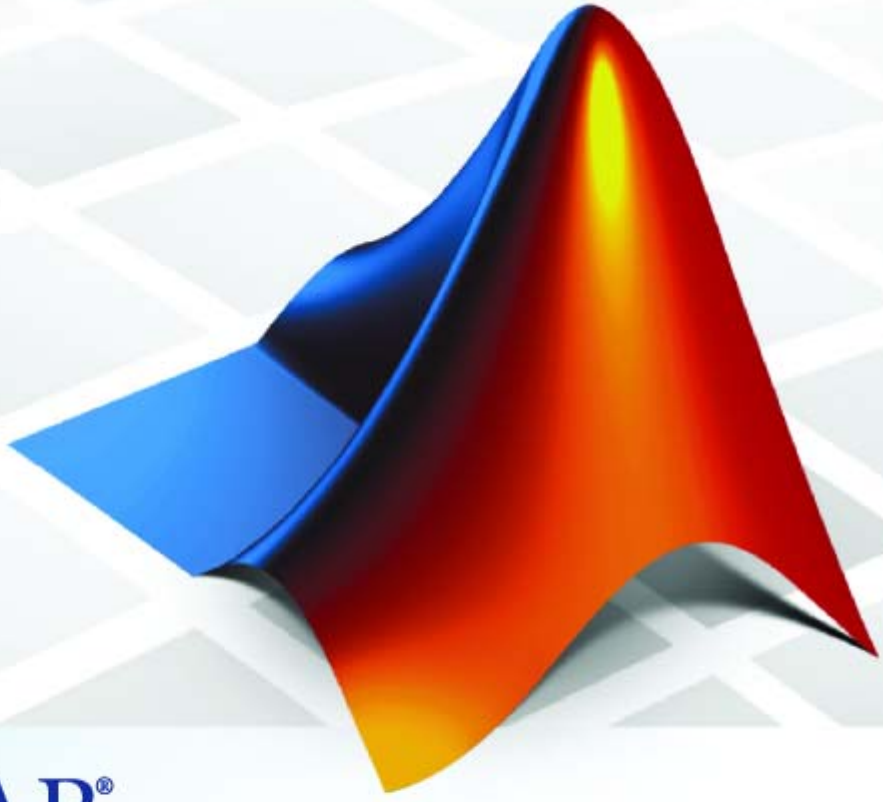
# Getting Started with **Simulink® 6**



MATLAB®
& SIMULINK®

The MathWorks
Accelerating the pace of engineering and science

### How to Contact The MathWorks

| | |
|---|---|
| `www.mathworks.com` | Web |
| `comp.soft-sys.matlab` | Newsgroup |
| `www.mathworks.com/contact_TS.html` | Technical Support |

| | |
|---|---|
| `suggest@mathworks.com` | Product enhancement suggestions |
| `bugs@mathworks.com` | Bug reports |
| `doc@mathworks.com` | Documentation error reports |
| `service@mathworks.com` | Order status, license renewals, passcodes |
| `info@mathworks.com` | 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 1990–2007 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, and SimBiology, SimEvents, and SimHydraulics are 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) |
| September 2006 | Online only | Revised for Simulink 6.5 (Release 2006b) |
| March 2007 | First printing | Revised for Simulink 6.6 (Release 2007a) |

# Contents

## 1 Introduction


**What Is Simulink?** . . . . . 1-2
Tool for Model-Based Design . . . . . 1-2
Tool for Simulation . . . . . 1-3
Tool for Analysis . . . . . 1-3

**Related Products** . . . . . 1-5


## 2 Running a Simulink Demo


**Running a Demo Model** . . . . . 2-2

**Anatomy of the Demo Model** . . . . . 2-13

**Other Useful Demos** . . . . . 2-17


## 3 Working with a Simple Model


**About the Simple Model** . . . . . 3-2

**Creating a Simple Model** . . . . . 3-3
Creating an Empty Model . . . . . 3-3
Adding Blocks . . . . . 3-5
Connecting the Blocks . . . . . 3-9

**Simulating the Model** . . . . . 3-13

Setting Simulation Options . . . . . 3-13
Running the Simulation . . . . . 3-15

## 4 Working with Simulink Preferences

**Setting Simulink Preferences** . . . . . 4-2

**About Simulink Preferences** . . . . . 4-5
General Simulink Preferences . . . . . 4-5
Simulink Fonts Preferences . . . . . 4-8
Simulink Simulation Preferences . . . . . 4-9

## Index

1

# Introduction


| | |
|---|---|
| What Is Simulink? (p. 1-2) | Introduces the features and uses of Simulink |
| Related Products (p. 1-5) | Directs you to products that extend the capabilities of Simulink |

# What Is Simulink?

Simulink® is software 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.

Simulink enables you to pose a question about a system, model it, and see what happens. With Simulink, you can easily build models from scratch, or take an existing model and add to it. Thousands of engineers around the world use Simulink to model and solve real problems in a variety of industries. The following topics highlight key aspects of Simulink:

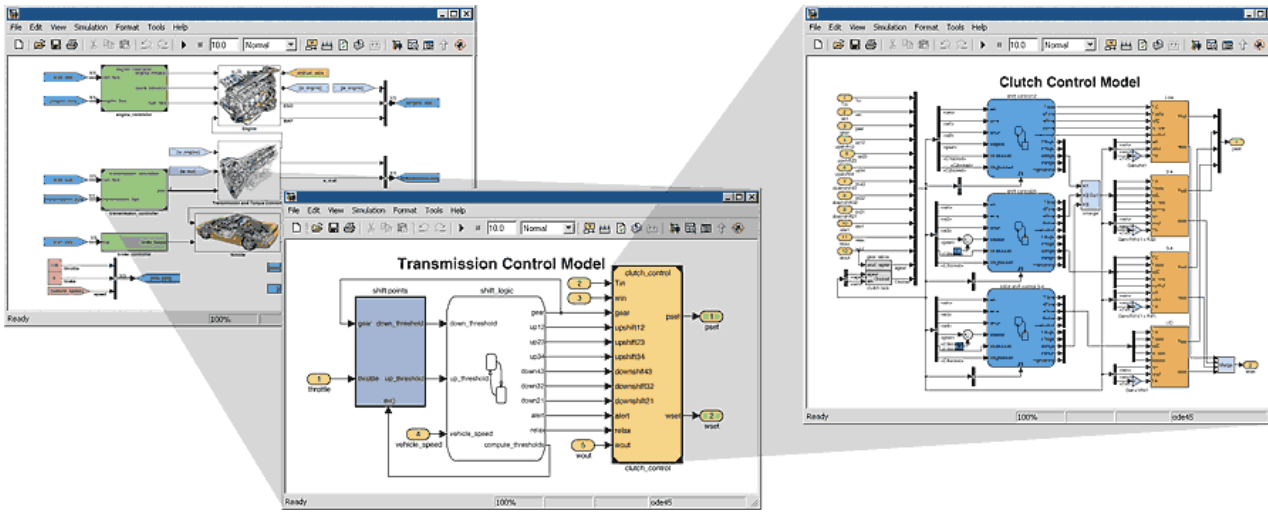

- "Tool for Model-Based Design" on page 1-2
- "Tool for Simulation" on page 1-3
- "Tool for Analysis" on page 1-3


## 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 "Other Useful Demos" on page 2-17.

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 interactive graphical environment is a far cry from 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*.

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.




## Tool for Simulation

After you define a model, you can simulate it, using a choice of mathematical integration methods, either from the Simulink menus or by entering commands in the MATLAB® Command Window. The menus are convenient for interactive work, while the command line is 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 runs. 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.

## Tool for Analysis

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. 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`.

2

# Running a Simulink Demo


| | |
|---|---|
| Running a Demo Model (p. 2-2) | Example of how to simulate a Simulink demo model |
| Anatomy of the Demo Model (p. 2-13) | Explains the infrastructure of the demo model |
| Other Useful Demos (p. 2-17) | Shows how to find other demos that illustrate key Simulink concepts and features |

# Running a Demo Model

Simulink includes a demo program that simulates the thermodynamics of a house: `sldemo_househeat`. You can use the system in this model to acquaint yourself with Simulink and explore some of its features.

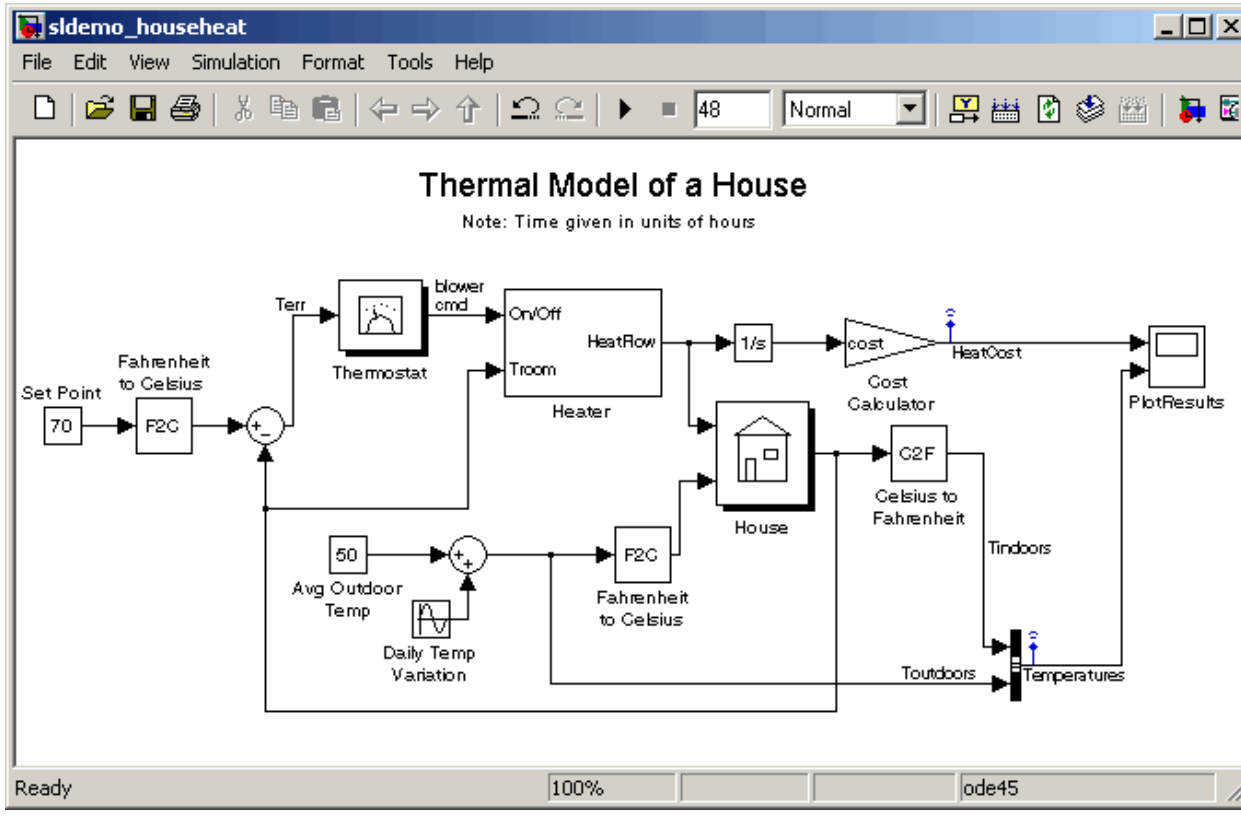
The system models the outdoor environment, the thermal characteristics of the house, and the house heating system. In the demo, the thermostat is set to 70 degrees Fahrenheit. The system models fluctuations in outdoor temperature by applying a sine wave with amplitude of 15 degrees to a base temperature of 50 degrees.

