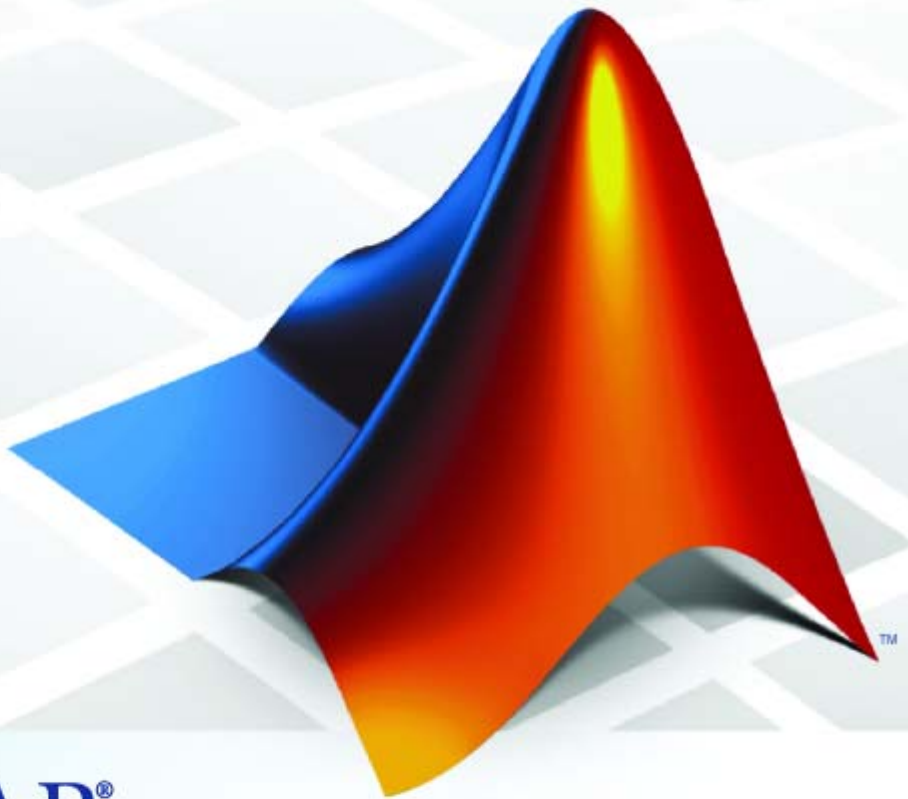


Simulink® Design Optimization™ 1

User's Guide



MATLAB®
& **SIMULINK®**

How to Contact The MathWorks



www.mathworks.com
[comp.soft-sys.matlab](mailto:comp.soft-sys.matlab@mathworks.com)
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.

Simulink® Design Optimization™ User's Guide

© COPYRIGHT 1993–2010 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 and Simulink are registered trademarks of The MathWorks, Inc. See www.mathworks.com/trademarks for a list of additional trademarks. Other product or brand names may be 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

March 2009	Online only	New for Version 1 (Release 2009a)
September 2009	Online only	Revised for Version 1.1 (Release 2009b)
March 2010	Online only	Revised for Version 1.1.1 (Release 2010a)

Data Analysis and Processing

1

Configuring a Model for Importing Data	1-2
Creating an Estimation Project	1-3
Importing Data into the GUI	1-5
Importing Time-Domain Data into the GUI	1-5
Importing Time-Series Data into the GUI	1-10
Importing Complex Data into the GUI	1-10
Plotting and Analyzing Data in the GUI	1-11
Why Plot the Data Before Parameter Estimation	1-11
How To Plot Data in the GUI	1-11
Preprocessing Data in the GUI	1-14
Ways to Preprocess Data Using the Data Preprocessing Tool	1-14
Opening the Data Preprocessing Tool	1-15
Handling Missing Data	1-16
Handling Outliers	1-18
Detrending Data	1-18
Filtering Data	1-18
Selecting Data	1-20
Adding Preprocessed Data Sets to an Estimation Project ..	1-28
Exporting Prepared Data to the MATLAB Workspace	1-31

Parameter Estimation

2

Overview of Parameter Estimation	2-2
---	------------

Configuring Parameter Estimation in the GUI	2-3
Creating an Estimation Task	2-3
Specifying Data for Parameter Estimation	2-4
Specifying Parameters to Estimate	2-6
Specifying Initial States	2-17
Selecting Views for Plotting	2-19
Specifying Estimation Options	2-23
Specifying Simulation Options	2-29
Specifying Display Options	2-35
Estimating Parameters in the GUI	2-36
Validating Parameters in the GUI	2-40
Basic Steps for Model Validation	2-40
Loading and Importing the Validation Data	2-41
Performing Validation	2-43
Comparing Residuals	2-47
Accelerating Model Simulations During Estimation ..	2-50
About Accelerating Model Simulations During Estimation	2-50
Limitations	2-50
Setting the Accelerator Mode for Parameter Estimation ..	2-50
Speeding Up Parameter Estimation Using Parallel Computing	2-52
When to Use Parallel Computing for Estimating Model Parameters	2-52
How Parallel Computing Speeds Up Parameter Estimation	2-53
Specifying Model Dependencies	2-56
Configuring Your System for Parallel Computing	2-56
How to Use Parallel Computing in the GUI	2-58
Troubleshooting	2-62
Estimating Initial States	2-65
How to Estimate Initial States in the GUI	2-65
Estimating Initial Conditions for Blocks with External Initial Conditions	2-66
Example — Estimating Initial States of a Mass-Spring-Damper System	2-67

Working with Estimation Projects	2-78
Structure of an Estimation Project	2-78
Managing Multiple Projects and Tasks	2-79
Adding, Deleting and Renaming an Estimation Project ...	2-80
Saving Control and Estimation Tools Manager Projects ..	2-81
Loading Control and Estimation Tools Manager Projects ..	2-82
Estimating Parameters at the Command Line	2-83
Workflow for Estimating Parameters at the Command Line	2-83
Example — Estimating Parameters and Initial States at the Command Line	2-84
Objects for Parameter Estimation	2-95
How to Use Parallel Computing at the Command Line ...	2-116

Parameter Optimization

3

Overview of Optimizing Model Parameters	3-2
Optimizing Parameters Using the GUI	3-3
Constraining Model Signals	3-3
Specifying Design Requirements	3-5
Specifying Parameters to Optimize	3-19
Specifying Optimization Options	3-27
Specifying the Simulation Options	3-32
Plotting Responses in the Signal Constraint Window	3-36
Running the Optimization	3-39
Optimizing Parameters for Model Robustness	3-42
What Is Model Robustness?	3-42
Sampling Methods for Computing Uncertain Parameter Values	3-43
How to Optimize Parameters for Model Robustness Using the GUI	3-46
Commands for Optimizing Parameters for Model Robustness	3-49
Example — Optimizing Parameters for Model Robustness Using the GUI	3-49

Accelerating Model Simulations During	
Optimization	3-58
About Accelerating Optimization	3-58
Limitations	3-58
Setting Accelerator Mode for Response Optimization	3-58
Speeding Up Response Optimization Using Parallel	
Computing	3-60
When to Use Parallel Computing for Response	
Optimization	3-60
How Parallel Computing Speeds Up Optimization	3-61
Configuring Your System for Parallel Computing	3-64
Specifying Model Dependencies	3-65
How to Use Parallel Computing in the GUI	3-66
How to Use Parallel Computing at the Command Line ...	3-70
Refining and Troubleshooting Optimization Results ..	3-72
Troubleshooting Optimization Results	3-72
Saving and Loading Response Optimization	
Projects	3-82
Saving Response Optimization Projects	3-82
Saving Additional Settings	3-85
Reloading Response Optimization Projects	3-86
Optimizing Parameters at the Command Line	3-87
Workflow for Optimizing Parameters at the Command	
Line	3-87
Configuring a Simulink Model for Optimizing	
Parameters	3-88
Creating or Extracting a Response Optimization Project ..	3-88
Specifying Design Requirements	3-89
Specifying Parameter Settings	3-93
Configuring Optimization and Simulation Settings	3-94
Running the Optimization	3-95

Optimization-Based Linear Control Design

4

Overview of Optimization-Based Compensator Design	4-2
Supported Time- and Frequency-Domain Requirements	4-4
Root Locus Diagrams	4-4
Open-Loop and Prefilter Bode Diagrams	4-6
Open-Loop Nichols Plots	4-6
Step/Impulse Response Plots	4-7
Designing Optimization-Based Controllers for LTI Systems	4-8
How to Design Optimization-Based Controllers for LTI Systems	4-8
Example — Frequency-Domain Optimization for LTI System	4-9
Designing Linear Controllers for Simulink Models ...	4-30

Lookup Tables

5

What Are Lookup Tables?	5-2
Static Lookup Tables	5-2
Adaptive Lookup Tables	5-3
Estimating Values of Lookup Tables	5-5
How to Estimate Values of a Lookup Table	5-5
Example — Estimating Lookup Table Values from Data ..	5-6
Example — Estimating Constrained Values of a Lookup Table	5-20
Capturing Time-Varying System Behavior Using Adaptive Lookup Tables	5-37

Building Models Using Adaptive Lookup Table Blocks . . .	5-37
Configuring Adaptive Lookup Table Blocks	5-41
Example — Modeling an Engine Using n-D Adaptive Lookup Table	5-44
Using Adaptive Lookup Tables in Real-Time Environment	5-60

Function Reference

6

Parameter Estimation	6-2
Parameter Optimization	6-3
Response Optimization Projects	6-3
Design Requirements	6-3
Parameters	6-4
Model Dependencies	6-4
Model Robustness	6-4
Optimization Options	6-4
Simulation Options	6-4

Functions — Alphabetical List

7

Block Reference

8

Examples

A

Parameter Estimation	A-2
Parameter Optimization	A-2
Optimization-Based Linear Control Design	A-2
Lookup Tables	A-2

Index

Data Analysis and Processing

- “Configuring a Model for Importing Data” on page 1-2
- “Creating an Estimation Project” on page 1-3
- “Importing Data into the GUI” on page 1-5
- “Plotting and Analyzing Data in the GUI” on page 1-11
- “Preprocessing Data in the GUI” on page 1-14

Configuring a Model for Importing Data

Before you can analyze and preprocess the estimation data, you must assign the data to the model's channels. In order to assign the data, the Simulink® model must contain one of the following elements:

- Top-level Inport block

Note You do not need an Inport block if your model already contains a fixed input block, such as a Step block.

- Top-level Outport block
- Logged signal. The logged signal can be a top-level signal in the model or a signal in the model subsystem.

For more information about the blocks and logged signals, see the Inport and Outport block reference pages and “Logging Signals” in the Simulink documentation.

In the Control and Estimation Tools Manager GUI, the rows in the **Input Data** tab correspond to the model's top-level Inport blocks. Similarly, the rows in the **Output Data** tab correspond to either the top-level Outport blocks or logged signals in the model.

Adding an Inport or Outport block or marking a signal for logging creates a new row in the corresponding **Input Data** or **Output Data** tab. You can use the new row to import estimation data for the corresponding signal. To view the new row, click **Update Task** in the **Estimation Task** node of the Control and Estimation Tools Manager GUI.

Creating an Estimation Project

Before you begin data import, you must create and set up an estimation project by configuring the appropriate parameters, solvers, and cost functions. Simulink® Design Optimization™ software provides a Graphical User Interface (GUI) that makes setting up the estimation project quick and easy.

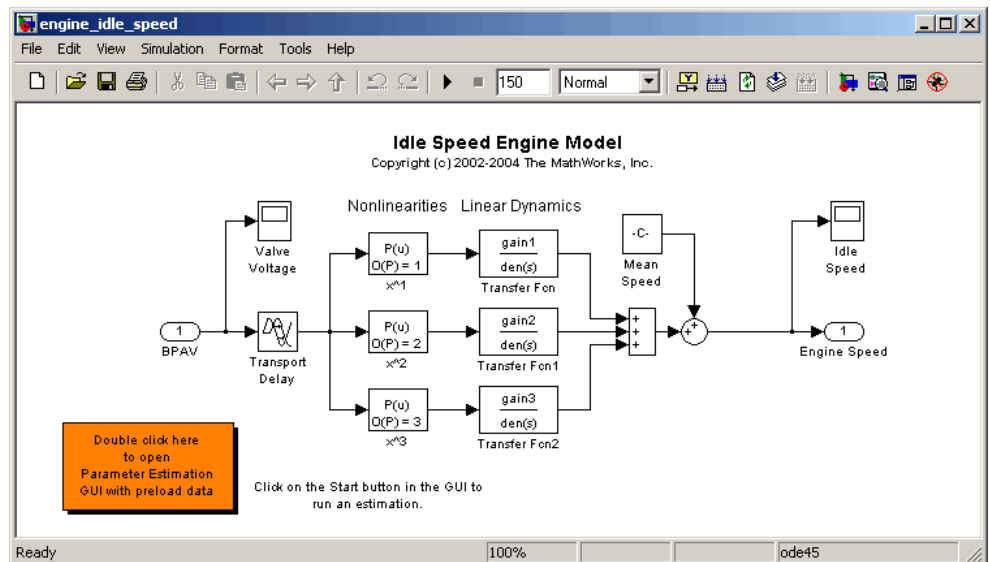
To create an estimation project:

- 1 Open the nonlinear idle speed model of an automotive engine by typing :

```
engine_idle_speed
```

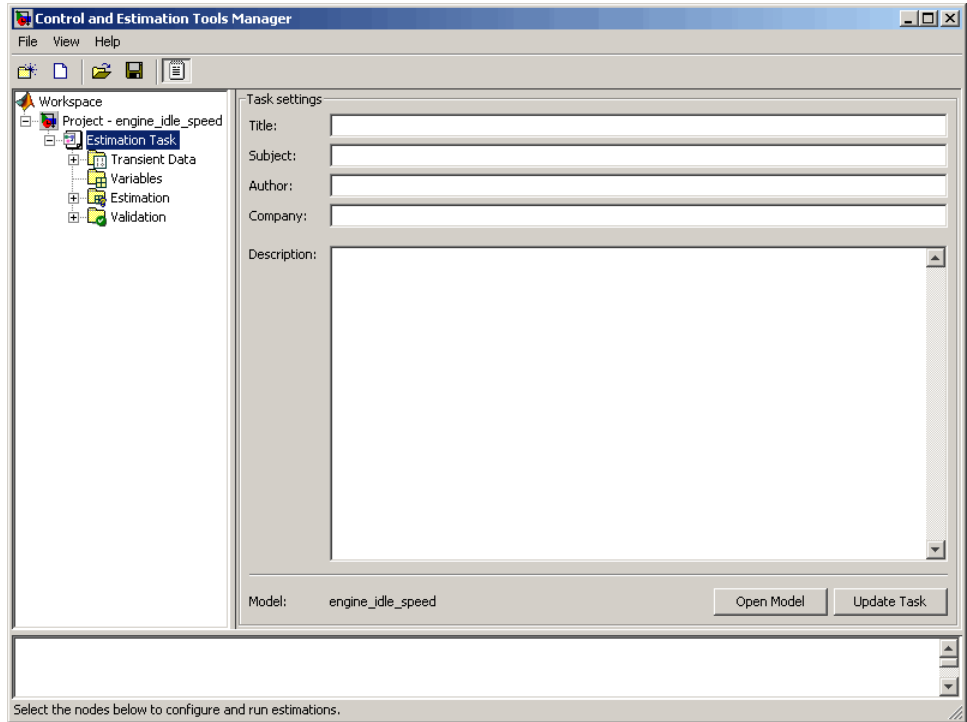
at the MATLAB® prompt.

