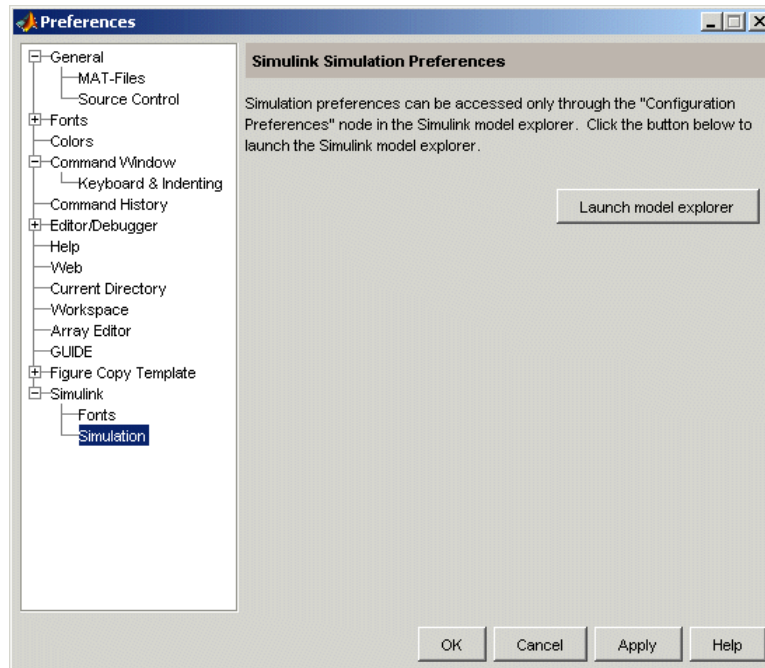
Selecting the Fonts subnode of the Simulink node in the left side of the dialog box displays a stack of tabbed panes on the right side of the dialog box.



The panes allow you to specify your preferred fonts for block and line labels and model annotations, respectively.

Simulation Preferences

Selecting the Simulation node beneath the Simulink node in the left side of the dialog box displays a button to start the Model Explorer (see “The Model Explorer” in “Using Simulink”).



Use the Model Explorer to set your simulation preferences.

Examples

Use this list to find examples in the documentation.

Working with Demos

“Running a Demo Model” on page 2-2

“Other Useful Demos” on page 2-8

Building a Model

“Creating an Empty Model” on page 3-3

“Adding Blocks” on page 3-5

“Connecting the Blocks” on page 3-9

“Configuring the Model” on page 3-12

“Running the Model” on page 3-13

B

building models
 exercise 3-2

C

Configuration Parameters menu item 3-12
connecting lines to input ports 3-9

L

lines
 carrying the same signal 3-9
 connecting to input ports 3-9

M

models
 building 3-2

P

parameters
 specifying configuration 3-12
preferences 4-1
Preferences dialog box 4-2

S

sample model 3-2
Simulink preferences 4-1