Simulate the demo model and observe how the thermostat setting and outdoor environment affect the indoor temperature and the cumulative heating cost. Follow these steps:

1. Start MATLAB. If you're not sure how to do this, see "Starting MATLAB" in *Getting Started with MATLAB*.

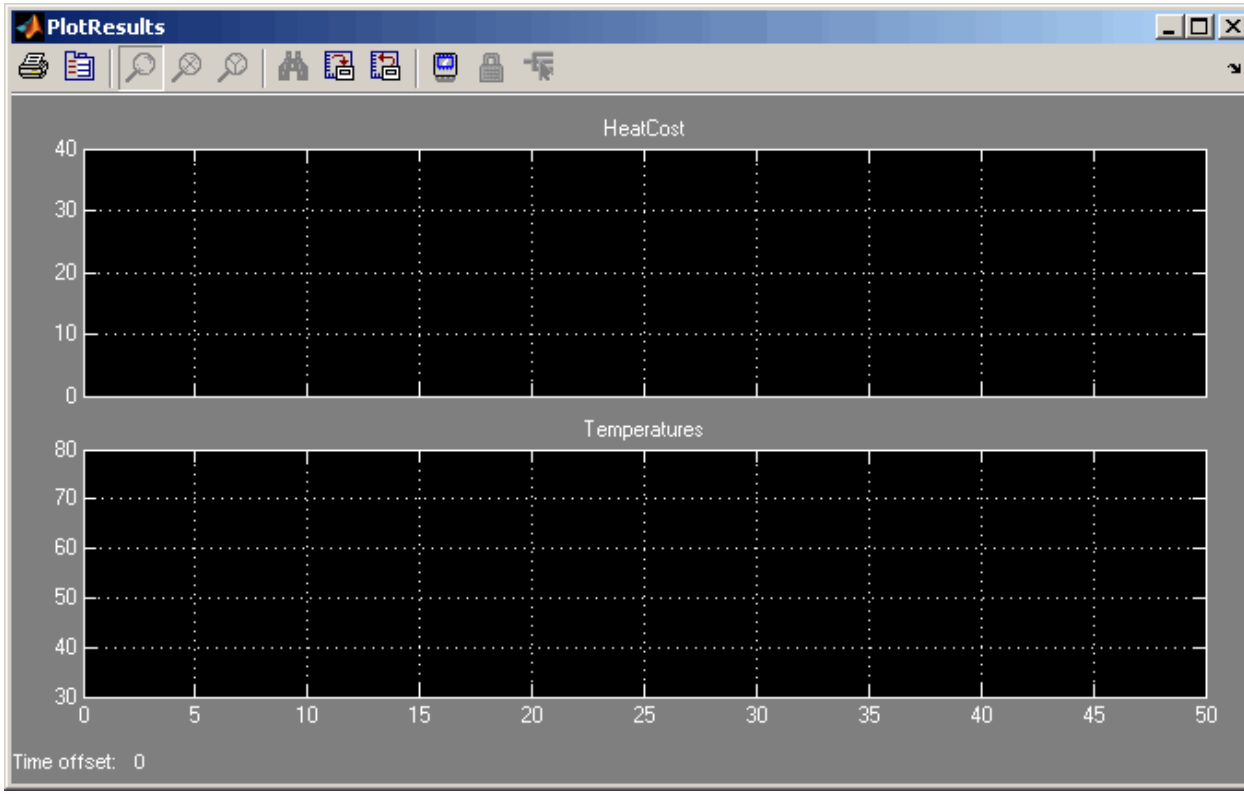
2. In the MATLAB Command Window, type `sldemo_househeat` to open the demo model.

   This command starts Simulink and opens the `sldemo_househeat` model.


sldemo_househeat
File Edit View Simulation Format Tools Help
48
Normal
Thermal Model of a House
Note: Time given in units of hours
Terr
blower cmd
On/Off
HeatFlow
Troom
Thermostat
Heater
1/s
cost
HeatCost
Cost Calculator
PlotResults
Set Point
70
Fahrenheit to Celsius
F2C
House
C2F
Celsius to Fahrenheit
Tindoors
50
Avg Outdoor Temp
Daily Temp Variation
F2C
Fahrenheit to Celsius
Toutdoors
Temperatures
Ready
100%
ode45

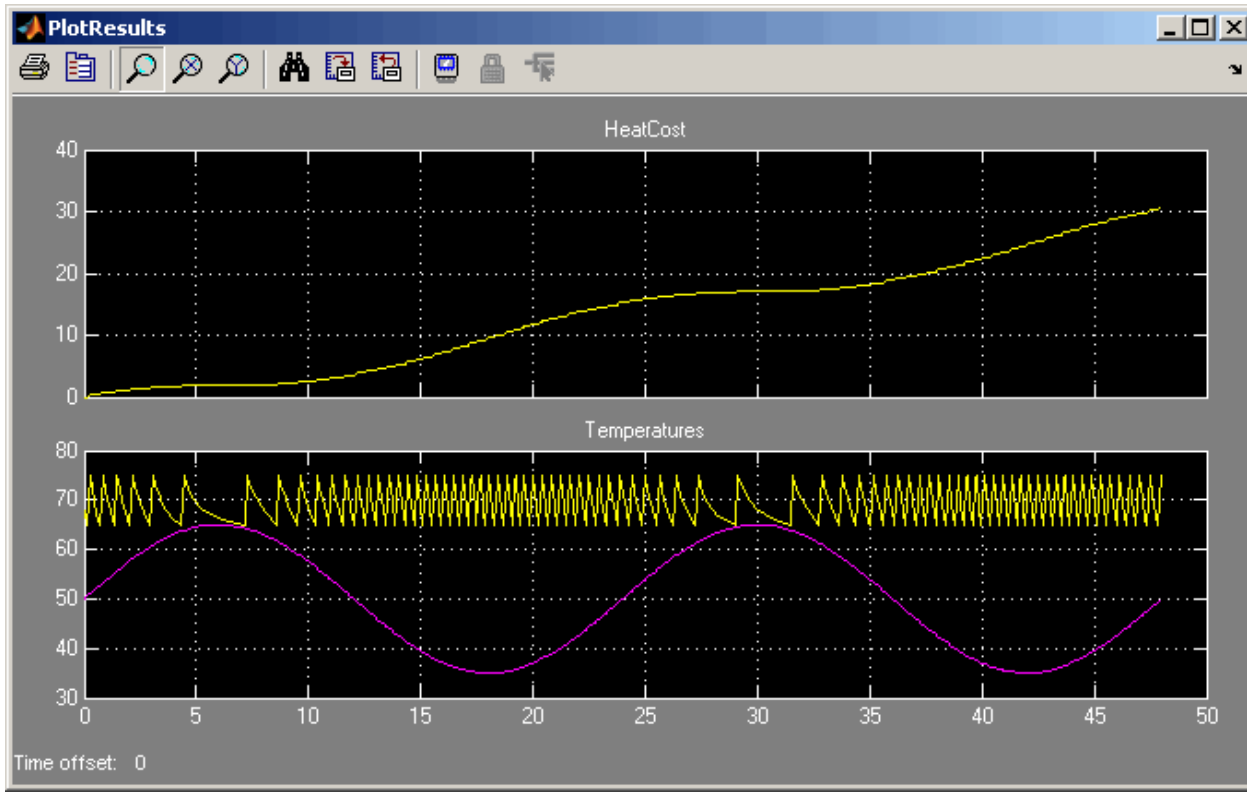
3 In the demo model window, double-click the Scope block named `PlotResults`.

Simulink opens a Scope window that contains two axes labeled “HeatCost” and “Temperatures.” It will display graphic output much as an actual oscilloscope does.



**4** Start the simulation. From the demo model window's **Simulation** menu, select **Start**.

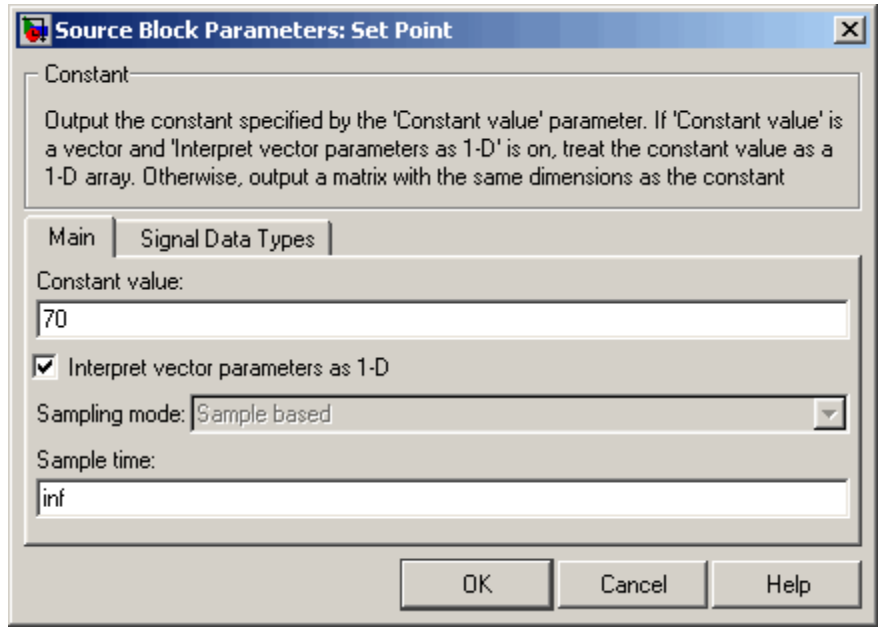
Simulink simulates the model. As the simulation runs, the cumulative heating cost appears on the "HeatCost" axes. The indoor and outdoor temperatures appear on the "Temperatures" axes as yellow (top) and magenta (bottom) signals, respectively.



5 Change the thermostat setting to 68 degrees Fahrenheit and observe how the model responds.

a In the demo model window, double-click the Constant block named `Set Point`.

Simulink opens the parameter dialog box for that block.



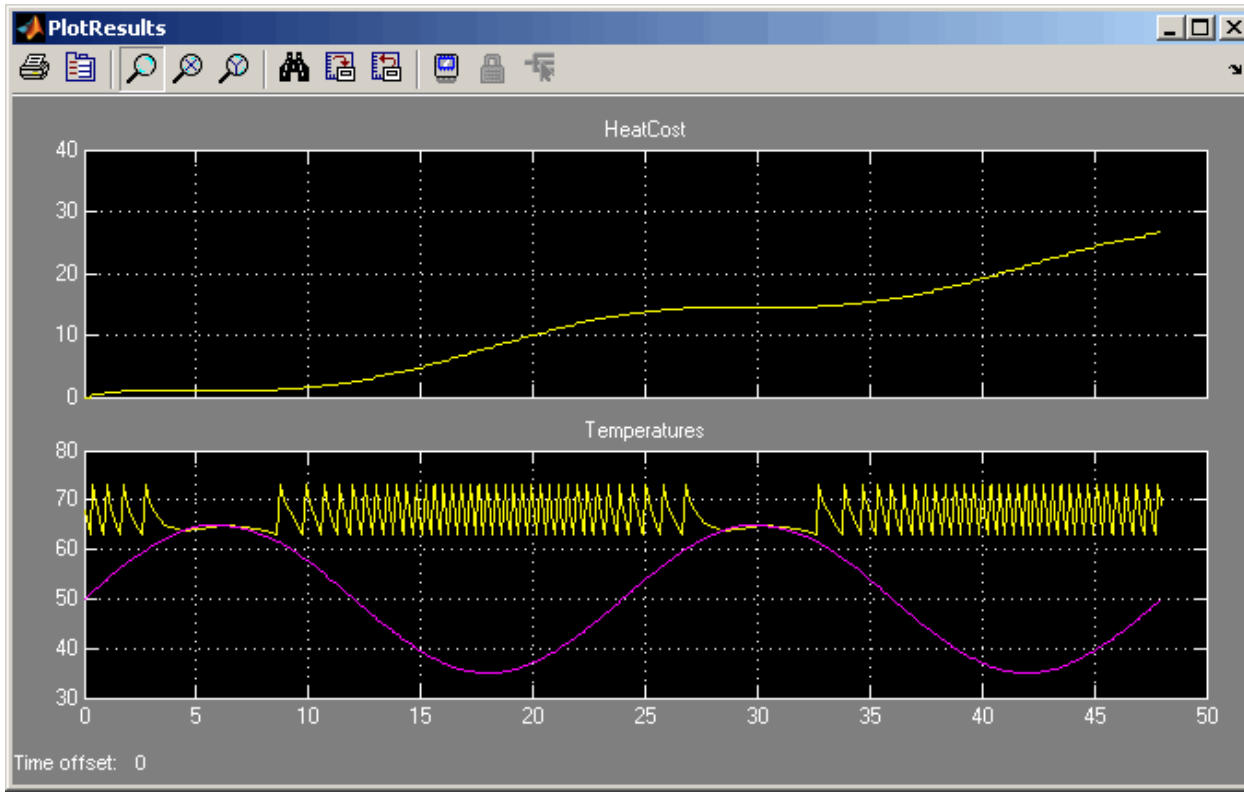

b In the **Constant value** box, enter `68`.

c Click the **OK** button.

Simulink applies your changes and closes the Constant block parameter dialog box.