The model appears as shown next.



The model contains the Inport block BPAV and Outport block Engine Speed for importing input and output data, respectively. To learn more, see “Configuring a Model for Importing Data” on page 1-2.

- 2 Open the Control and Estimation Tools Manager GUI by selecting **Tools > Parameter Estimation** in the Simulink model window.



Control and Estimation Tools Manager GUI

The project tree displays the project name **Project - engine_idle_speed**. Estimation tasks are organized inside the **Estimation Task** node.

Note The Simulink model must remain open to perform parameter estimation tasks.

Importing Data into the GUI

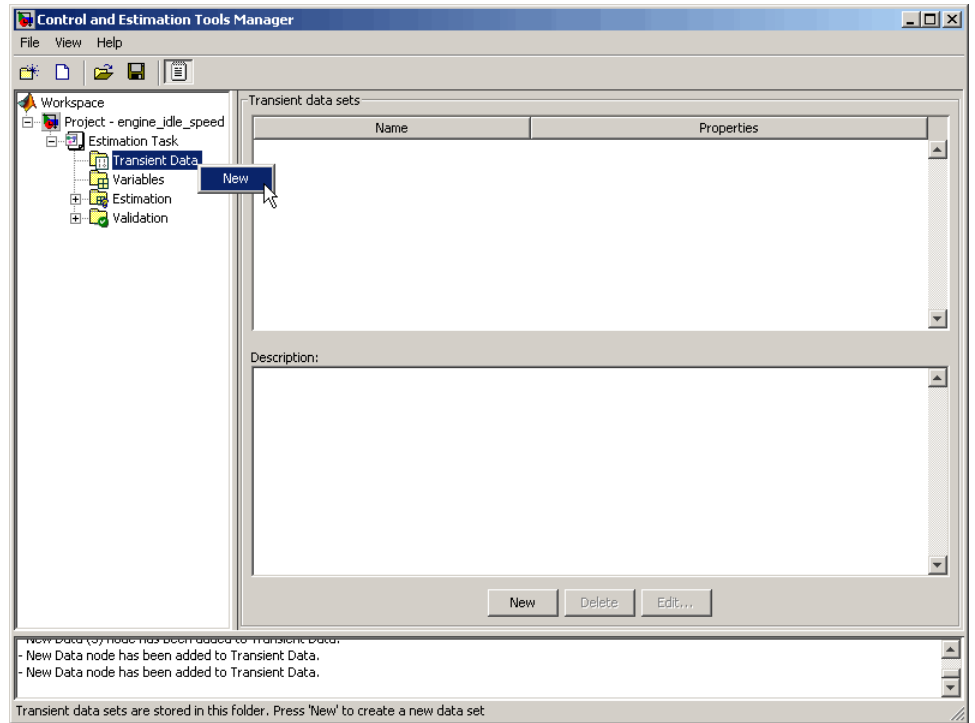
In this section...
“Importing Time-Domain Data into the GUI” on page 1-5
“Importing Time-Series Data into the GUI” on page 1-10
“Importing Complex Data into the GUI” on page 1-10

Importing Time-Domain Data into the GUI

After you create an estimation project, as described in “Creating an Estimation Project” on page 1-3, you can import the estimation data into the GUI. To learn more about the types of data for parameter estimation, see “Types of Data for Parameter Estimation” in the *Simulink Design Optimization Getting Started Guide*.

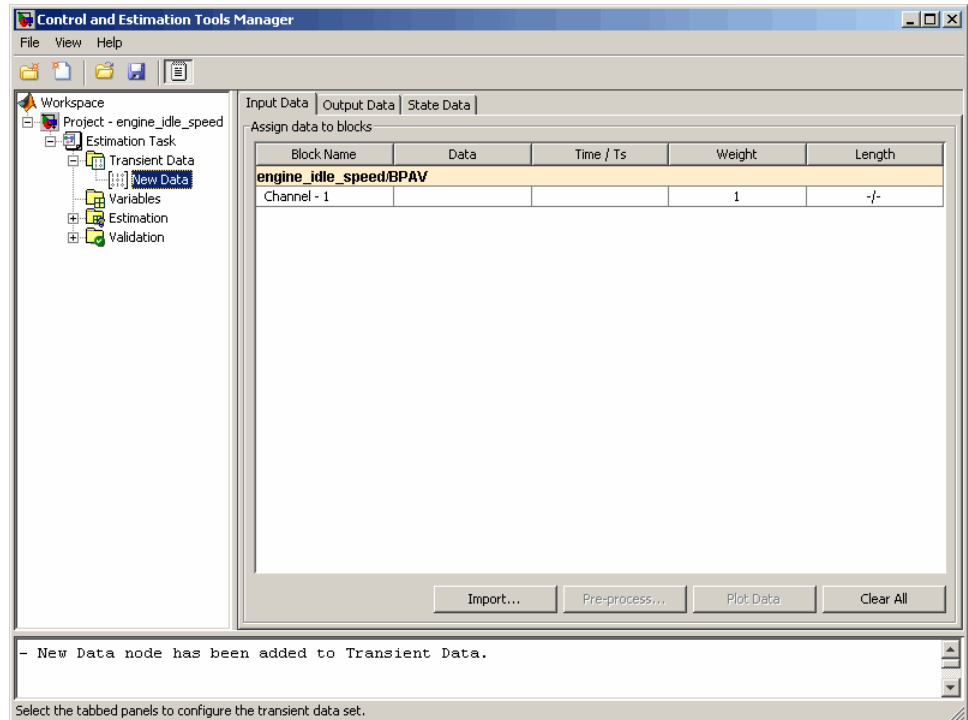
To import *transient* (measured) data for your dynamic system:

- 1 In the Control and Estimation Tools Manager, select **Transient Data** under the **Estimation Task** node of the **Workspace** tree.
- 2 Right-click **Transient Data** and select **New** to create a **New Data** node. Alternatively, you can use the **New** button to create this node.



3 Select the **New Data** node under the **Transient Data** node.

The Control and Estimation Tools Manager GUI now resembles the next figure.



Import Data into the Control and Estimation Tools Manager

The table rows in the **Input Data** tab corresponds to the Inport block BPAV in the `engine_idle_speed` model. Similarly, the rows in the **Output Data** tab corresponds to the Outport block Engine Speed.

Note The Simulink model must contain an Inport or Outport block or logged signals to enable importing data. For more information, see “Configuring a Model for Importing Data” on page 1-2.

The idle-speed model of an automotive engine contains the measured data stored in the `iodata` array. The array contains two columns: the first for input data, and the second for output data. You must import both the input and the output data, as described in the following sections:

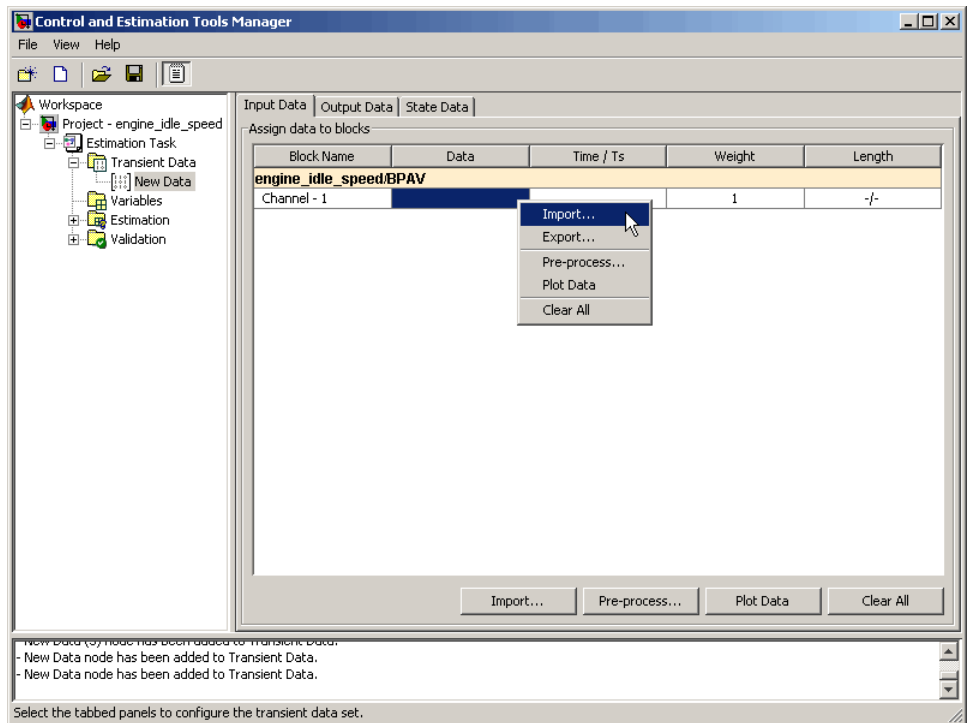
- “Importing Input Data and Time Vector” on page 1-8

- “Importing Output Data and Time Vector” on page 1-9

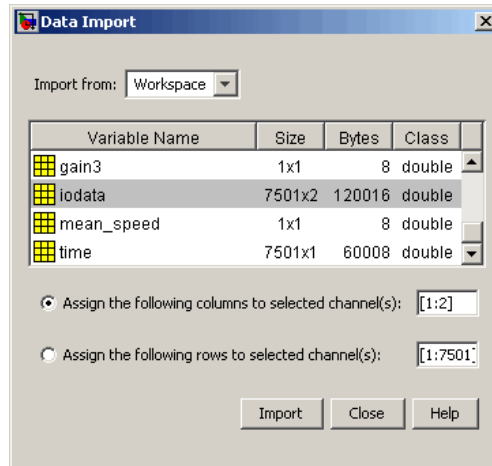
Importing Input Data and Time Vector

To import the input data for the port BPAV:

- 1 In the **New Data** node, click the **Input Data** tab.
- 2 Right-click the **Data** cell and select **Import** to open the Data Import dialog box. Alternatively, you can use the **Import** button to open this dialog box.



- 3 In the Data Import dialog box, select **iodata** from the list of variables.



- 4** Enter 1 in the **Assign the following columns to selected channel(s)** field, and then click **Import**.
- 5** In the **Input Data** tab, select the **Time/Ts** cell.
- 6** Select `time` in the Data Import dialog box.
- 7** Click **Import** to import the time vector for the input data.
- 8** Click **Close** to close the Data Import dialog box.

Importing Output Data and Time Vector

To import the output data for the port Engine Speed:

- 1** In the **New Data** node, select the **Output Data** tab.
- 2** Right-click the **Data** cell and select **Import** to open the Data Import dialog box.
- 3** In the Data Import dialog box, select `iodata` from the list of variables.
- 4** Enter 2 in the **Assign the following columns to selected channel(s)** field to use the second column of `iodata`, and then click **Import**.
- 5** In the **Output Data** tab, select the **Time/Ts** cell.

- 6** Select **time** in the Data Import dialog box.
- 7** Click **Import** to import the time vector for the output data.
- 8** Click **Close** to close the Data Import dialog box.

Importing Time-Series Data into the GUI

Time-series data is stored in time-series objects. For more information, see “Time Series Objects” in the MATLAB documentation.

When you import time-series data for parameter estimation, specify the data and time vector as *t.data* and *t.time* in the **Data** and **Time/Ts** columns of the **New Data** node, respectively. For more information on how to import data into the GUI, see “Importing Time-Domain Data into the GUI” on page 1-5.

Importing Complex Data into the GUI

Complex-valued data is data whose value is a complex number. For example, a signal with the value $1+2j$ is complex. You can use complex data to estimate parameters of electrical systems, such as the magnitude and phase.

Note You must sample the real and imaginary parts of the data as a function of the same time vector.

To use complex data for parameter estimation:

- 1** Split the data into two data sets that contain the real and imaginary parts. To split the data, use the MATLAB functions `real`, and `imag`.
- 2** Import both data sets into the GUI, as described in “Importing Time-Domain Data into the GUI” on page 1-5.
- 3** Specify both the data sets together as estimation data, as described in “Specifying Data for Parameter Estimation” on page 2-4.
- 4** Estimate the parameters, as described in “Estimating Parameters in the GUI” on page 2-36.