**d** Rerun the simulation. From the demo model window's **Simulation** menu, select **Start**.

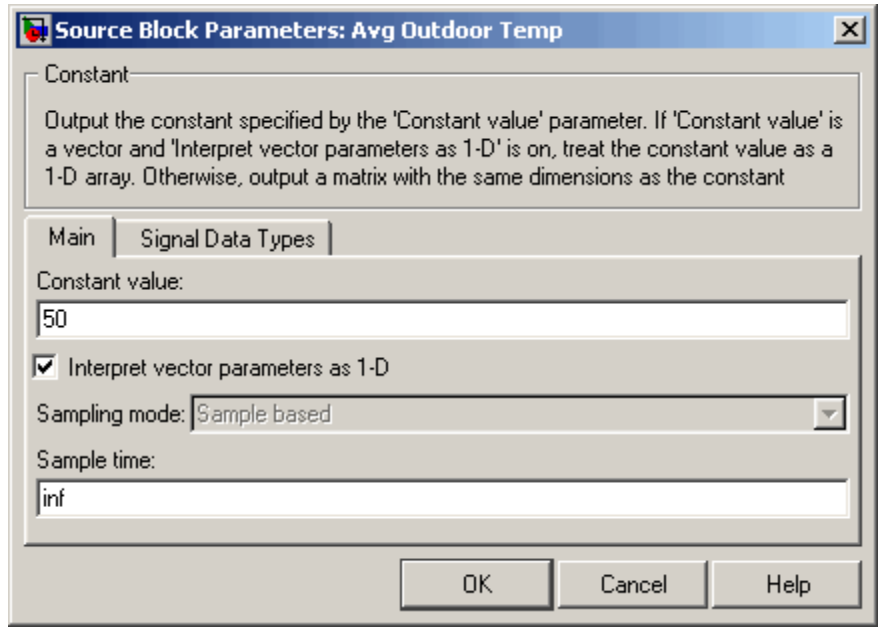
Simulink simulates the model. In the Scope window, notice that a lower thermostat setting reduces the cumulative heating cost.



6 Change the average outdoor temperature to 45 degrees Fahrenheit and observe how the model responds.

a In the demo model window, double-click the Constant block named `Avg Outdoor Temp`.

Simulink opens the parameter dialog box for that block.



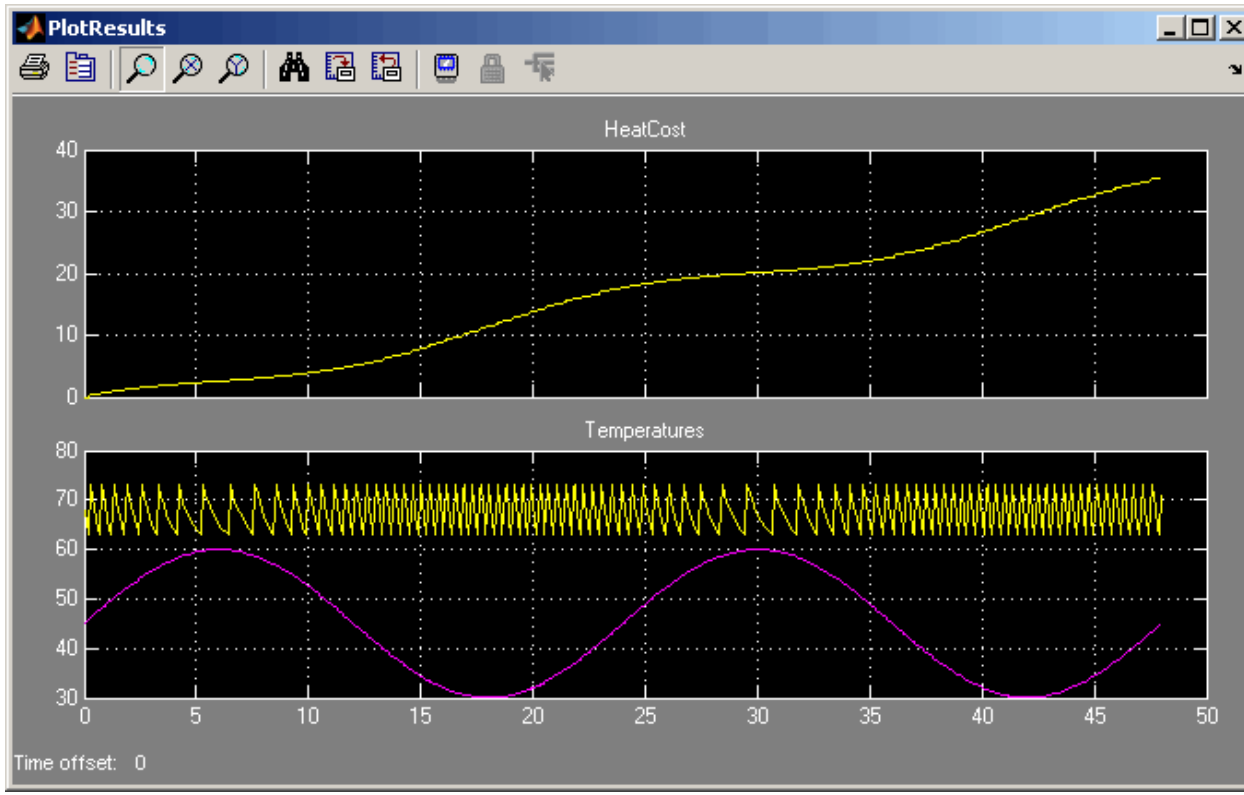

b In the **Constant value** box, enter `45`.

c Click the **OK** button.

Simulink applies your changes and closes the Constant block parameter dialog box.

**d** Rerun the simulation. From the demo model window's **Simulation** menu, select **Start**.

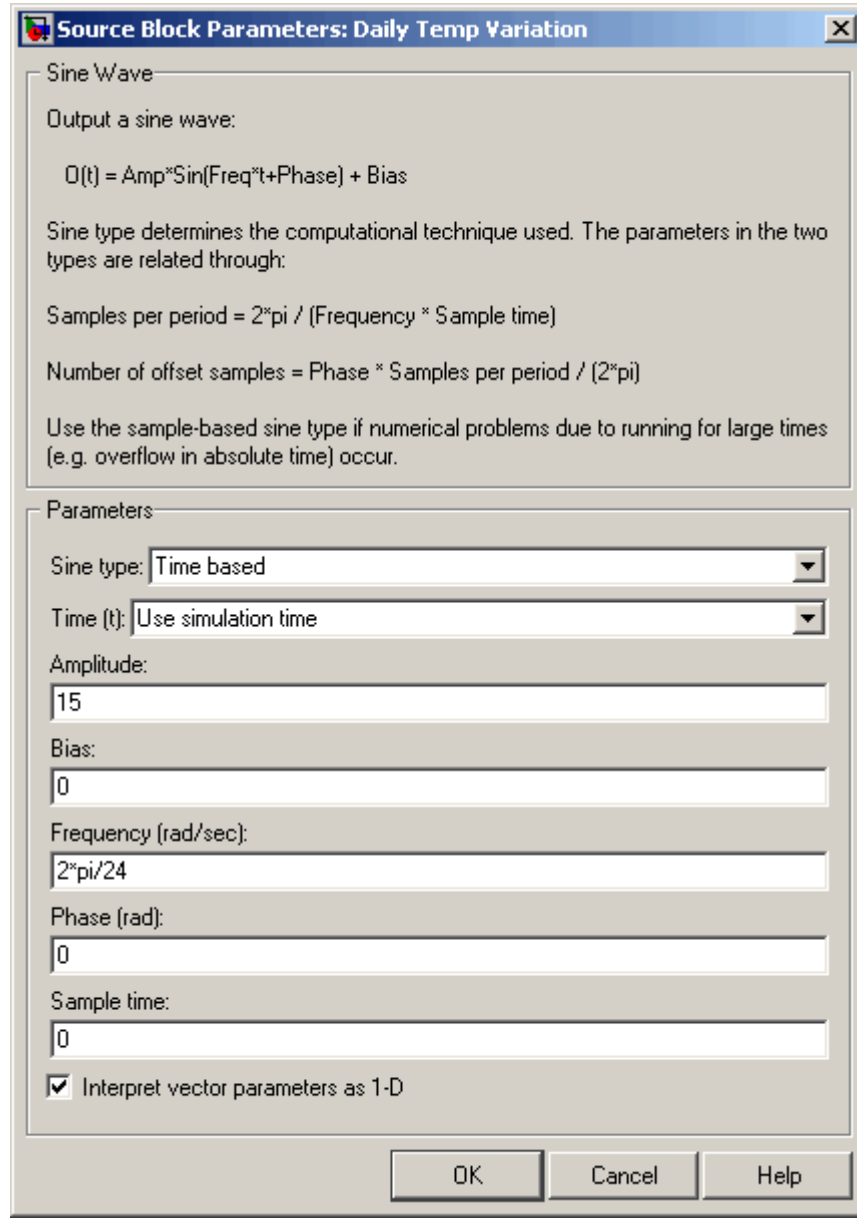
Simulink simulates the model. In the Scope window, notice that a colder outdoor temperature increases the cumulative heating cost.




**7** Change the daily temperature variation to 5 degrees Fahrenheit and observe how the model responds.

**a** In the demo model window, double-click the Sine Wave block named `Daily Temp Variation`.

Simulink opens the parameter dialog box for that block.


Source Block Parameters: Daily Temp Variation
Sine Wave
Output a sine wave:
O(t) = Amp*Sin(Freq*t+Phase) + Bias
Sine type determines the computational technique used. The parameters in the two types are related through:
Samples per period = 2*pi / (Frequency * Sample time)
Number of offset samples = Phase * Samples per period / (2*pi)
Use the sample-based sine type if numerical problems due to running for large times (e.g. overflow in absolute time) occur.
Parameters
Sine type: Time based
Time (t): Use simulation time
Amplitude:
15
Bias:
0
Frequency (rad/sec):
2*pi/24
Phase (rad):
0
Sample time:
0
Interpret vector parameters as 1-D
OK
Cancel
Help

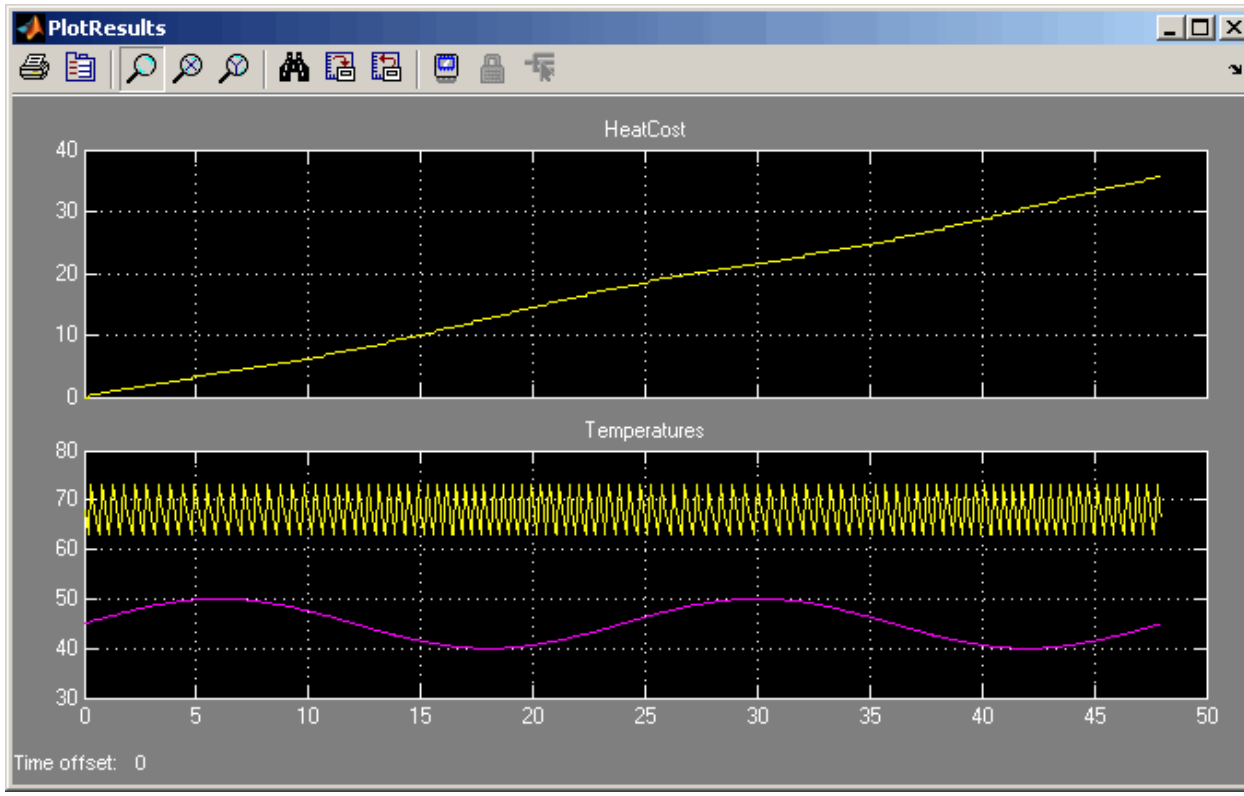
**b** In the **Amplitude** box, enter `5`.