Plotting and Analyzing Data in the GUI

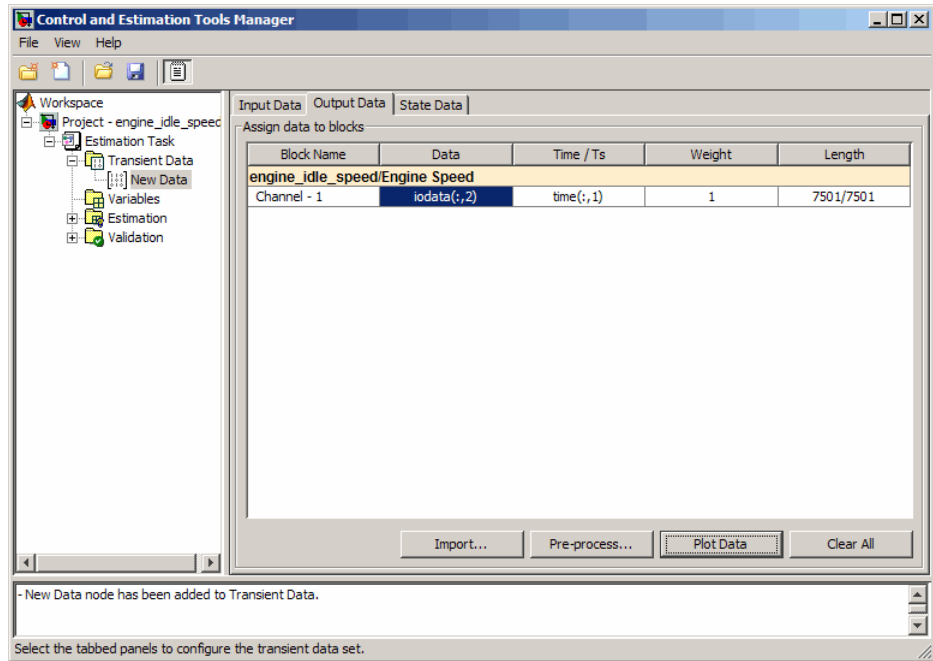
In this section...
“Why Plot the Data Before Parameter Estimation” on page 1-11
“How To Plot Data in the GUI” on page 1-11

Why Plot the Data Before Parameter Estimation

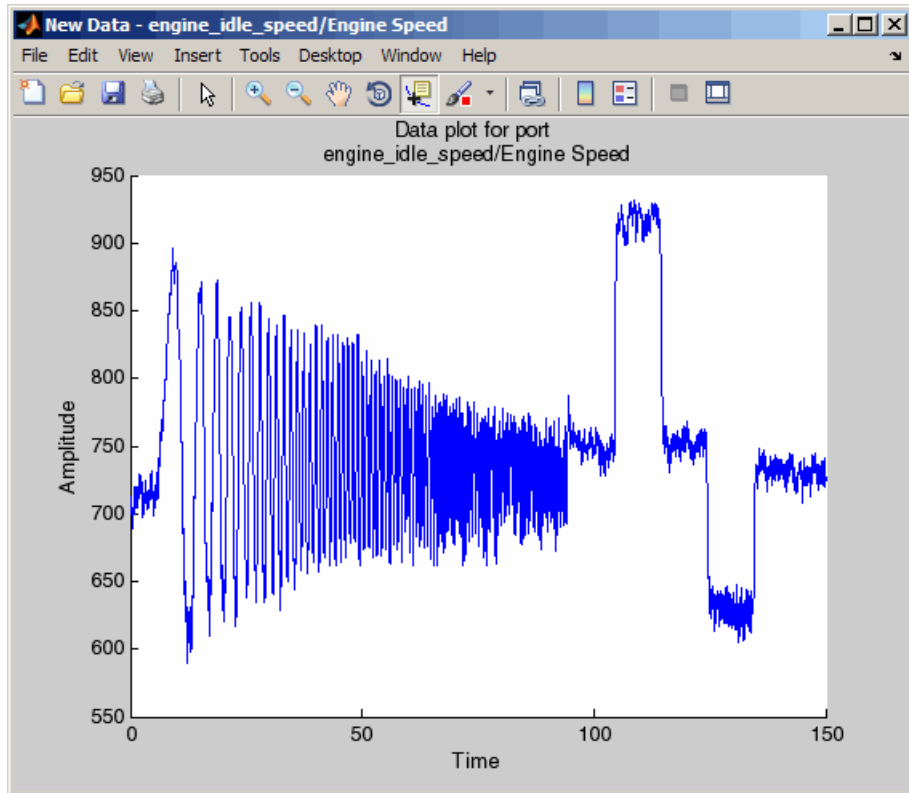
After you import the estimation data, as described in “Importing Data into the GUI” on page 1-5, it is useful to remove outliers, smooth, detrend, or otherwise treat the data to make it more tractable for analysis and estimation purposes. To view and analyze the data characteristics, you must plot the data on a time plot.

How To Plot Data in the GUI

To plot a data set, select the **Data** cell that you want to plot in the **Transient Data** node of the Control and Estimation Tools Manager GUI, and click **Plot Data**.



The data is plotted on a time plot, as shown in the next figure.



Using the time plot, you can examine the data characteristics such as noise, outliers and portions of the data to use for estimating parameters. After you analyze the data, you the preprocess the data as described in “Preprocessing Data in the GUI” on page 1-14.

Preprocessing Data in the GUI

In this section...
“Ways to Preprocess Data Using the Data Preprocessing Tool” on page 1-14
“Opening the Data Preprocessing Tool” on page 1-15
“Handling Missing Data” on page 1-16
“Handling Outliers” on page 1-18
“Detrending Data” on page 1-18
“Filtering Data” on page 1-18
“Selecting Data” on page 1-20
“Adding Preprocessed Data Sets to an Estimation Project” on page 1-28
“Exporting Prepared Data to the MATLAB Workspace” on page 1-31

Ways to Preprocess Data Using the Data Preprocessing Tool

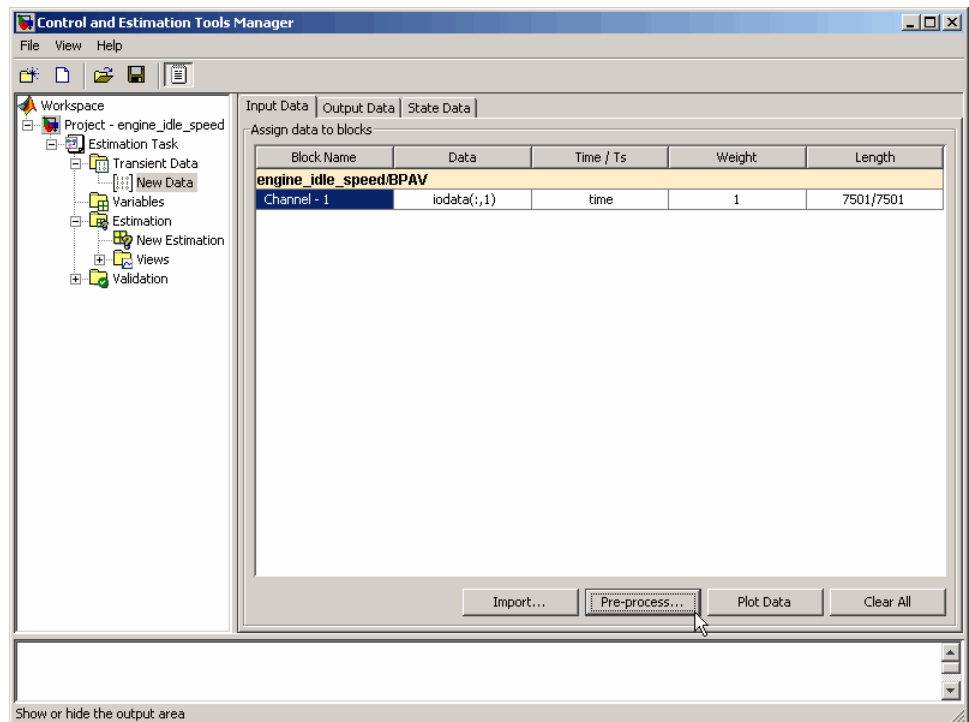
After you import the estimation data, as described in “Importing Data into the GUI” on page 1-5, you can perform the following preprocessing operations using the Data Preprocessing Tool in Simulink Design Optimization software:

- Exclusion — Exclude a portion of the data from the estimation process. You can exclude data by:
 - Selecting it with your mouse.
 - Graphically by selecting regions on a plot.
 - Using rules, such as upper or lower bounds.
- Handle missing data — Remove missing data, or compute missing data using interpolation.
- Handle outliers — Remove outliers.
- Detrend — Remove mean values or a straight line trend.
- Filter — Smooth data using a first-order filter, an arbitrary transfer function, or an ideal filter.

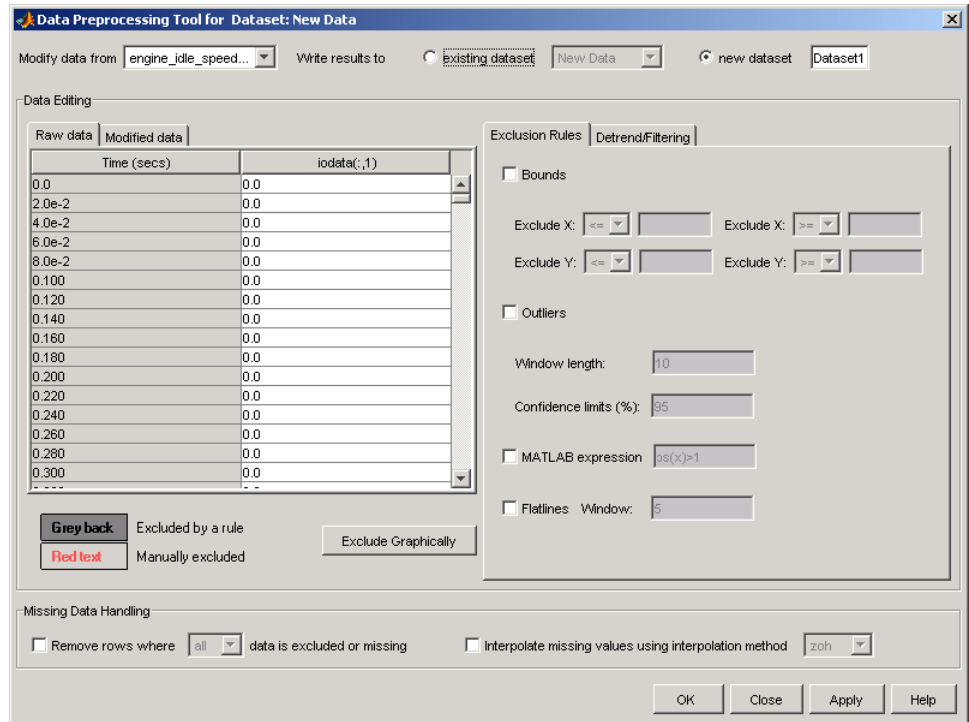
Opening the Data Preprocessing Tool

To open the Data Preprocessing Tool:

- 1 In the Control and Estimation Tools Manager GUI, select the **Transient Data** node under the **Estimation Task** node, and then choose the data you want to preprocess either in the **Input Data**, or **Output Data** tab. This enables the **Pre-process** button.



- 2 Click **Pre-process** to open the Data Preprocessing Tool.



Tip When you have multiple data sets, select the data set that you want to preprocess from the **Modify data from** drop-down list in the Data Preprocessing Tool.

In this section, the sample data set imported for preprocessing is the same as used in the `engine_idle_speed` Simulink model. For an overview of creating estimation projects and importing data sets, see “Configuring a Model for Importing Data” on page 1-2, and “Creating an Estimation Project” on page 1-3.

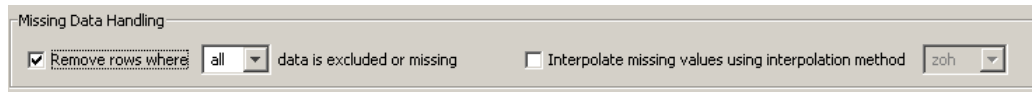
Handling Missing Data

- “Removing Missing Data” on page 1-17

- “Interpolating Missing Data” on page 1-17

Removing Missing Data

Rows of missing or excluded data are represented by NaNs. To remove the rows containing missing or excluded data, select the **Remove rows where** check box in the **Missing Data Handling** area of the Data Preprocessing Tool GUI.



When the data set contains multiple columns of data, select **all** to remove rows in which all the data is excluded. Select **any** to remove any excluded cell. In the case of one-column data, **any** and **all** are equivalent.

Tip You can view the modified data in the **Modified data** tab of the Data Preprocessing Tool GUI.

Interpolating Missing Data

The interpolation operation computes the missing data values using known data values. When you select the **Interpolate missing values using interpolation method** check box in the **Missing Data Handling** area of the Data Preprocessing Tool GUI, the software interpolates the missing data values.



You can compute the missing data values using one of the following interpolation methods:

- Zero-order hold (zoh) — Fills the missing data sample with the data value immediately preceding it.
- Linear interpolation (Linear) — Fills the missing data sample with the average of the data values immediately preceding and following it.

By default, the interpolation method is set to `zoh`. You can select the Linear interpolation method from the **Interpolate missing values using interpolation method** drop-down list.

Tip You can view the results of interpolation in the **Modified data** tab of the Data Preprocessing Tool GUI.

Handling Outliers

Outliers are data values that deviate from the mean by more than three standard deviations. When estimating parameters from data containing outliers, the results may not be accurate.

To remove outliers, select the **Outliers** check box to activate outlier exclusion. You can set the **Window length** to any positive integer, and use confidence limits from 0 to 100%. The window length specifies the number of data points used when calculating outliers.

Removing outliers replaces the data samples containing outliers with NaNs, which you can interpolate in a subsequent operation. To learn more, see “Interpolating Missing Data” on page 1-17.