**c** Click the **OK** button.

Simulink applies your changes and closes the Sine Wave block parameter dialog box.

**d** Rerun the simulation. From the demo model window's **Simulation** menu, select **Start**.

Simulink simulates the model. In the Scope window, notice that a stabler outdoor temperature alters the frequency with which the heater operates.

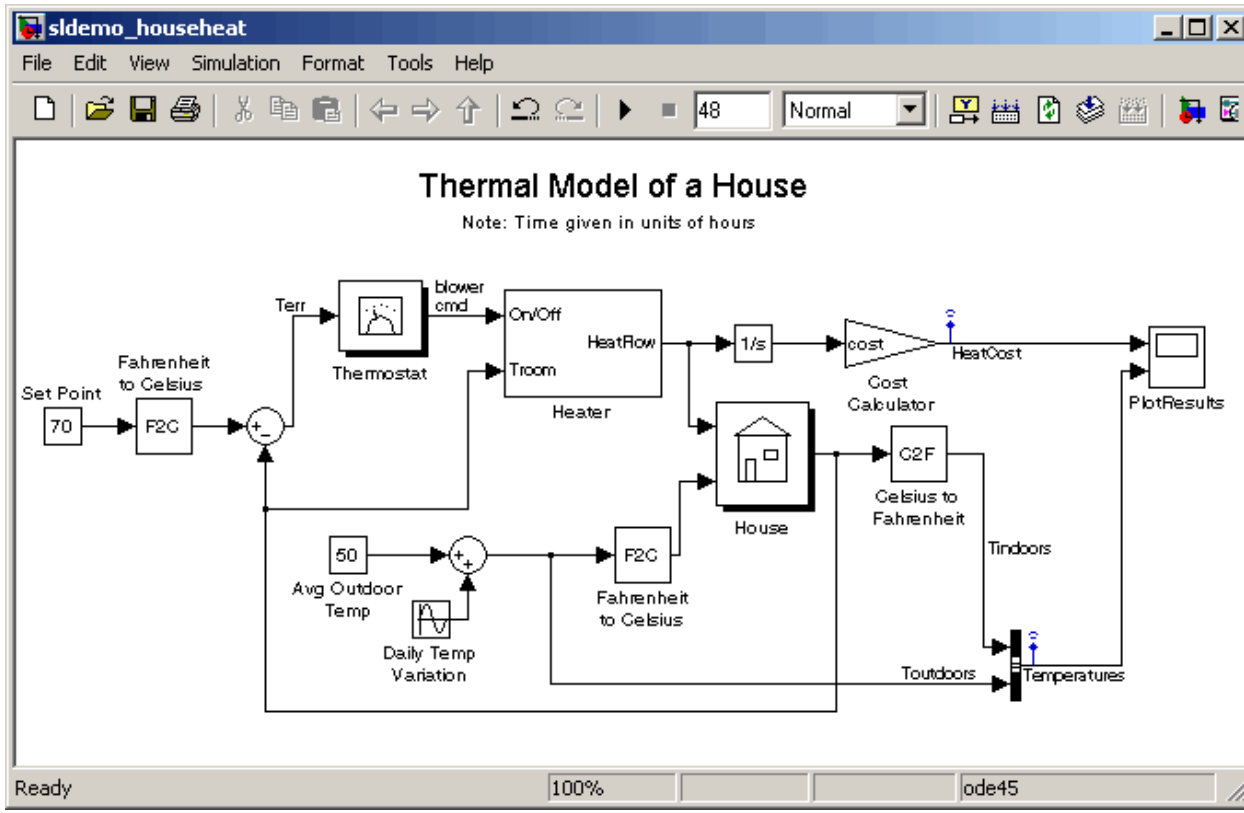


8 You might like to see what happens when you specify other values for the system and rerun the simulation. When you're finished working with the demo, close the model. From the demo model window's **File** menu, select **Close**.

# Anatomy of the Demo Model

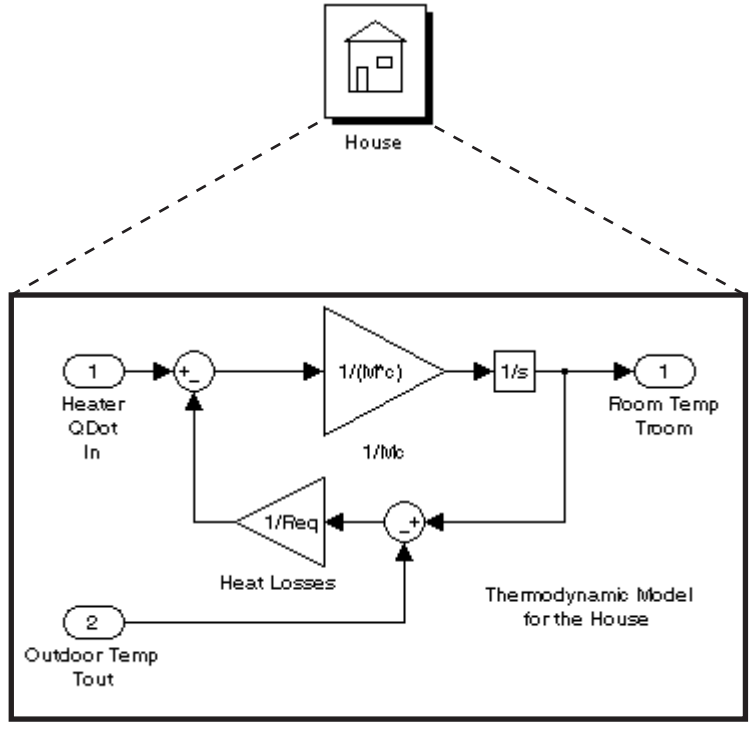
The `sldemo_househeat` demo model uses the following *subsystems* to model the thermodynamics of a house:

- Thermostat
- Heater
- House
- Fahrenheit to Celsius
- Celsius to Fahrenheit



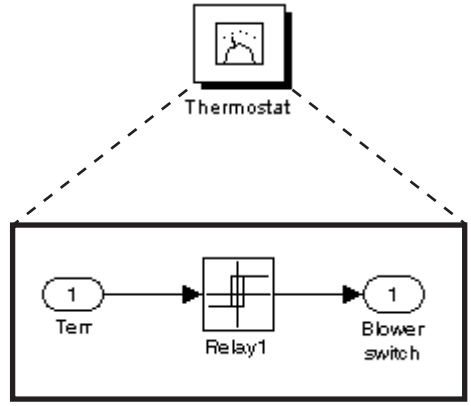
These subsystems simplify the block diagram's appearance, create reusable components, and customize the appearance of blocks. A subsystem is a hierarchical grouping of blocks that a Subsystem block encapsulates.

Subsystems can be complex and contain many blocks that might otherwise clutter a diagram. For example, consider the `House` subsystem. It receives heat flow and external temperature as inputs, which it uses to compute the current room temperature. In the demo model window, double-click the `House` block to see the underlying blocks in that subsystem.




Contents of House subsystem

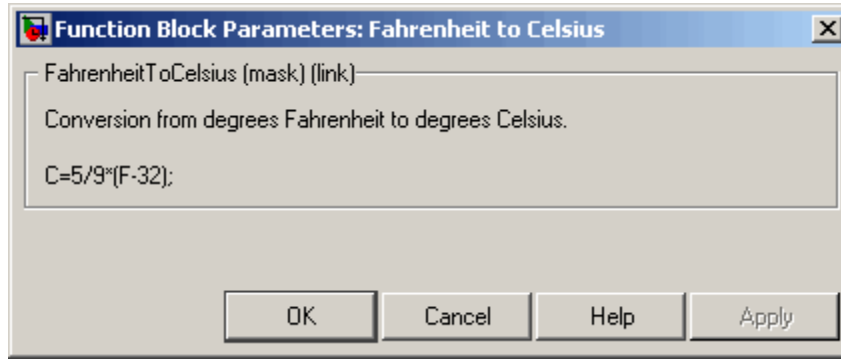
Then again, subsystems can be simple and contain only a few blocks. For example, the `Thermostat` subsystem models the operation of a thermostat, determining when the heating system is on or off. In the demo model window, double-click the `Thermostat` block to see the underlying blocks in that subsystem.



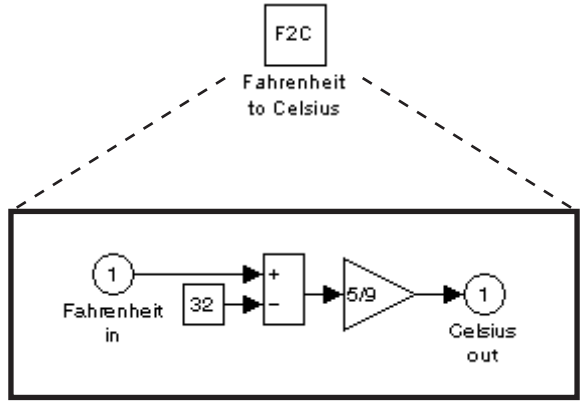

Contents of Thermostat subsystem

Subsystems are reusable, enabling you to implement an algorithm once and use it multiple times. For example, the model contains two instances of identical subsystems named `Fahrenheit to Celsius`. These subsystems convert the inside and outside temperatures from degrees Fahrenheit to degrees Celsius.

You can customize a subsystem's appearance using a process known as *masking*. Masking a subsystem allows you to specify a unique icon and dialog box for the Subsystem block. For example, the `House` and `Thermostat` subsystems display custom icons that depict each of those objects. Also, the conversion subsystems display custom dialog boxes when you double-click them.




To view the underlying blocks in a conversion subsystem, right-click the subsystem and select **Look Under Mask**. The contents of a `Fahrenheit to Celsius` subsystem are shown here:




Contents of Fahrenheit to Celsius subsystem

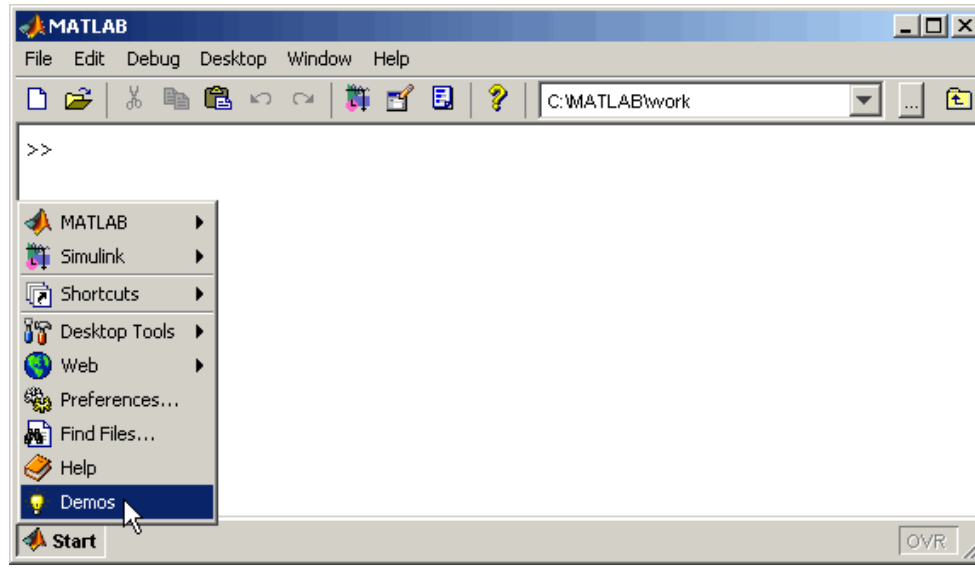
For more information about working with subsystems and creating masks, see "Creating Subsystems" and "Creating Block Masks" in *Using Simulink*.