Detrending Data

To detrend, select the **Detrending** check box. You can choose constant or straight line detrending. Constant detrending removes the mean of the data to create zero-mean data. Straight line detrending finds linear trends (in the least-squares sense) and then removes them.

Filtering Data

- “Types of Filters” on page 1-18
- “How to Filter Data” on page 1-19

Types of Filters

You have these choices for filtering your data:

- **First order** — A filter of the type $\frac{1}{\tau s + 1}$ where τ is the time constant that you specify in the associated field.

- **Transfer function** — A filter of the type

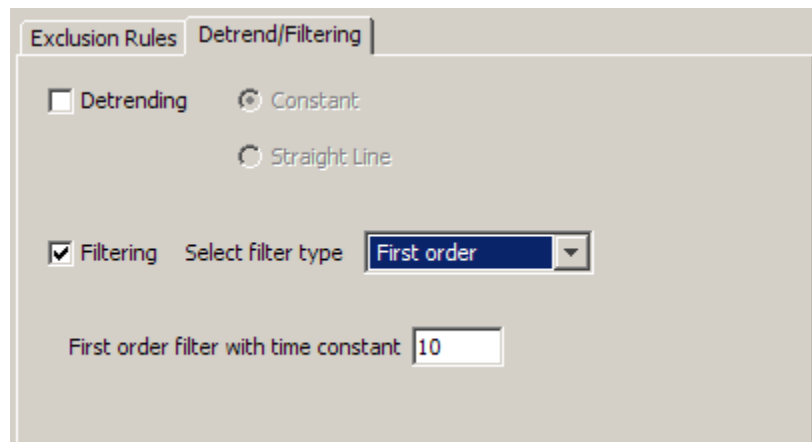
$$\frac{a_n s^n + a_{n-1} s^{n-1} + \dots + a_0}{b_m s^m + b_{m-1} s^{m-1} + \dots + b_0}$$

where you specify the coefficients as vectors in the associated **A coefficients** and **B coefficients** fields.

- **Ideal** — An idealized (noncausal) filter, either stop or pass band. Specify either filter as a two-element vector in the **Range (Hz)** field. These filters are ideal in the sense that there is no finite rolloff or ripple; the ends of the ranges are perfectly horizontal in the frequency domain.

How to Filter Data

To filter the data to remove noise, select the **Detrend/Filtering** tab in the Data Preprocessing Tool GUI. Select the **Filtering** check box, and choose the type of filter from the **Select filter type** drop-down list.



Selecting Data

- “Techniques for Excluding Data in the Data Preprocessing Tool” on page 1-20
- “Graphically Selecting Data” on page 1-20
- “Using Rules to Select Data Samples” on page 1-23
- “Using the Data Table to Select Data Samples” on page 1-25

Techniques for Excluding Data in the Data Preprocessing Tool

You can use the Data Preprocessing Tool to select a portion of the data to be excluded from the estimation process. You can choose one of the following techniques:

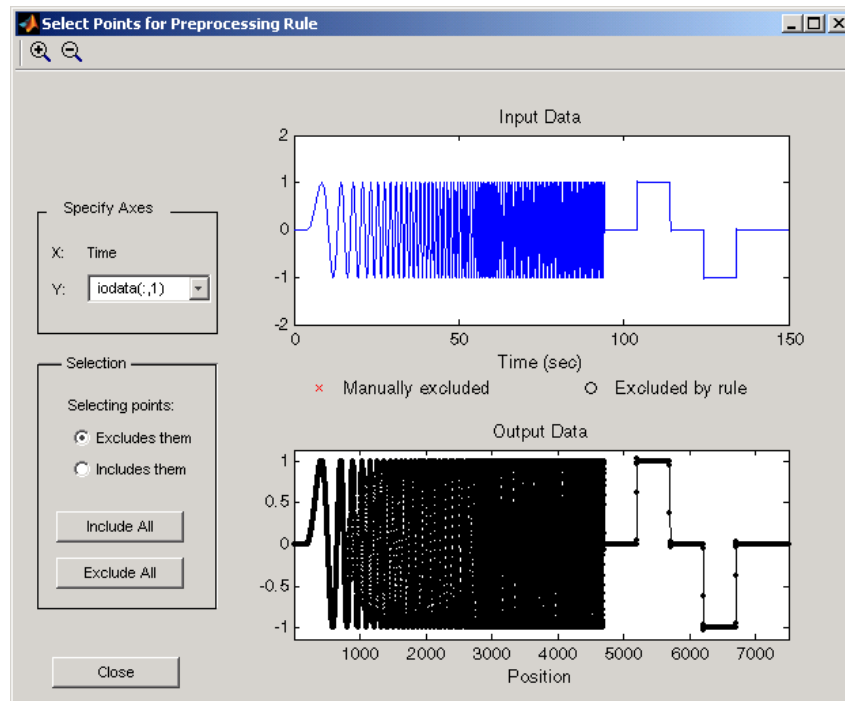
- Selecting data from the Data Editing Table.
- Selecting data from a plot of the data.
- Specifying a rule.

You accomplish the first two manually, and for the last you specify a rule. When you exclude data using manual selection, the excluded data is shown as red. When you exclude data using a rule, the background color of the cell becomes gray. When a portion of the data is excluded both manually and by a rule, the data is red, and the background is gray.

Note Changes in data are visible everywhere. When you use the **Data Editing** table, you can view the results in the data plot.

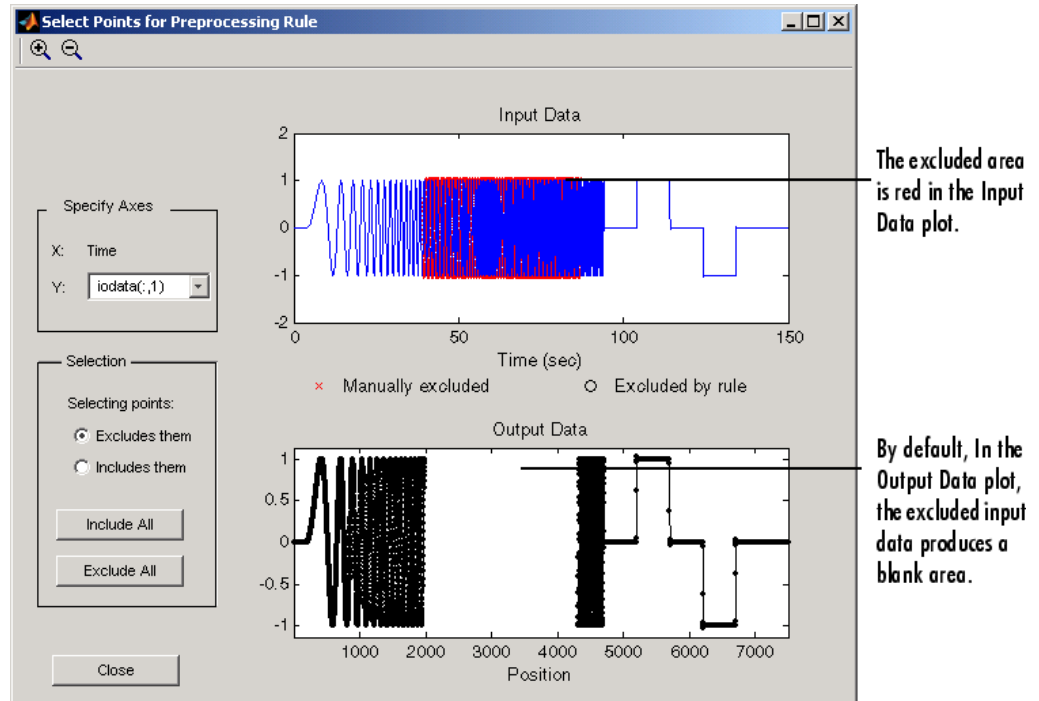
Graphically Selecting Data

You can exclude data graphically. Click **Exclude Graphically** to open the Select Points for Preprocessing Rule window.



The way you exclude data is similar to the way you select a region for zooming: place your cursor in the **Input Data** plot and drag the mouse to draw a region of exclusion.

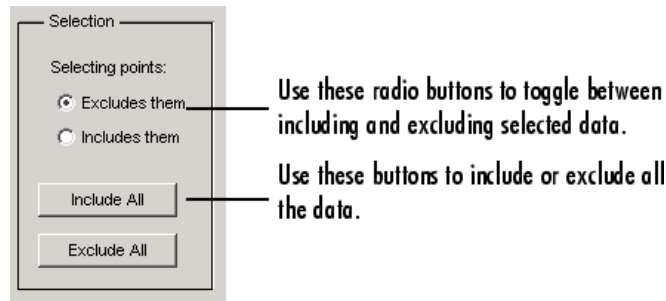
This figure shows an example of resulting data exclusion in the input data.



In the **Output Data** plot, the excluded input data produces a blank area by default. This corresponds to the NaNs that now represent excluded data. If you choose to interpolate or remove the excluded data, the output data shows the interpolated points.

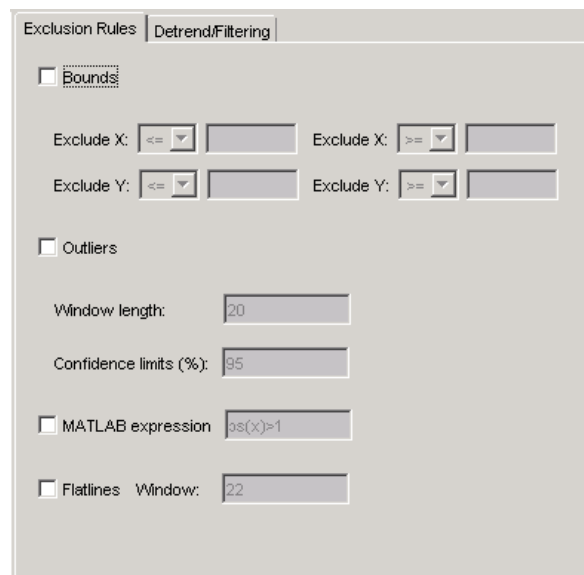
When you make changes in the Select Points for Preprocessing Rule window, they immediately appear in the **Data Editing** pane, and vice versa.

Selection Pane. By default, any box that you draw with your mouse selects data for exclusion, but you can toggle between exclusion and inclusion using the **Selection** pane on the left side of the Select Points for Preprocessing Rule window.



Using Rules to Select Data Samples

A more precise way to exclude data is to use mathematical rules. The **Exclusion Rules** pane in the Data Preprocessing Tool allows you to enter customized rules for excluding data.



These are the rules you can use to exclude data:

- “Upper and Lower Bounds” on page 1-24
- “MATLAB Expressions” on page 1-24

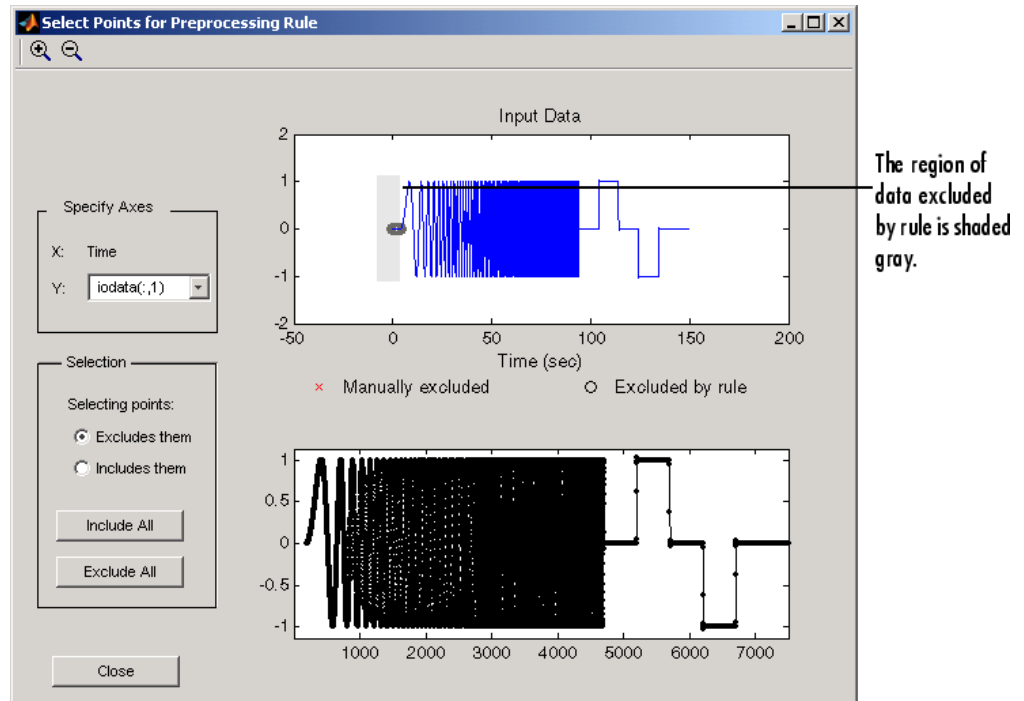
- “Flatlines” on page 1-24

Upper and Lower Bounds. Select the **Bounds** check box to activate upper and lower bound exclusion. Enter numbers in the **Exclude X** and **Exclude Y** fields for upper and lower bound exclusion. By default, the exclusion rule is to include the boundary values, but you can use the menu to exclude the boundaries as well.

MATLAB Expressions. Use the **MATLAB expression** field to enter any mathematical expression using MATLAB code. Use `x` as the variable name in your expression for the data being tested.

Flatlines. If you have areas of your data set where the data is constant, providing no new information, then you can choose to exclude those data points as flatlines. The **Window length** field sets the minimum number of constant data points required to define the area as a flatline.

Example of Rule Exclusion. This figure shows data with a region of the x -axis excluded.



Using the Data Table to Select Data Samples

The **Data Editing** table lists both the raw data set and the modified data that you create.