# Other Useful Demos

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

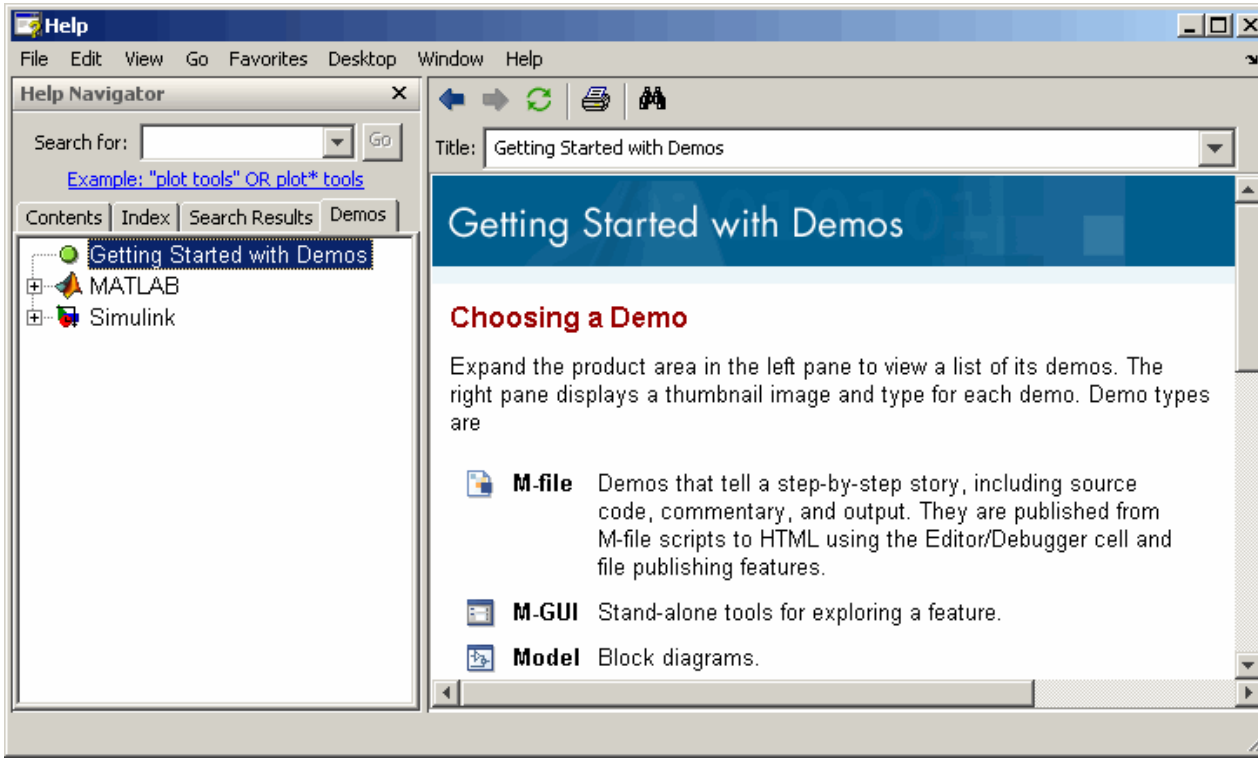
1 On the bottom left corner of the MATLAB Command Window, click the **Start** button.

The **Start** menu appears.




2 From the **Start** menu, select **Demos**.

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




**3** In the **Demos** pane, expand the **Simulink** node.

The node expands to show groups of Simulink demos.

Use the browser to navigate to particular demos. The browser displays explanations of each demo and includes a link to the demo model itself. Click the link to access the demo model.

3

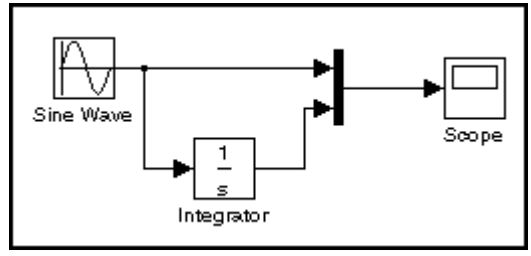
# Working with a Simple Model


| | |
|---|---|
| About the Simple Model (p. 3-2) | Introduces the simple Simulink model you will create and simulate |
| Creating a Simple Model (p. 3-3) | Example of how to create a simple Simulink model |
| Simulating the Model (p. 3-13) | Example of how to use Simulink to simulate a model |

# About the Simple Model

This chapter shows you how to create a simple model using many of the model construction techniques that you will use to create your own models. Afterward, the chapter instructs you to simulate the model that you construct. The instructions for constructing and simulating the example model are brief. However, all the tasks are described in more detail in *Using Simulink*.

In the sections that follow, you will construct a simple model that integrates a sine wave and displays the result along with the sine wave. When completed, the block diagram that constitutes your example model should appear similar to this.



# Creating a Simple Model

The following sections show you how to model a simple dynamic system using Simulink:

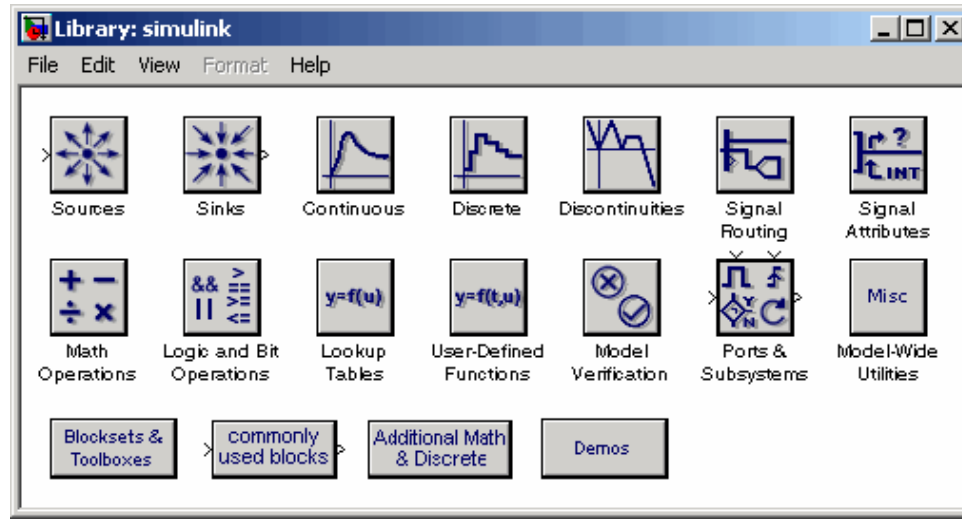

- "Creating an Empty Model" on page 3-3
- "Adding Blocks" on page 3-5
- "Connecting the Blocks" on page 3-9


## Creating an Empty Model

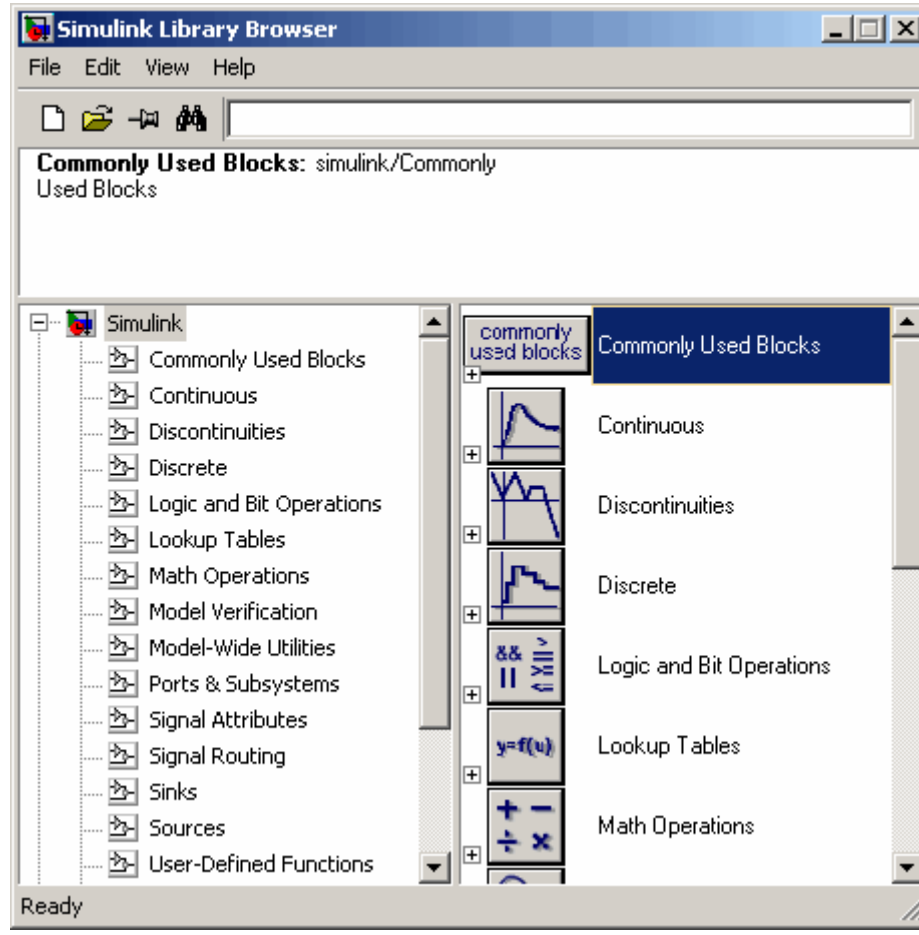
To create an empty model:

1 Start Simulink. In the MATLAB Command Window, enter `simulink`.

- On UNIX, the Simulink library window appears.



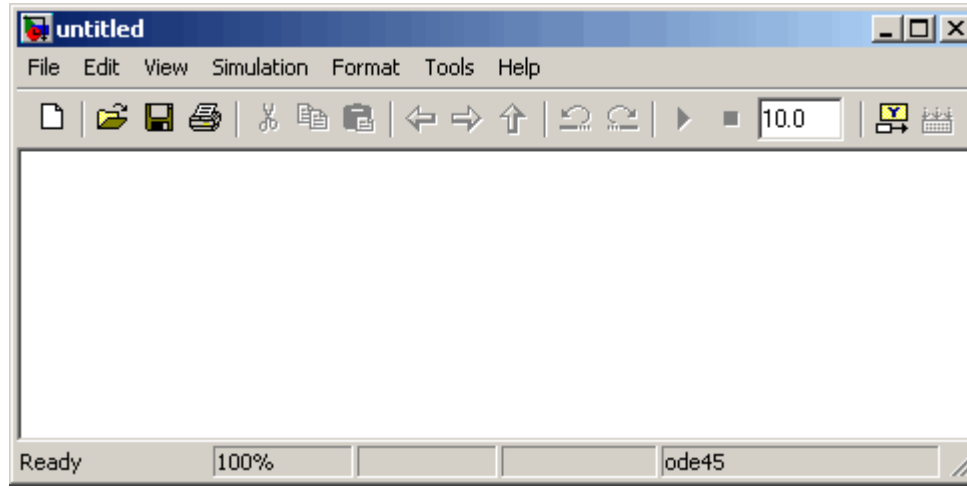
- On Microsoft Windows, the Simulink Library Browser appears.




**2** Create a new model.

- On UNIX, from the Simulink library window's **File** menu, select **New > Model**.
- On Windows, click the **New model** button on the Simulink Library Browser's toolbar.

Simulink displays an empty model window.



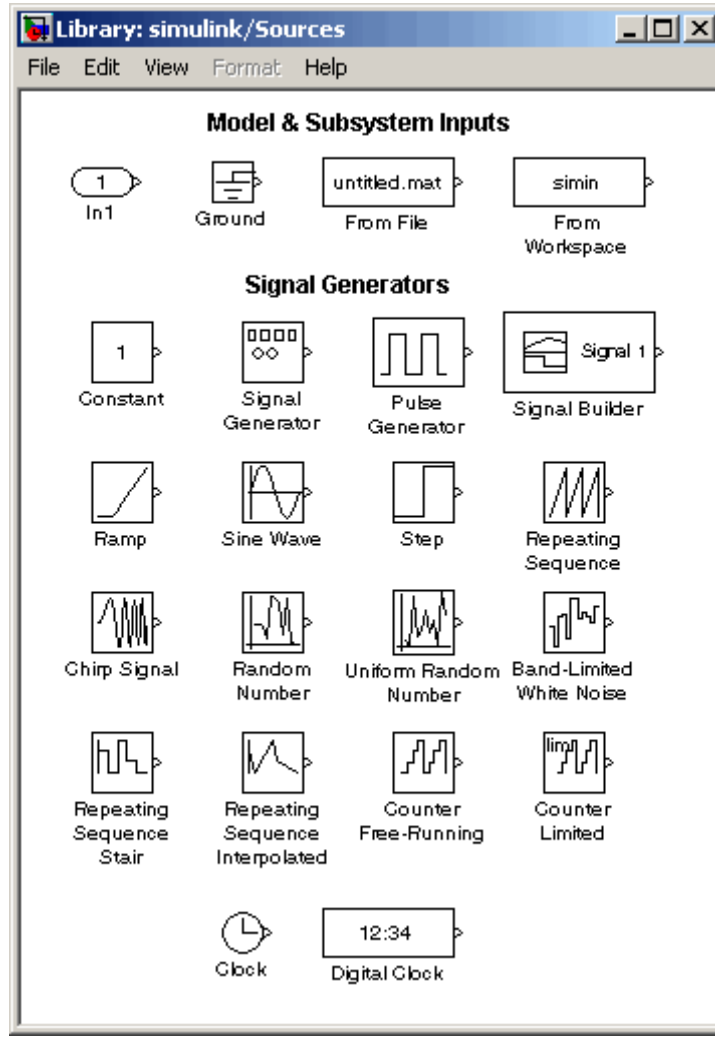

## Adding Blocks

To construct the example model that this chapter describes, you need to copy blocks into the model window that the previous section instructed you to create (see “Creating an Empty Model” on page 3-3). You can copy blocks from either the Library Browser (Windows only) or a library window (UNIX and Windows). The following steps describe how to add blocks to your example model using each of these methods:

1 Locate the Sine Wave block in the Sources library.

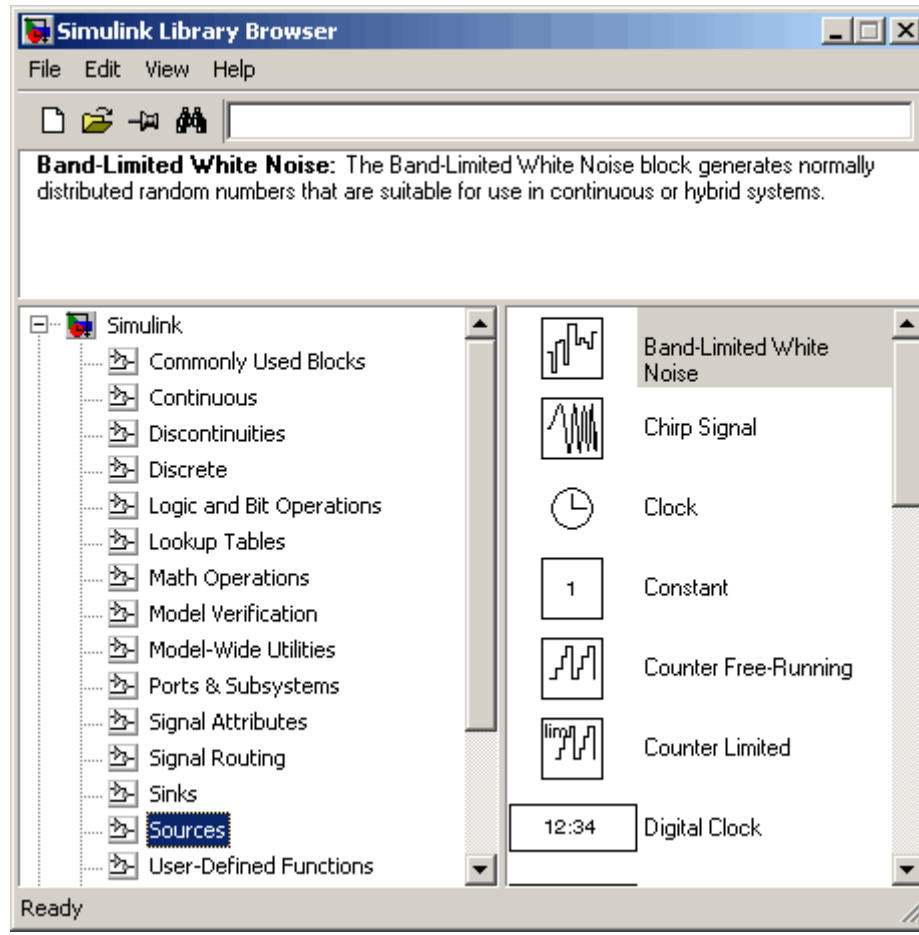
- On UNIX, in the Simulink library window, double-click the Sources library.

  Simulink displays the Sources library window.




- On Windows, in the left side of the Simulink Library Browser under the **Simulink** node, select the **Sources** library.

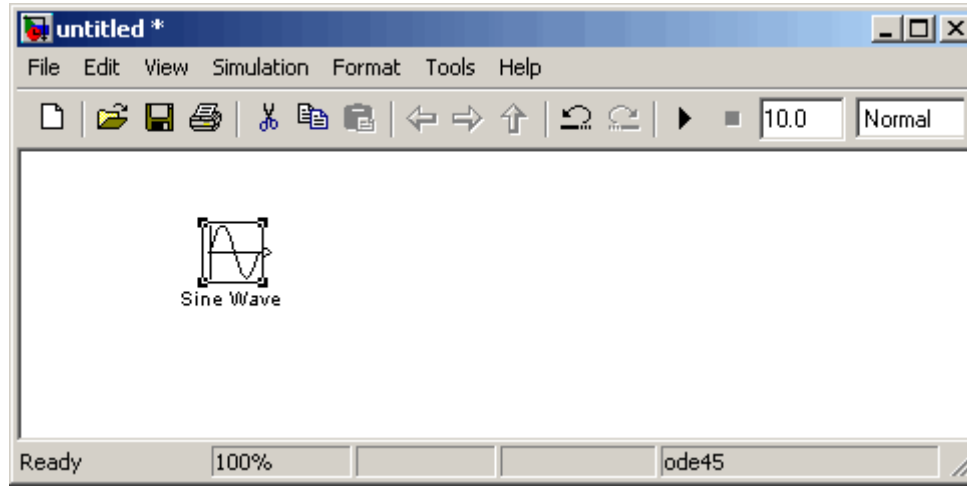
  The right pane of the Simulink Library Browser displays the contents of the Sources library.




**2** Copy the Sine Wave block to your model window.

- On UNIX, click and drag the Sine Wave block from the Sources library to your model window.
- On Windows, from the right pane of the Simulink Library Browser, click and drag the Sine Wave block to your model window.

A copy of the Sine Wave block appears in your model window.




3 Continue to copy the following blocks from their respective libraries into your model window:

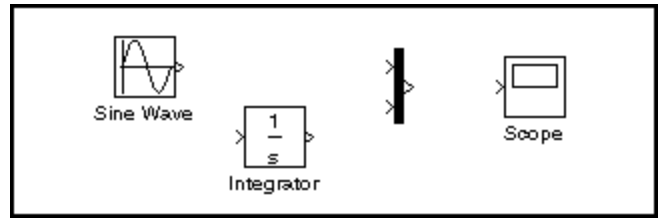
- Scope block (from the Sinks library)
- Integrator block (from the Continuous library)
- Mux block (from the Signal Routing library)

---

**Note** 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.

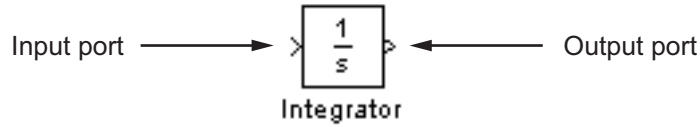
---

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



## Connecting the Blocks

Now it's time to connect the blocks. 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*.




The following sections describe how you connect blocks by drawing lines from their output ports to their input ports or by drawing branch lines to their input ports:

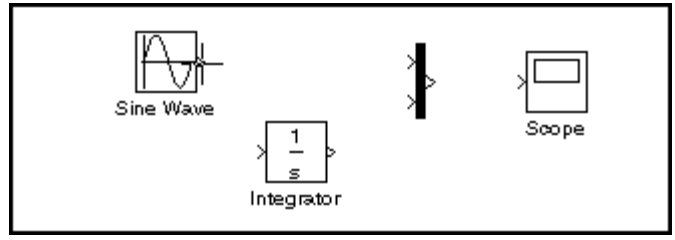

- "Drawing a Line Between Blocks" on page 3-9
- "Drawing a Branch Line" on page 3-11


### Drawing a Line Between Blocks

To connect blocks, you draw lines from their output ports to their input ports. For instance, to connect the Sine Wave block to the top input port of the Mux block in your example model:

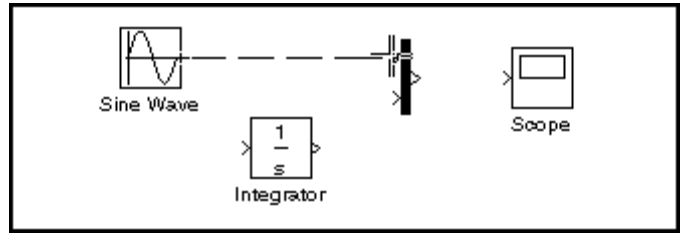
1 Position the pointer over the output port on the right side of the Sine Wave block.

  Notice that the cursor shape changes to crosshairs.



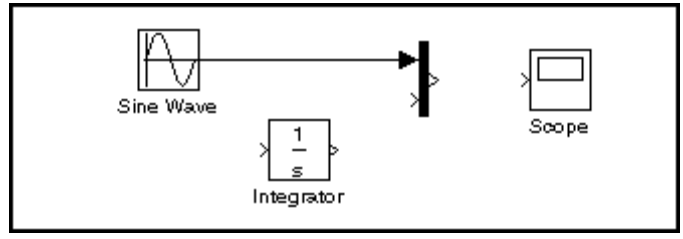
2 Hold down the mouse button and move the cursor to the top input port of the Mux block.

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.




3 Release the mouse button.

Simulink connects the blocks with an arrow that indicates the direction of signal flow.

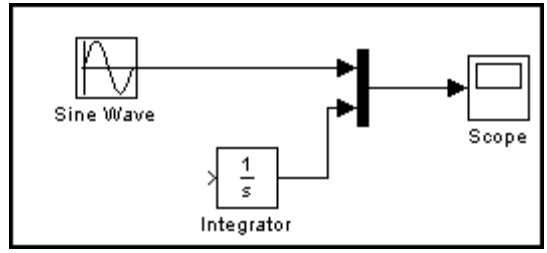



**Note** You can also connect the line to the block by releasing the mouse button while the pointer is over the block. If you do, the line is connected to the input port closest to the cursor's position.

In your example model, use this same procedure to connect the

- Integrator block output port to the bottom input port of the Mux block
- Mux block output port to the Scope block

When you're done, your model should look something like this.

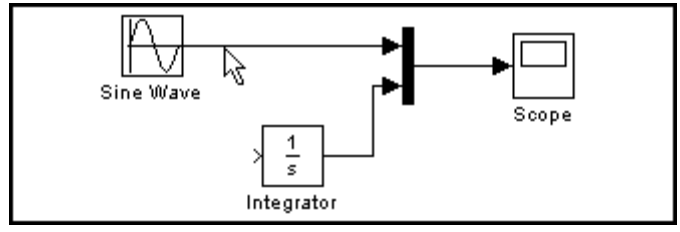



## Drawing a Branch Line

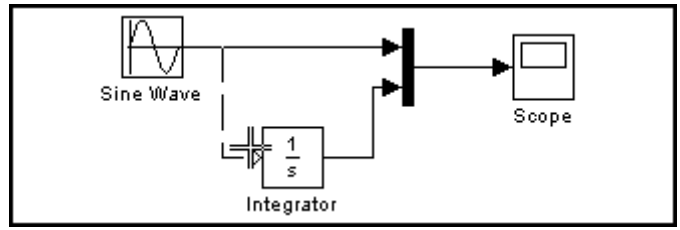
If you look again at the model in "About the Simple Model" on page 3-2, you'll notice that most of the lines connect output ports of blocks to input ports of other blocks. However, one line connects a *line* to the input port of another block. This line, called a *branch line*, connects the Sine Wave 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 lines you just drew (see "Drawing a Line Between Blocks" on page 3-9). To weld a connection to an existing line, follow these steps:

1 Position the pointer *on the line* between the Sine Wave and the Mux block.

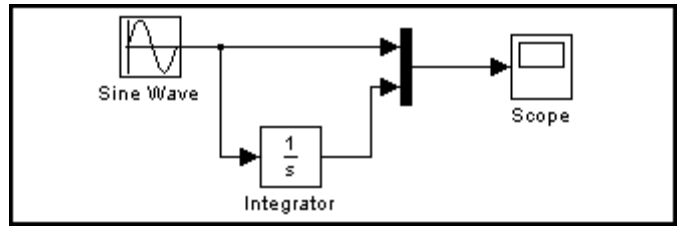


2 Press and hold down the **Ctrl** key. Press the mouse button, then drag the pointer to the Integrator block's input port or over the Integrator block itself.




3 Release the mouse button.

Simulink draws a line between the starting point and the Integrator block's input port.