Data Editing

Raw data | Modified data

Time (secs)	iodata(:,1)
21.680	-0.472
21.700	-0.515
21.720	-0.557
21.740	-0.598
21.760	-0.638
21.780	-0.676
21.800	-0.712
21.820	-0.746
21.840	-0.779
21.860	-0.809
21.880	-0.838
21.900	-0.865
21.920	-0.889
21.940	-0.911
21.960	-0.931
21.980	-0.948

Use your mouse to select groups of cells for exclusion. Selected cells become blue. Right-click and select **Exclude**. The background becomes white, but the numbers are now red.

Click this button to view the data graphically.

Grey back Excluded by a rule
Red text Manually excluded
Exclude Graphically

There are two tabs in the **Data Editing** pane: **Raw data** and **Modified data**. The **Raw Data** pane shows the working copy of the data. For example, if you exclude rows of data in the **Raw data** pane, the corresponding rows of numbers become red in this table. By default the **Modified data** pane represents the rows you removed by inserting NaNs.

Data Editing

Raw data Modified data

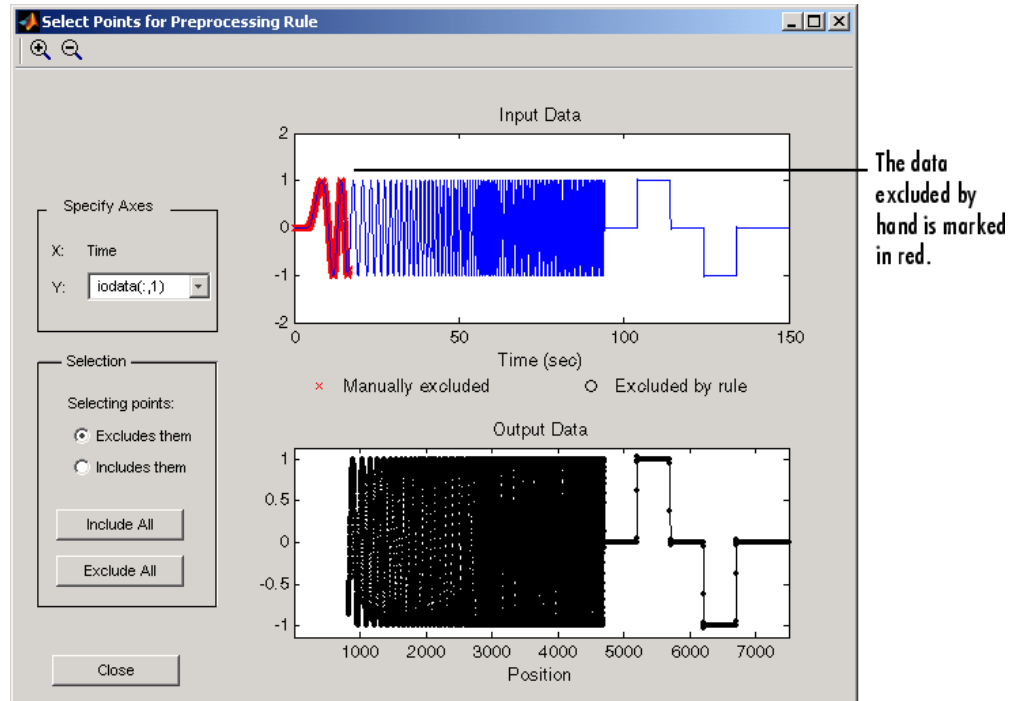
Time (secs)	iodata(:,1)*
8.760	0.979
8.780	0.976
8.800	0.972
8.820	0.969
8.840	NaN
8.860	NaN
8.880	NaN
8.900	NaN
8.920	NaN
8.940	NaN
8.960	NaN
8.980	NaN
9.0	NaN
9.20	NaN
9.40	NaN
9.60	NaN
...	...

Grey back Excluded by a rule
 Red text Manually excluded

By default, data that you excluded from the **Raw Data** table is represented by NaNs in the **Modified Data** table. If you choose to interpolate or remove missing data, the results of that action are shown in the **Modified Data** table.

In the **Modified data** pane, you can choose to remove the excluded data completely or interpolate it. See “Handling Missing Data” on page 1-16 for more information.

After you select data for exclusion, you can view it graphically by clicking **Exclude Graphically**.



As you make changes in the **Data Editing** pane, they immediately appear in the Select Points for Preprocessing Rule window, and vice versa.

Adding Preprocessed Data Sets to an Estimation Project

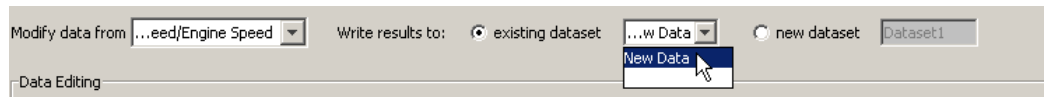
After you preprocess the data using the techniques described in “Ways to Preprocess Data Using the Data Preprocessing Tool” on page 1-14, you can add the data set to an estimation project either by overwriting an existing data set or creating a new data set.

- “Overwriting an Existing Data Set” on page 1-29
- “Creating a New Data Set” on page 1-30

Overwriting an Existing Data Set

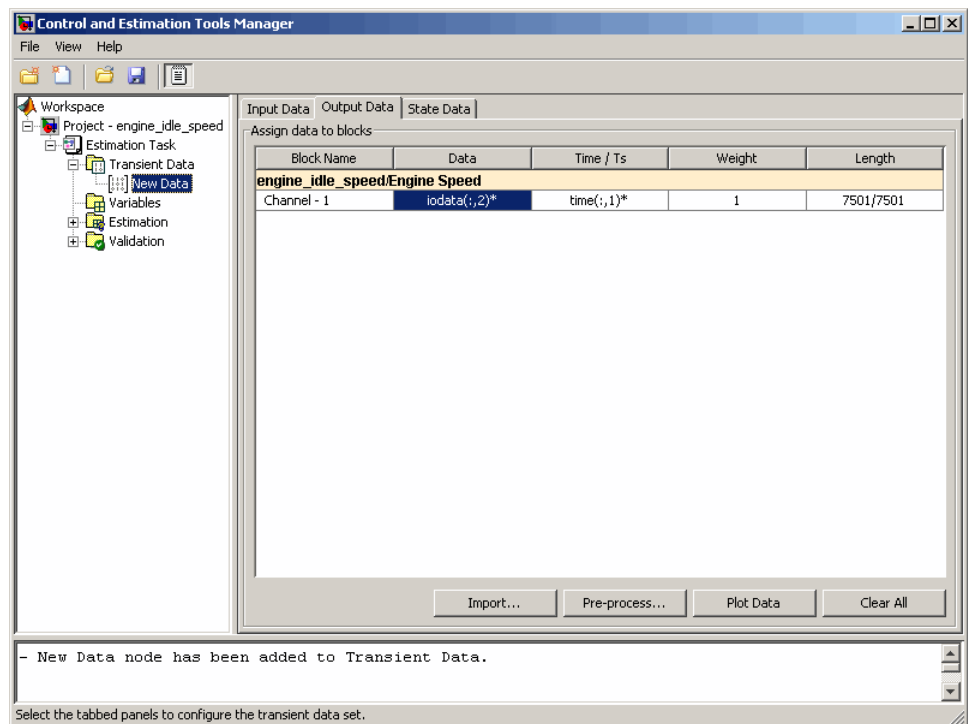
To overwrite an existing data set with the preprocessed data:

- 1 In the **Write results to** area of the Data Preprocessing Tool GUI, select the **existing dataset** option.
- 2 Choose the data set you want to overwrite from the drop-down list.



- 3 Click **Add**.

This action overwrites the selected data set with the modified data in the Control and Estimation Tools Manager GUI.



Tip You can export the preprocessed data to the MATLAB Workspace, as described in “Exporting Prepared Data to the MATLAB Workspace” on page 1-31.

Creating a New Data Set

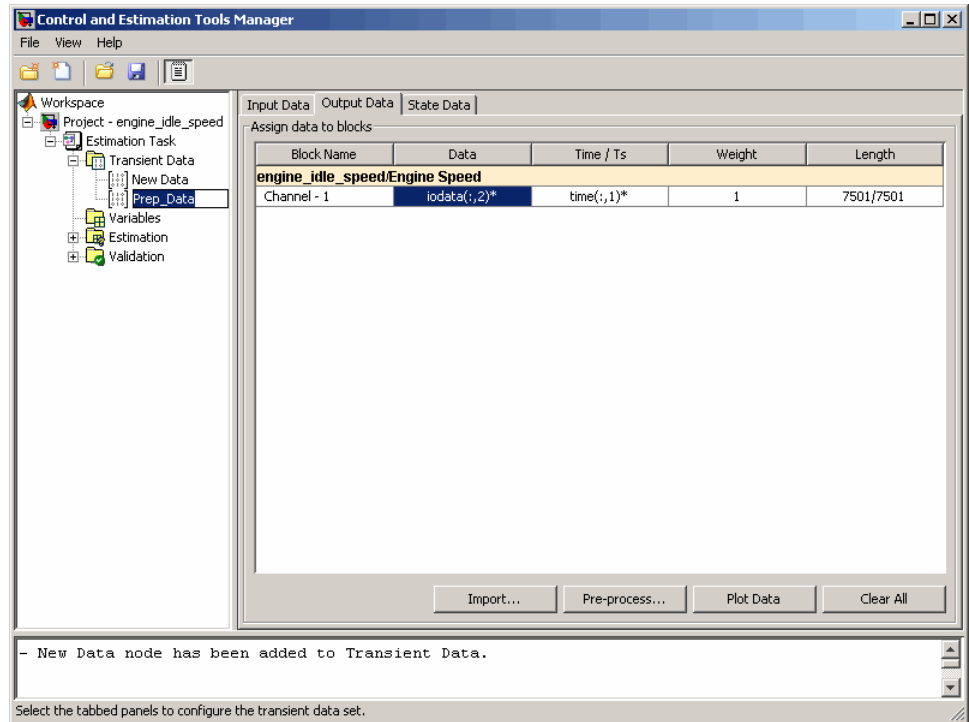
If you do not want to overwrite an existing data set with the preprocessed data, as described in “Overwriting an Existing Data Set” on page 1-29, you can create a new data set for the preprocessed data:

- 1 In the **Write results to** area of the Data Preprocessing Tool GUI, select the **new dataset** option.
- 2 Specify the name of the data set in the adjacent field.



- 3 Click **Add**.

This action adds a new data node in the Control and Estimation Tools Manager GUI containing the modified data.



Tip You can export the preprocessed data to the MATLAB Workspace, as described in “Exporting Prepared Data to the MATLAB Workspace” on page 1-31.

Exporting Prepared Data to the MATLAB Workspace

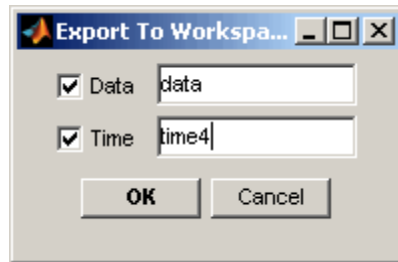
After you add the preprocessed data to an estimation project, as described in “Adding Preprocessed Data Sets to an Estimation Project” on page 1-28, you can export the data set to the MATLAB Workspace. You can use the data to further prepare it or estimate parameters using the data.

- 1 In the **Transient Data** node of the Control and Estimation Tools Manager GUI, select the node containing the prepared data set.

- 2** Right-click the table **Data** cell containing the data that you want to export, and select **Export**.

The Export to Workspace dialog box opens.

- 3** Specify the MATLAB variable names for the prepared data and the corresponding time vector in the **Data** and **Time** fields, respectively.



- 4** Click **OK**.

The resulting MATLAB variables `data` and `time4` appear in the MATLAB Workspace browser.

Parameter Estimation

- “Overview of Parameter Estimation” on page 2-2
- “Configuring Parameter Estimation in the GUI” on page 2-3
- “Estimating Parameters in the GUI” on page 2-36
- “Validating Parameters in the GUI” on page 2-40
- “Accelerating Model Simulations During Estimation” on page 2-50
- “Speeding Up Parameter Estimation Using Parallel Computing” on page 2-52
- “Estimating Initial States” on page 2-65
- “Working with Estimation Projects” on page 2-78
- “Estimating Parameters at the Command Line” on page 2-83

Overview of Parameter Estimation

When you estimate model parameters, Simulink Design Optimization software compares the measured data with data generated by a Simulink model. Using optimization techniques, the software estimates the parameter and (optionally) initial conditions of states to minimize a user-selected cost function. The cost function typically calculates a least-square error between the empirical and model data signals.

After you import and preprocess the estimation data, as described in “Importing Data into the GUI” on page 1-5 and “Preprocessing Data in the GUI” on page 1-14, follow these steps to estimate model parameters:

- 1** “Creating an Estimation Task” on page 2-3
- 2** “Specifying Data for Parameter Estimation” on page 2-4
- 3** “Specifying Parameters to Estimate” on page 2-6
- 4** “Specifying Initial States” on page 2-17
- 5** “Selecting Views for Plotting” on page 2-19
- 6** “Specifying Estimation Options” on page 2-23
- 7** Estimating Parameter
- 8** Validating Parameters

Note The Simulink model must remain open to perform parameter estimation tasks.

To learn how to estimate parameters at the command line, see “Estimating Parameters at the Command Line” on page 2-83.

Configuring Parameter Estimation in the GUI

In this section...