# Simulating the Model

The following sections describe how to use Simulink to simulate the example model that the previous section instructed you to create (see "Creating a Simple Model" on page 3-3):

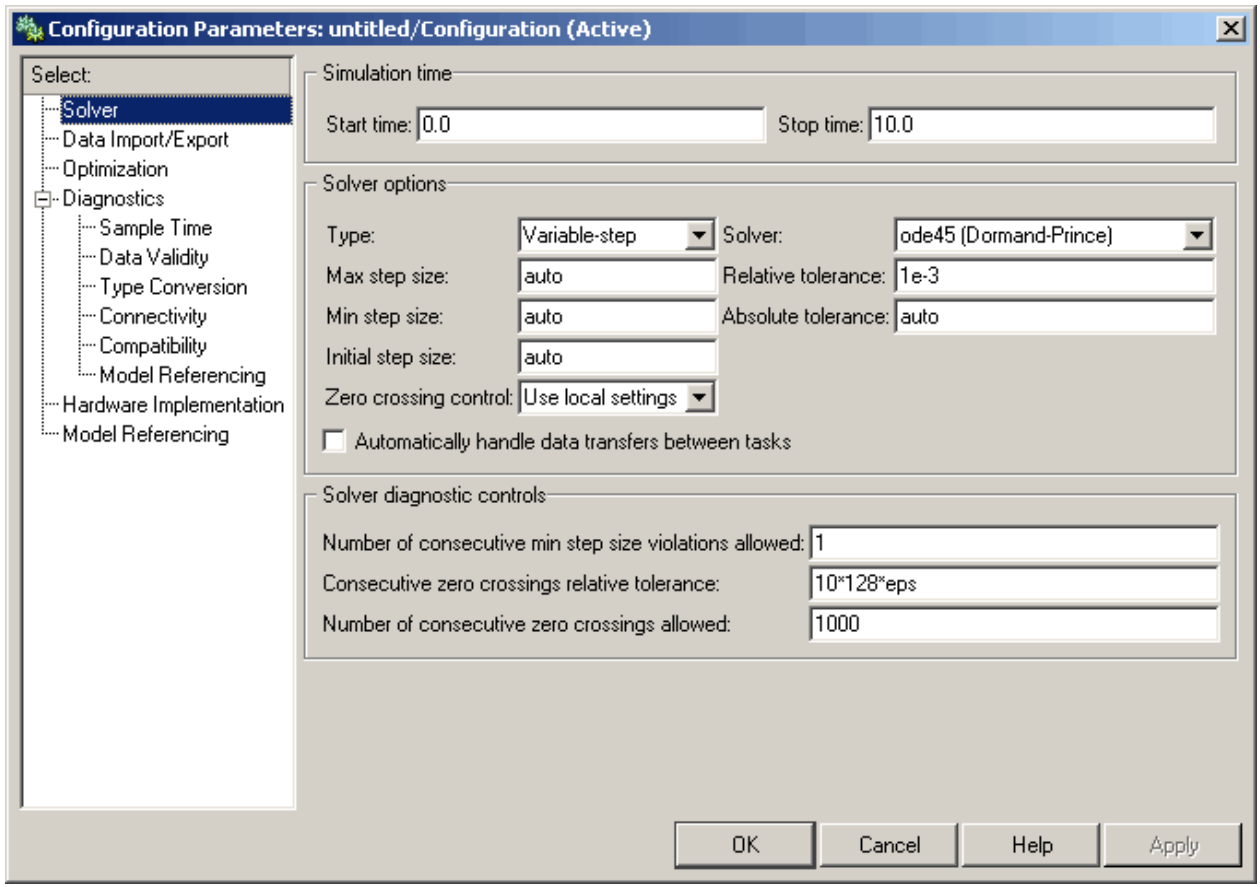

- "Setting Simulation Options" on page 3-13
- "Running the Simulation" on page 3-15


## Setting Simulation Options

Before simulating a model, you may want to set various simulation options, such as the simulation's start and stop time or the type of solver used to solve the model at each simulation time step. You can use the Configuration Parameters dialog box to specify a model's simulation options. For instance, to specify that Simulink should simulate your example model for 20 seconds:

1 From your example model window's **Simulation** menu, select **Configuration Parameters**.

  Simulink displays the Configuration Parameters dialog box. Notice that the **Stop time** parameter is set to `10.0` (its default value).




**2** In the **Stop time** box, enter 20.

**3** Click the **OK** button.

Simulink applies your changes to the parameters and closes the Configuration Parameters dialog box.

## Running the Simulation

Now you are ready to simulate your example model, observe its simulation results, and then save it.

1 From your example model window's **Simulation** menu, select **Start**.

Simulink simulates your model, stopping when it reaches the stop time specified in the Configuration Parameters dialog box or when you select **Stop** from the **Simulation** menu.

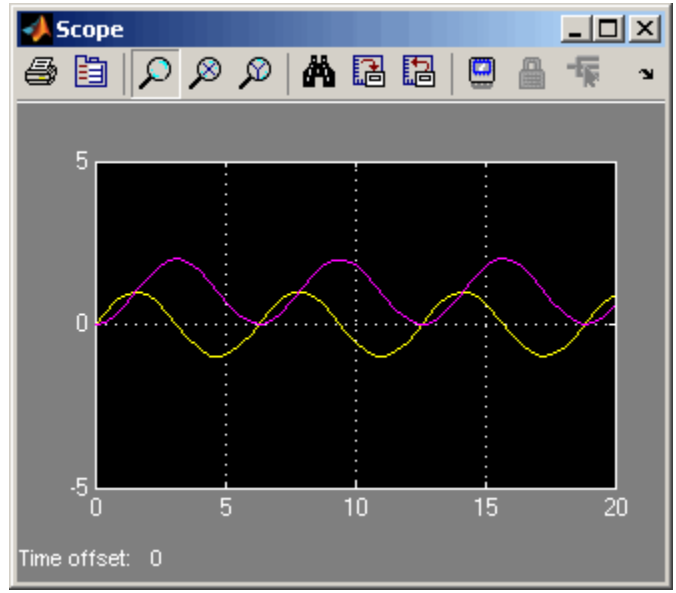
---

**Tip** On Windows, you can click the **Start simulation** button ▶ and **Stop simulation** button ■ in a model window's toolbar to start and stop its simulation.

---

2 In your example model window, double-click the Scope block.

The Scope window displays the simulation results as shown here:



3 From your example model window's **File** menu, select **Save** and enter a file name and location.

Simulink saves your example model using the specified name and location.

4

# Working with Simulink Preferences

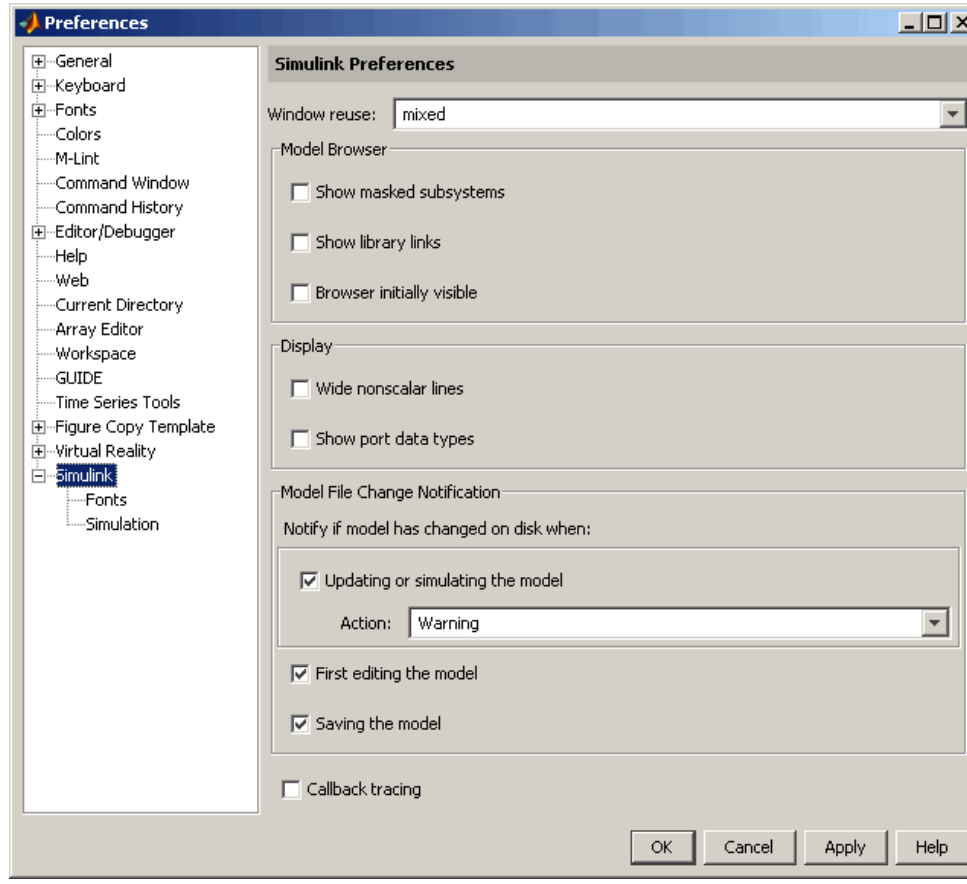

| | |
|---|---|
| Setting Simulink Preferences (p. 4-2) | How to specify default settings for miscellaneous Simulink options |
| About Simulink Preferences (p. 4-5) | Describes the Simulink options for which you can specify default settings |

# Setting Simulink Preferences

To display the Preferences dialog box and specify default settings for particular Simulink options:

1 From the Simulink or MATLAB **File** menu, select **Preferences**.

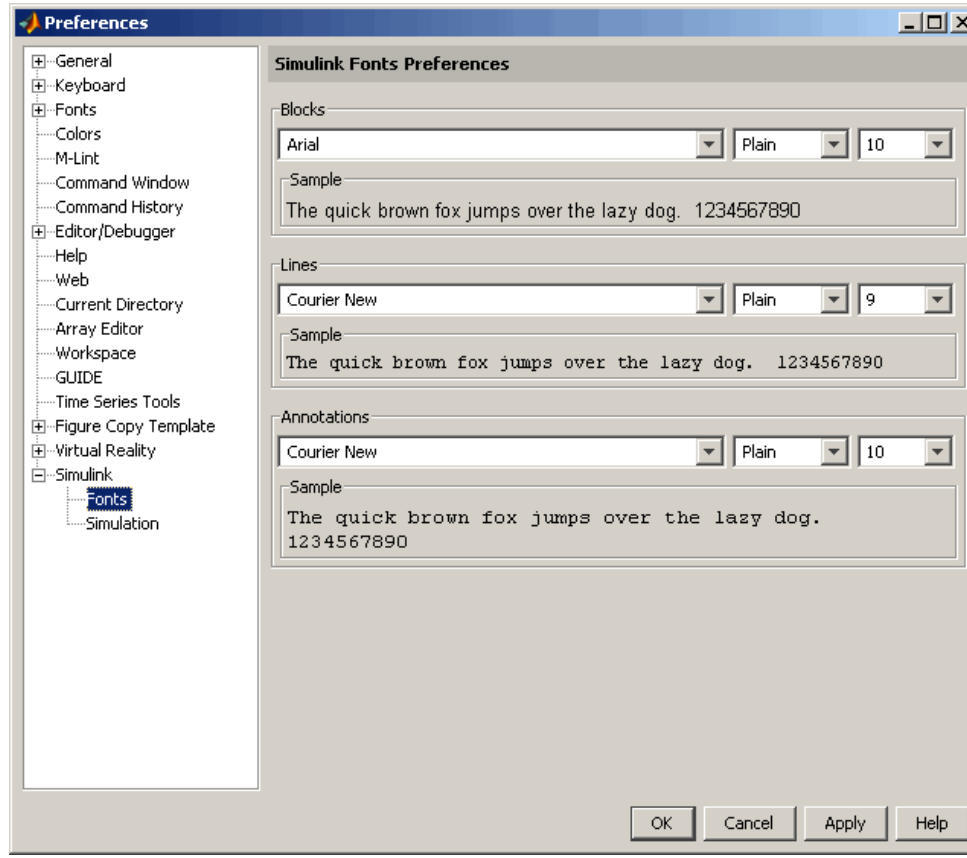
The Preferences dialog box appears.



**2** In the left pane of the Preferences dialog box, select one of the following nodes to access particular Simulink options:

| Preference Node | Simulink Options You Can Specify |
|---|---|
| **Simulink** | Window reuse, model browser, display, model file change notification, and callback tracing (see "General Simulink Preferences" on page 4-5) |
| **Simulink > Fonts** | Fonts for block labels, line labels, and annotations (see "Simulink Fonts Preferences" on page 4-8) |
| **Simulink > Simulation** | Simulation options (see "Simulink Simulation Preferences" on page 4-9) |

The Preferences dialog box displays on its right side the Simulink options associated with the selected node. For example, selecting the **Simulink > Fonts** node displays the **Simulink Fonts Preferences** pane, as shown.