“Creating an Estimation Task” on page 2-3

“Specifying Data for Parameter Estimation” on page 2-4

“Specifying Parameters to Estimate” on page 2-6

“Specifying Initial States” on page 2-17

“Selecting Views for Plotting” on page 2-19

“Specifying Estimation Options” on page 2-23

“Specifying Simulation Options” on page 2-29

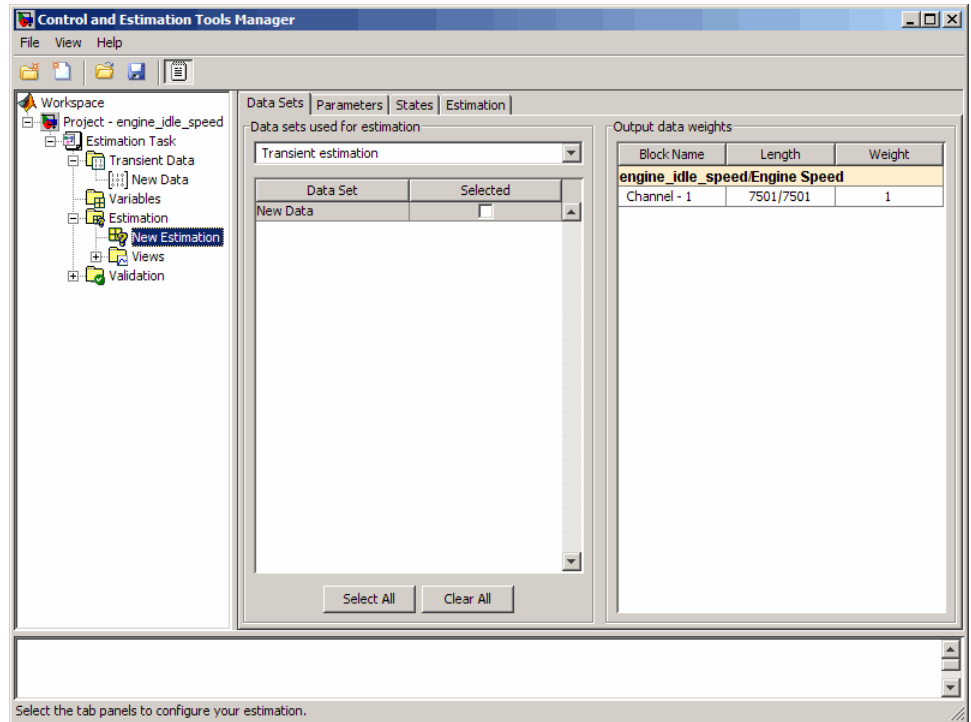
“Specifying Display Options” on page 2-35

Creating an Estimation Task

This section describes how to use the GUI to estimate parameters. After you import the transient data, as described in “Importing Data into the GUI” on page 1-5, you must create an estimation task and configure the estimation settings. To create a container that stores the estimation settings:

- 1** In the Control and Estimation Tools Manager, right-click the **Estimation** node in the **Workspace** tree and select **New**.
- 2** Select the **New Estimation** node.

The Control and Estimation Tools Manager now resembles the next figure.



Specifying Data for Parameter Estimation

- “Prerequisite for Specifying Data” on page 2-4
- “How to Specify Data in the GUI” on page 2-5

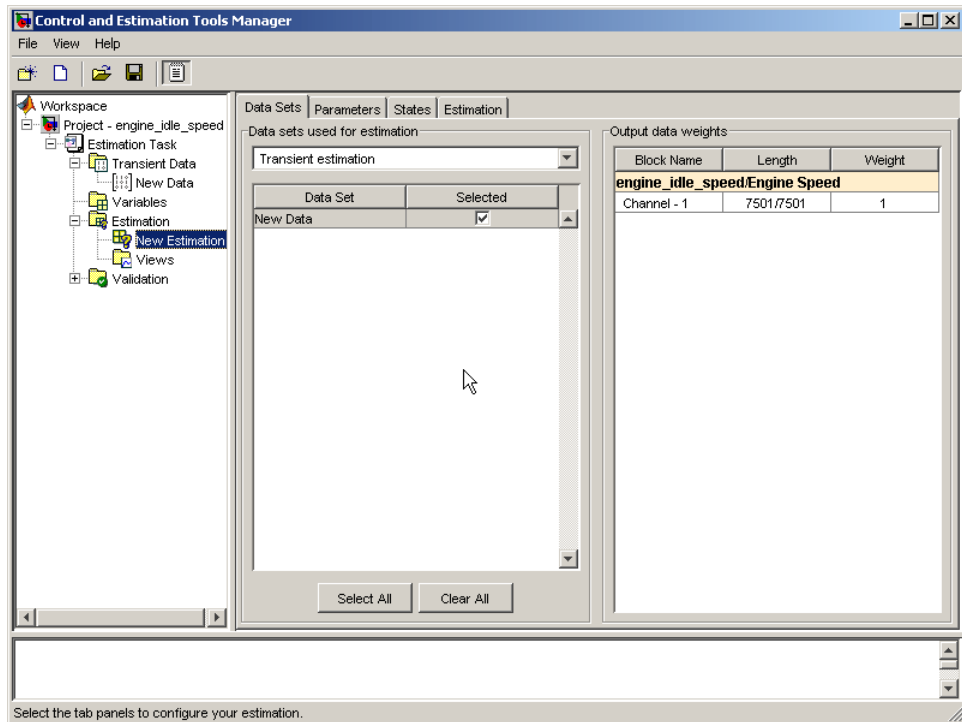
Prerequisite for Specifying Data

To specify a data set for estimation, you must have already imported the data in the GUI and created an Estimation Task, as described in “Creating an Estimation Task” on page 2-3. If your data contains noise or outliers, you must also preprocess the data, as described in “Preprocessing Data in the GUI” on page 1-14.

How to Specify Data in the GUI

After you select the **New Estimation** node, the **Data Sets** tab appears. Here you select the data set that you want to use in the estimation.

Select the **Selected** check box to the right of the **New Data** data set.



Note If you imported multiple data sets, you can select them for estimation by selecting the check box to the right of each desired data set. When using several data sets, you increase the estimation precision. However, you also increase the number of required simulations: for N parameters and M data sets, there are $M \cdot (2N + 1)$ simulations per iteration.

Then, specify the weight of each output from this model by setting the **Weight** column in the **Output data weights** table.

The relative weights are used to place more or less emphasis on specific output variables. The following are a few guidelines for specifying weights:

- Use less weight when an output is noisy.
- Use more weight when an output strongly affects parameters.
- Use more weight when it is more important to accurately match this model output to the data.

Specifying Parameters to Estimate

- “Choosing Which Parameters to Estimate First” on page 2-6
- “How to Specify Parameters for Estimation in the GUI” on page 2-6
- “Specifying Initial Guesses and Upper/Lower Bounds” on page 2-11
- “Specifying Parameter Dependency” on page 2-13
- “Example: Specifying Independent Parameters for Estimation” on page 2-14

Choosing Which Parameters to Estimate First

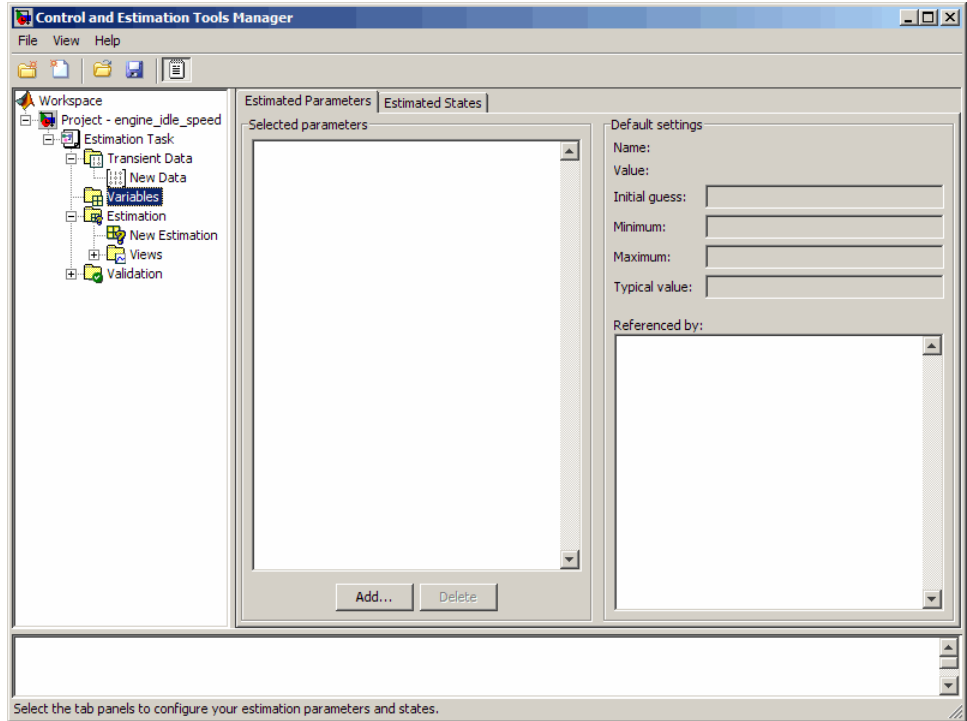
Simulink Design Optimization software lets you estimate scalar, vector and matrix parameters. Estimating model parameters is an iterative process. Often, it is more practical to estimate a small group of parameters and use the final estimated values as a starting point for further estimation of parameters that are trickier. When you have a large number of parameters to estimate, select the parameters that influence the output the most to be estimated first. Making these sorts of choices involves experience, intuition, and a solid understanding of the strengths and limitations of your Simulink model.

After you estimate a subset of parameters and validate the estimated parameters, select the remaining parameters for estimation.

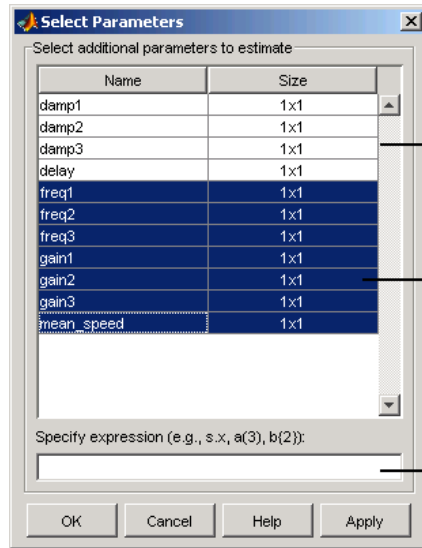
How to Specify Parameters for Estimation in the GUI

To select parameters for estimation:

- 1 In the Control and Estimation Tools Manager, select the **Variables** node in the **Workspace** tree to open the **Estimated Parameters** pane.



- 2** In the **Estimated Parameters** pane, click **Add** to open the Select Parameters dialog box.



By default, the Select Parameters dialog box looks at all variables in the model workspace and the MATLAB workspace that are used by the model.

List of parameters

Use your mouse to select data. To select adjacent parameters, hold down the Shift key while clicking the first and last parameter in the selection. To select nonadjacent parameters, hold down the Ctrl key while clicking each parameter.

Use the text field to get the parameters contained in either a Simulink parameter object, MATLAB array, structure, or cell array. Note that you cannot use mathematical expressions such as $x + 5$.

The dialog box lists all the variables in the model workspace and the MATLAB workspace that the model uses. You can use the mouse to select the parameters to estimate.

You can also enter parameters, separated by commas, in the **Specify expression** field of the Select Parameters dialog box. The parameters can be stored in one of the following:

- Simulink software parameter object

Example: For a Simulink parameter object *k*, type *k.value*.

- Structure

Example: For a structure *S*, type *S.fieldname* (where *fieldname* represents the name of the field that contains the parameter).

- Cell array

Example: Type *C{1}* to select the first element of the *C* cell array.

- MATLAB array

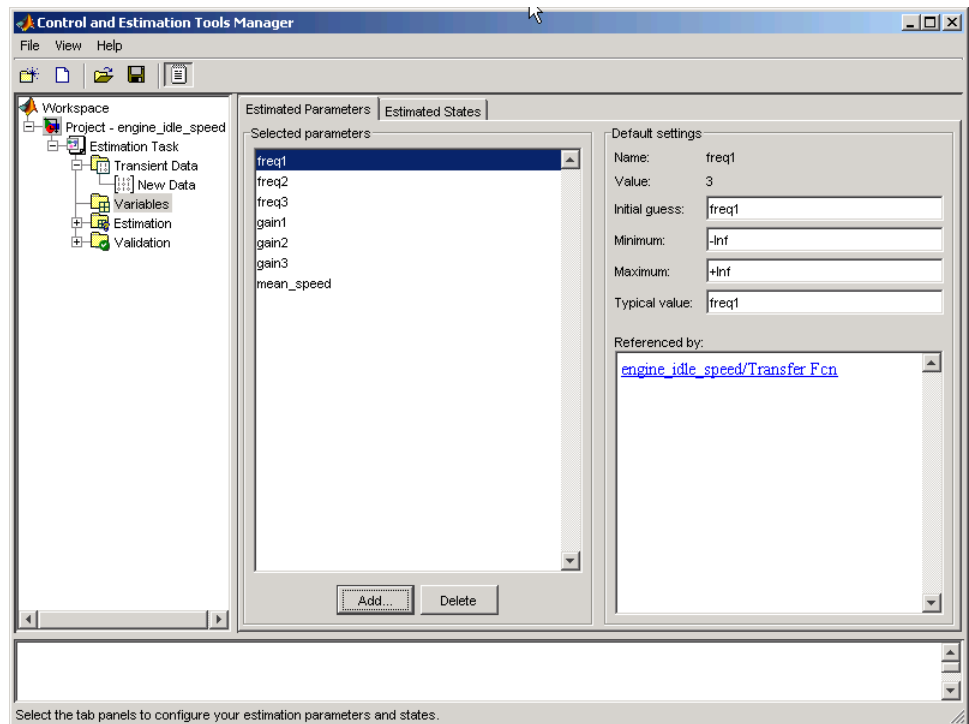
Example: Type *a(1:2)* to select the first column of a 2-by-2 array called *a*.

Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k=a+b$, where a and b are not defined in the model but k is used. To add these independent parameters to the Select Parameters dialog box, see “Specifying Parameter Dependency” on page 2-13.

- 3 Select the last seven parameters: `freq1`, `freq2`, `freq3`, `gain1`, `gain2`, `gain3`, and `mean_speed`, and then click **OK**.

Note You need not estimate the parameters selected here all at once. You can first select all the parameters that you are interested in, and then later select the ones to estimate as described in the next step.

The Control and Estimation Tools Manager now resembles the next figure.



To learn how to specify the settings in the **Default settings** area of the pane, see “Specifying Initial Guesses and Upper/Lower Bounds” on page 2-11.

- 4** In the **New Estimation** node of the Control and Estimation Tools Manager GUI, select the **Parameters** tab . In this pane, you select which parameters to estimate and the range of values for the estimation.
 - a** Select the parameters you want to estimate by selecting the check box in the **Estimate** column.
 - b** Enter initial values for your parameters in the **Initial Guess** column.

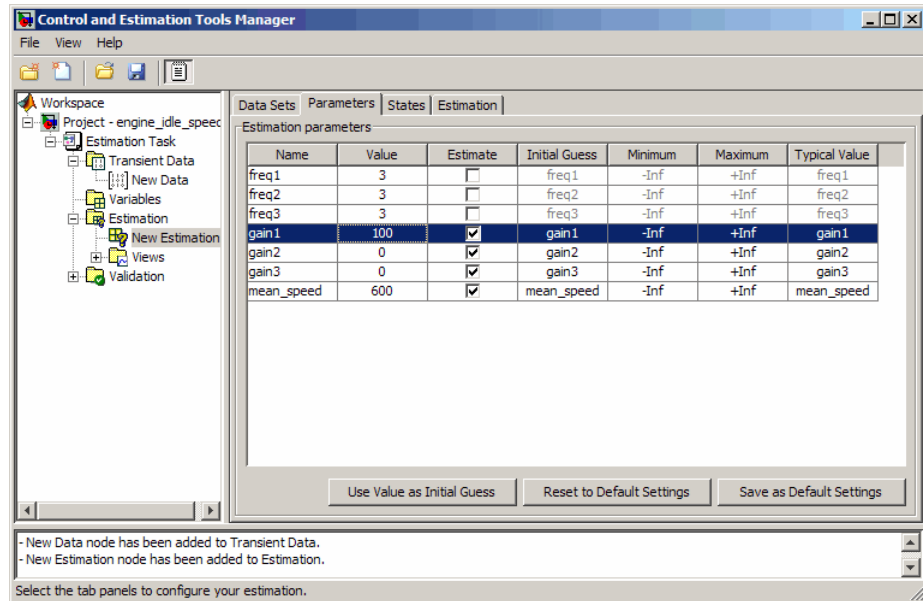
The default values in the **Minimum** and **Maximum** columns are -Inf and +Inf, respectively, but you can select any range you want. For more information, see “Specifying Initial Guesses and Upper/Lower Bounds” on page 2-11.

Note When you specify the **Minimum** and **Maximum** values for the parameters here, it does not affect your settings in the **Variables** node. You make these choices on a per estimation basis. You can move data to and from the **Variables** node into the **Estimation** node.

For this example, select `gain1`, `gain2`, `gain3` and `mean_speed` for estimation and set `gain1` to 10, `gain2` to 100, `gain3` to 50, and `mean_speed` to 500. Alternatively, use any initial values you like.

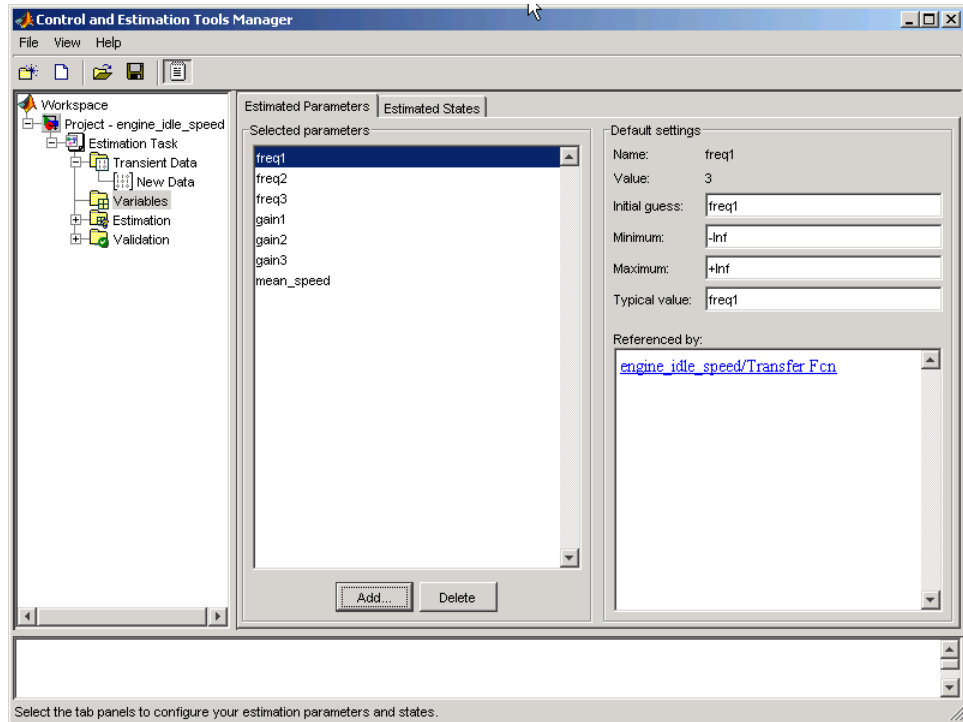
If you have good reason to believe a parameter lies within a finite range, it is usually best not to use the default minimum and maximum values. Often, there are computational advantages in specifying finite bounds if you can. It can be very important to specify lower and upper bounds. For example, if a parameter specifies the weight of a part, be sure to specify 0 as the absolute lower bound if better knowledge is unavailable.

The Control and Estimation Tools Manager now resembles the next figure.



Specifying Initial Guesses and Upper/Lower Bounds

After you select parameters for estimation in the **Variables** node of the Control and Estimation Tools Manager GUI, the **Estimated Parameters** tab in the Control and Estimation Tools Manager looks like the following figure.



For each parameter, use the **Default settings** pane to specify the following:

- **Initial guess** — The value the estimation uses to start the process.
- **Minimum** — The smallest allowable parameter value. The default is $-\text{Inf}$.
- **Maximum** — The largest allowable parameter value. The default is $+\text{Inf}$.
- **Typical value** — The average order of magnitude. If you expect your parameter to vary over several orders of magnitude, enter the number that specified the average order of magnitude you expect. For example, if your initial guess is 10, but you expect the parameter to vary between 10 and 1000, enter 100 (the average of the order of magnitudes) for the typical value.

You use the typical value in two ways:

- To scale parameters with radically different orders of magnitude for equal emphasis during the estimation. For example, try to select the typical values so that

$$\frac{\text{anticipated value}}{\text{typical value}} \cong 1$$

or

$$\frac{\text{initial value}}{\text{typical value}} \cong 1$$

- To put more of less emphasis on specific parameters. Use a larger typical value to put more emphasis on a parameter during estimation.

Specifying Parameter Dependency

Sometimes parameters in your model depend on independent parameters that do not appear in the model. The following steps give an overview of how to specify independent parameters for estimation:

- 1** Add the independent parameters to the model workspace (along with initial values).
- 2** Define a Simulation Start function that runs before each simulation of the model. This Simulation Start function defines the relationship between the dependent parameters in the model and the independent parameters in the model workspace.
- 3** The independent parameters now appear in the Select Parameters dialog box. Add these parameters to the list of parameters to be estimated.

Caution Avoid adding independent parameters together with their corresponding dependent parameters to the lists of parameters to be estimated. Otherwise the estimation could give incorrect results. For example, when a parameter x depends on the parameters a and b , avoid adding all three parameters to the list.

For an example of how to specify independent parameters, see “Example: Specifying Independent Parameters for Estimation” on page 2-14.

Example: Specifying Independent Parameters for Estimation

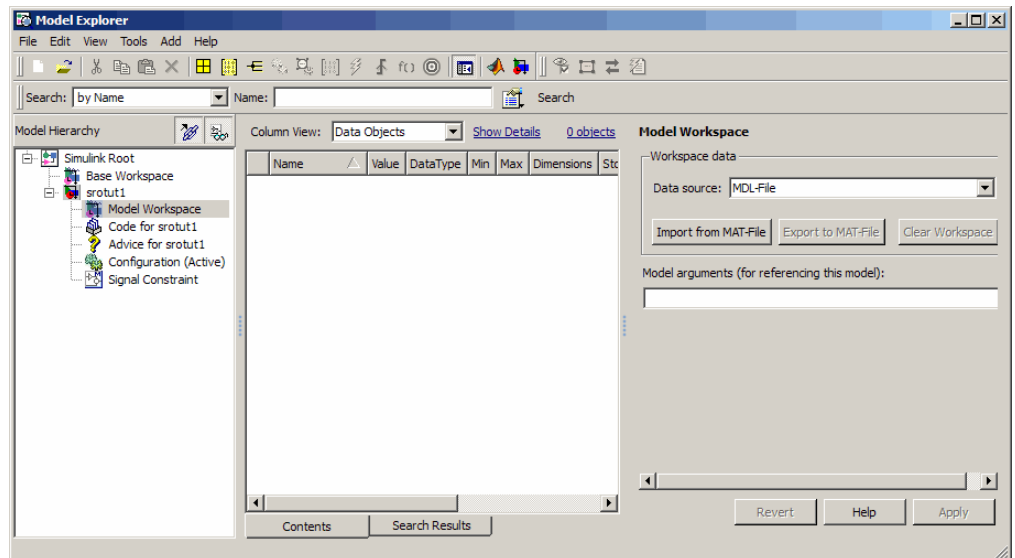
Assume that the parameter K_{int} in the model `srotut1` is related to the parameters x and y according to the relationship $K_{int}=x+y$. Also assume that the initial values of x and y are 1 and -0.7 respectively. To estimate x and y instead of K_{int} , first define these parameters in the model workspace. To do this:

- 1 At the MATLAB prompt, type

```
srotut1
```

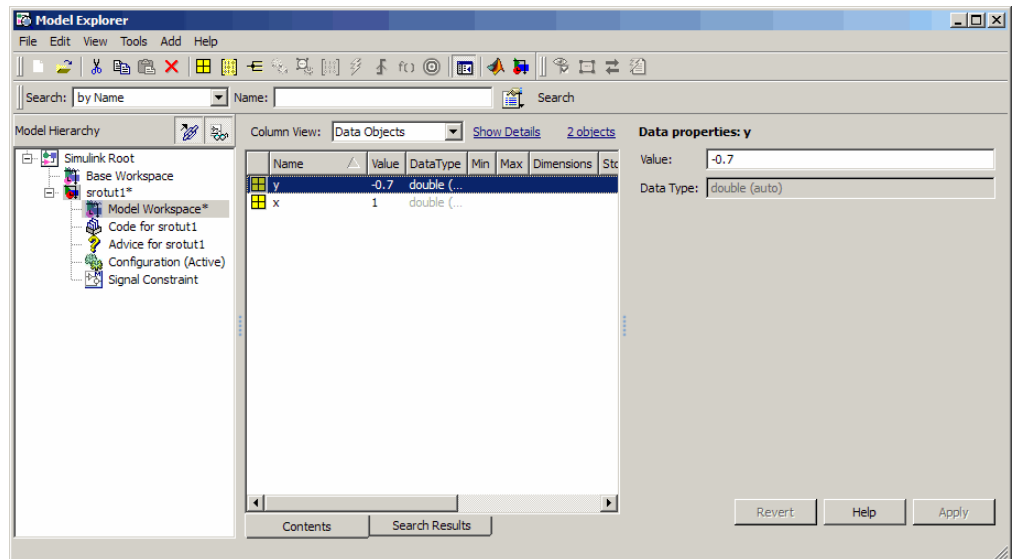
This opens the `srotut1` model window.

- 2 Select **View > Model Explorer** from the `srotut1` window to open the Model Explorer window.
- 3 In the Model Hierarchy tree, select `srotut1 > Model Workspace`.



- 4 Select **Add > MATLAB Variable** to add a new variable to the model workspace. A new variable with a default name **Var** appears in the **Name** column.
- 5 Double-click **Var** to make it editable and change the variable name to **x**. Edit the initial **Value** to 1.
- 6 Repeat steps 4 and 5 to add a variable **y** with an initial value of **-0.7**.

The Model Explorer window resembles the following figure.



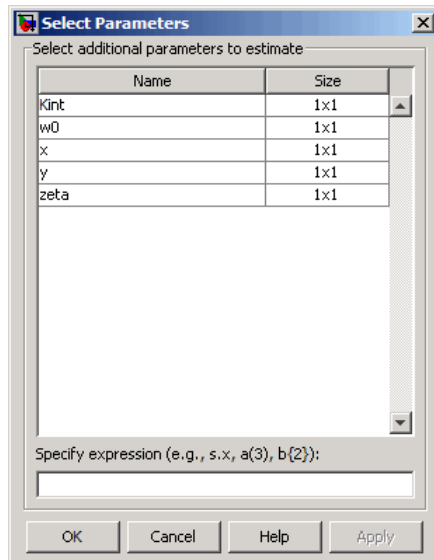
- 7 To add the Simulation Start function that defines the relationship between **Kint** and the independent parameters **x** and **y**, select **File > Model Properties** in the **srotut1** model window.
- 8 In the Model Properties window, click the **Callbacks** tab.
- 9 To enter a Simulation start function, select **StartFcn***, and type the name of a new function. For example, **srotut1_start** in the **Simulation start function** panel. Then, click **OK**.

- 10** Create a MATLAB file named `srotut1_start`. The content of the file defines the relationship between the parameters in the model and the parameters in the workspace. For this example, the content resembles the following:

```
wks = get_param(gcs, 'ModelWorkspace')
x = wks.evalin('x')
y = wks.evalin('y')
Kint = x+y;
```

Note You must first use the `get_param` function to get the variables `x` and `y` from the model workspace before you can use them to define `Kint`.