3 On the right side of the Preferences dialog box, specify your desired default settings for the Simulink options displayed.

4 Click either the **OK** or the **Apply** button to apply your changes.

The **OK** button applies all the changes you made and closes the Preferences dialog box. The **Apply** button applies the changes but leaves the Preferences dialog box open so that you can continue to set default options.

For more information about working with the Preferences dialog box, see "Preferences" in *MATLAB Desktop Tools and Development Environment*.

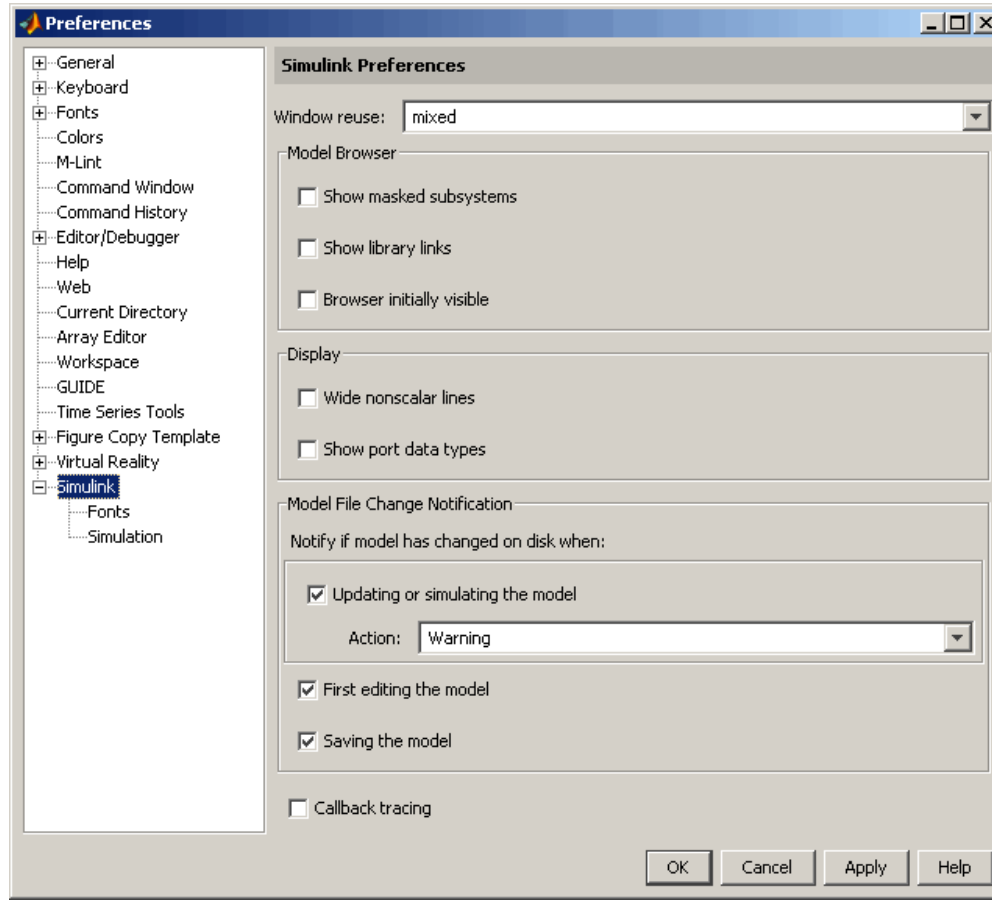
# About Simulink Preferences

The Preferences dialog box allows you to specify default settings for the following types of Simulink options:


- "General Simulink Preferences" on page 4-5
- "Simulink Fonts Preferences" on page 4-8
- "Simulink Simulation Preferences" on page 4-9


## General Simulink Preferences

Selecting the **Simulink** node in the left side of the Preferences dialog box displays the **Simulink Preferences** pane on the right side of the dialog box.

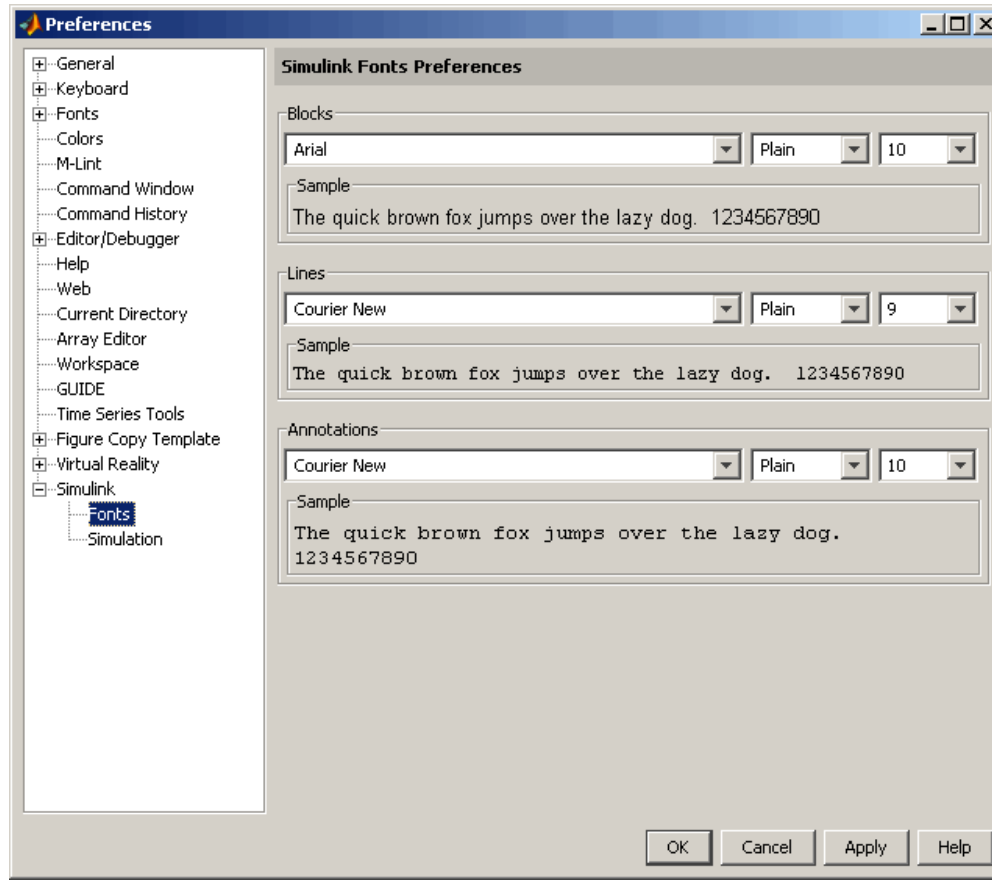

Preferences
General
Keyboard
Fonts
Colors
M-Lint
Command Window
Command History
Editor/Debugger
Help
Web
Current Directory
Array Editor
Workspace
GUIDE
Time Series Tools
Figure Copy Template
Virtual Reality
Simulink
Fonts
Simulation
Simulink Preferences
Window reuse:
mixed
Model Browser
Show masked subsystems
Show library links
Browser initially visible
Display
Wide nonscalar lines
Show port data types
Model File Change Notification
Notify if model has changed on disk when:
Updating or simulating the model
Action:
Warning
First editing the model
Saving the model
Callback tracing
OK
Cancel
Apply
Help

This pane allows you to specify default settings for the following Simulink options:

| Option | Description |
|---|---|
| **Windows reuse** | Specifies whether Simulink uses the current window 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 (see "Display Options" in *Using Simulink*). |
| **Model File Change Notification** | Specifies whether to notify if the model has changed on disk when updating, simulating, editing, or saving the model (see "Model File Change Notification" in *Using Simulink*). This can occur, for example, with source control operations and multiple users. |
| **Callback tracing** | Specifies whether to display the model callbacks that Simulink invokes when simulating a model (see "Using Callback Functions" in *Using Simulink*). |

## Simulink Fonts Preferences

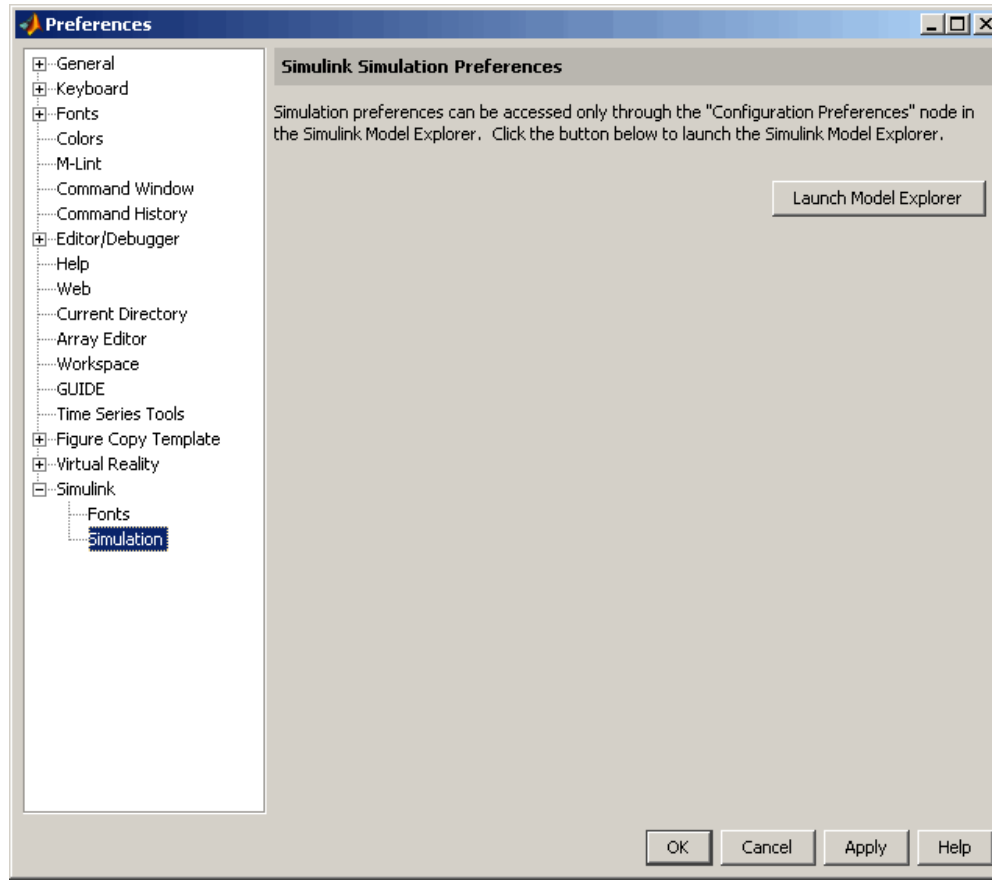
Selecting the **Simulink > Fonts** node in the left side of the Preferences dialog box displays the **Simulink Fonts Preferences** pane on the right side of the dialog box.




This pane contains three sections that allow you to specify your preferred fonts for block and line labels and model annotations, respectively.

## Simulink Simulation Preferences

Selecting the **Simulink > Simulation** node in the left side of the Preferences dialog box displays the **Simulink Simulation Preferences** pane on the right side of the dialog box.




This pane displays a button to start the Model Explorer (see “The Model Explorer” in *Using Simulink*). Use the Model Explorer to set your preferences for options such as simulation start and stop times.

# Index


## B

blocks
  connecting 3-9
  copying 3-5
  masking 2-16
  moving 3-8
  ports 3-9
  subsystems 2-13

## C

Configuration Parameters dialog box 3-13
connecting lines to input ports 3-9
copying blocks to a model 3-5
creating models
  exercise 3-3

## D

demos
  accessing 2-17
  running 2-2

## E

example model 3-2

## F

fonts
  setting preferences 4-8

## I

input port 3-9

## L

lines
  branching a signal 3-11
  carrying the same signal 3-9
  connecting to input ports 3-9

## M

masking 2-16
models
  creating 3-3
  empty 3-3
  new 3-3
  saving 3-16
  simulating 3-13

## O

output port 3-9

## P

preferences 4-1
Preferences dialog box 4-2
product overview 1-2

## R

running demos
  exercise 2-2

## S

simulating models
  exercise 3-13
simulation
  setting preferences 4-9
  specifying options 3-13
  starting 3-15
  stopping 3-15
Simulink
  demos 2-17
  general preferences 4-5
  overview 1-2
  related products 1-5

setting preferences 4-1
starting 3-3
Simulink Library Browser 3-3
Simulink library window 3-3
subsystems 2-13