When you select parameters for estimation in the **Variables** node of Control and Estimation Tools Manager, `x` and `y` appear in the Select Parameters dialog box.



Specifying Initial States

- “When to Specify Initial States Versus Estimate Initial States” on page 2-17
- “How to Specify Initial States in the GUI” on page 2-17

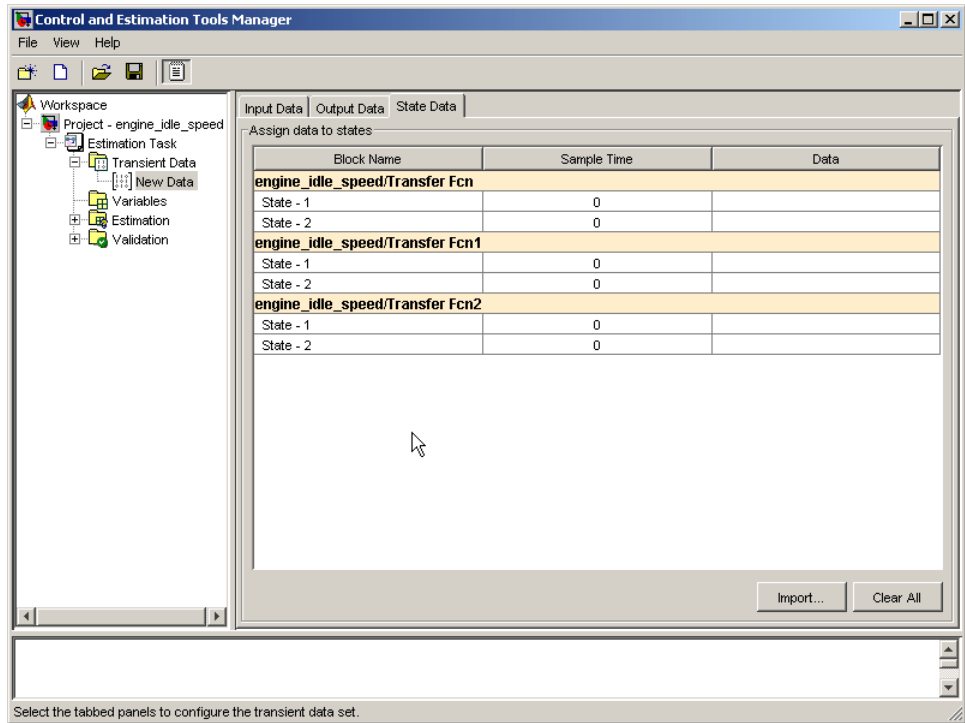
When to Specify Initial States Versus Estimate Initial States

Often, sets of measured data are collected at various times and under different initial conditions. When you estimate model parameters using one data set and subsequently run another estimation with a second data set, your parameter values may not match. Given that the Simulink Design Optimization software attempts to find constant values for parameters, this is clearly a problem.

You can estimate the initial conditions using procedures that are similar to those you use to estimate parameters. You can then use these initial condition estimates as a basis for estimating parameters for your Simulink model. The Control and Estimation Tools Manager has an **Estimated States** pane that lists the states available for initial condition estimation. To learn how to estimate initial states, see “Estimating Initial States” on page 2-65.

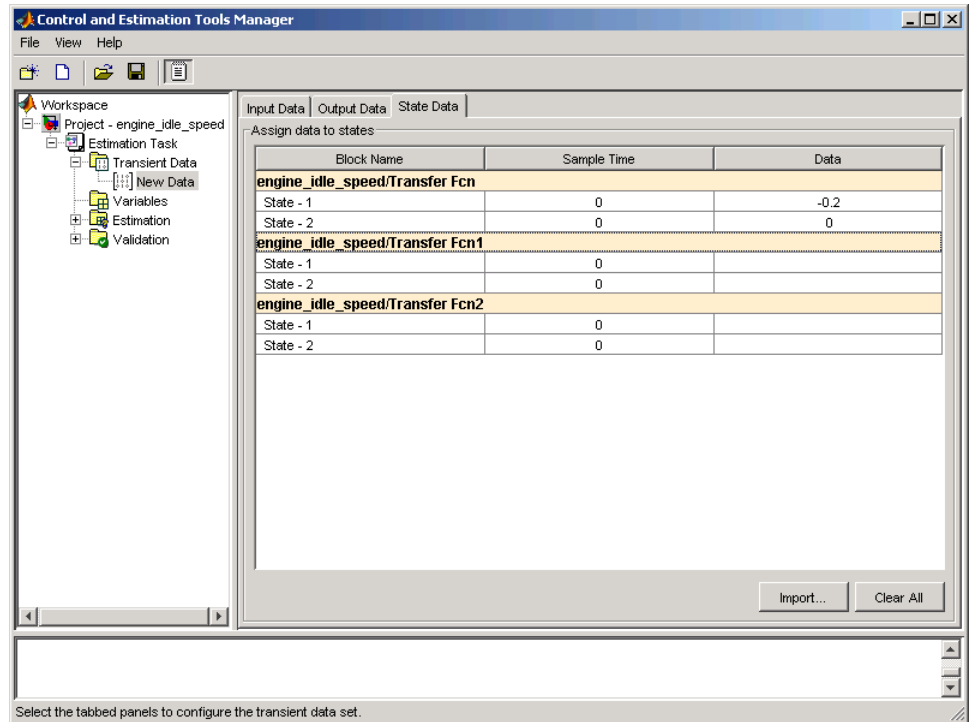
How to Specify Initial States in the GUI

After you select parameters for estimation, as described in “Specifying Parameters to Estimate” on page 2-6, you can specify initial conditions of states in your model. By default, the estimation uses initial conditions specified in the Simulink model. If you want to specify initial conditions other than the defaults, use the **State Data** tab. You can select the **State Data** tab in the **New Data** node under the **Transient Data** node in the **Workspace** tree.



To specify the initial condition of a state for the `engine_idle_speed` model:

- 1 Select the **Data** cell associated with the state.
- 2 Enter the initial conditions. In this example, enter `-0.2` for **State - 1** of the `engine_idle_speed/Transfer Fcn`. For **State - 2**, enter `0`.



Selecting Views for Plotting

- “Types of Plots” on page 2-19
- “Basic Steps for Creating Plots” on page 2-20

Types of Plots

You can choose the plot type from the **Plot Type** drop-down list. The following types of plots are available for viewing and evaluating the estimation:

- **Cost function** — Plot the cost function values.
- **Measured and simulated** — Plot empirical data against simulated data.
- **Parameter sensitivity** — Plot the rate of change of the cost function as a function of the change in the parameter. That is, plot the derivative of the cost function with respect to the parameter being varied.
- **Parameter trajectory** — Plot the parameter values as they change.
- **Residuals** — Plot the error between the experimental data and the simulated output.

Basic Steps for Creating Plots

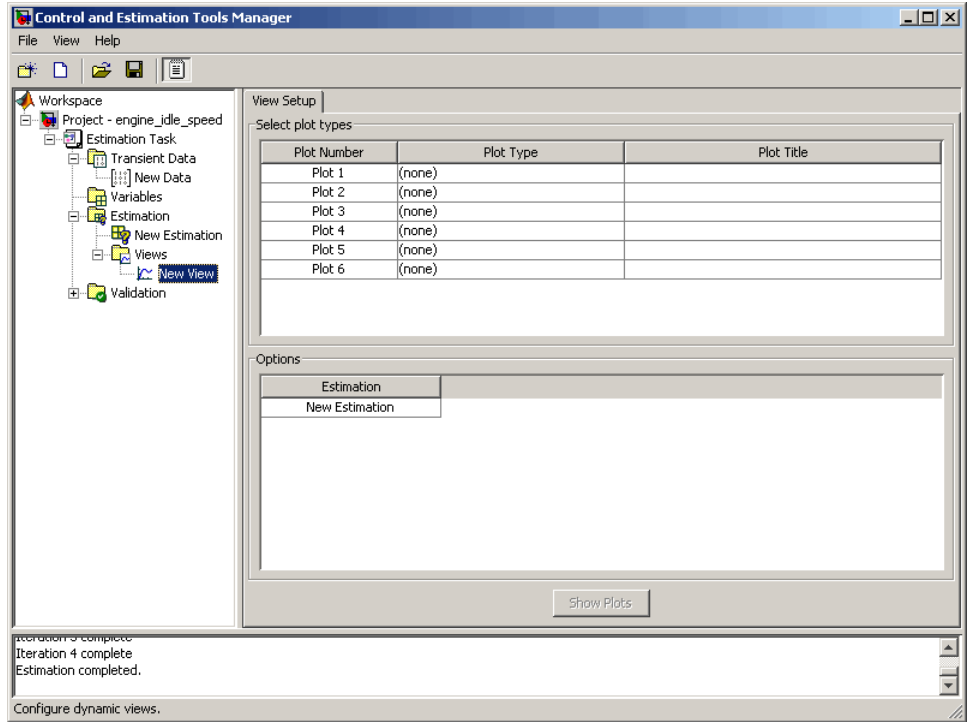
Before you begin estimating the parameters, you must create the plots for viewing the progress of the estimation.

Note An estimation must be created before creating views. Otherwise, the **Options** table will be empty. To learn more, see “Creating an Estimation Task” on page 2-3.

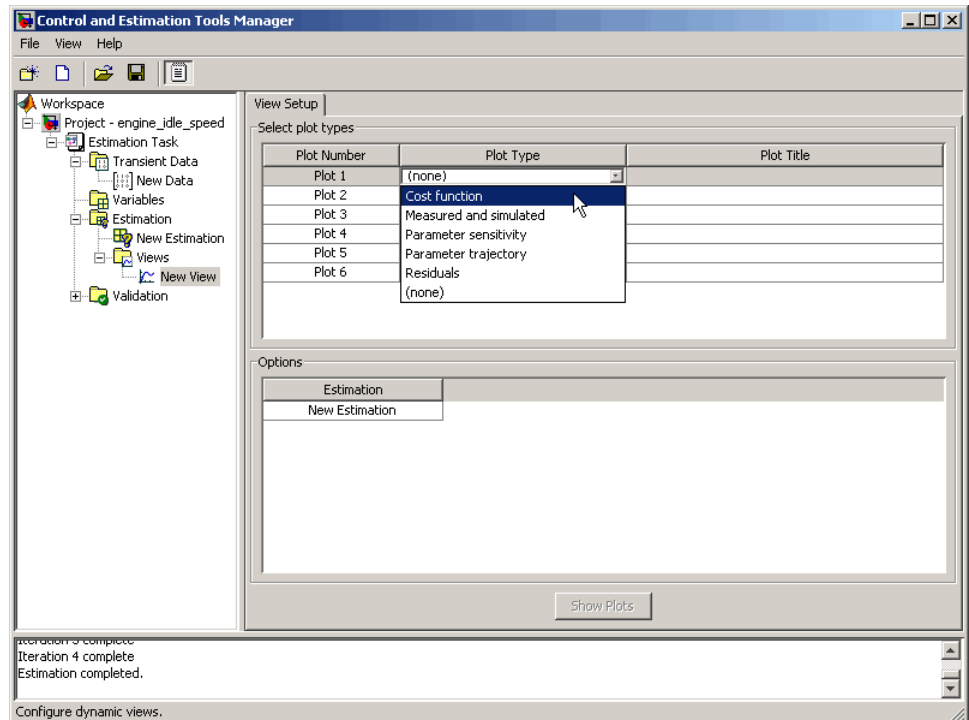
To create plots for viewing the estimation progress, follow the steps below:

- 1** Right-click the **Views** node in the Control and Estimation Tools Manager and select **New**.

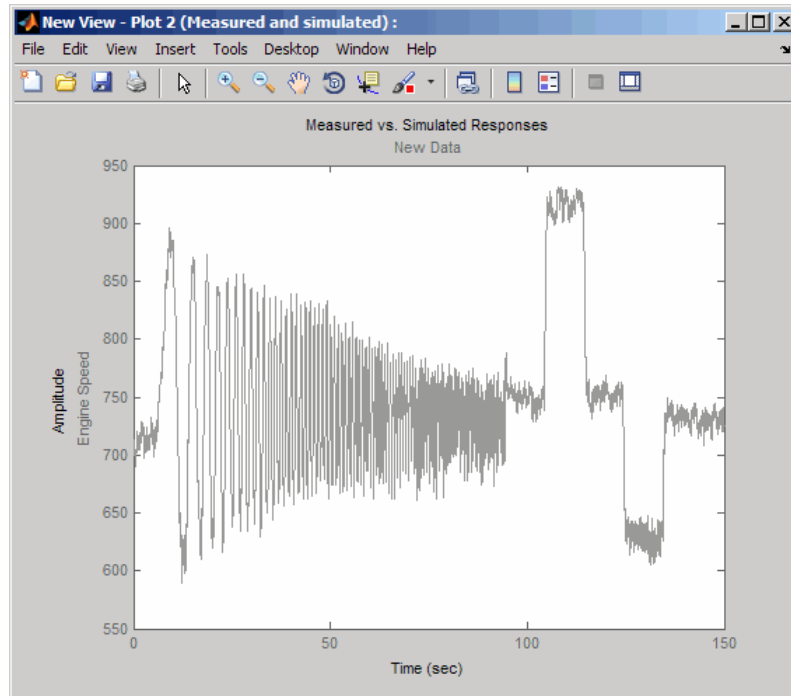
2 In the **Workspace** tree, select **New View** to open the **View Setup** pane.



- 3** In the **Select plot types** table, select the **Plot Type** from the drop-down list. In this example, select **Cost function**.



- 4** Select **Measured and simulated** as the **Plot Type** for **Plot 2**. This plot will be used in validating estimated parameters.
- 5** In the **Options** area, select the check-box for both **Plot 1** and **Plot 2**.
- 6** Click **Show Plots**. This displays an empty cost function plot and a plot of the measured data.



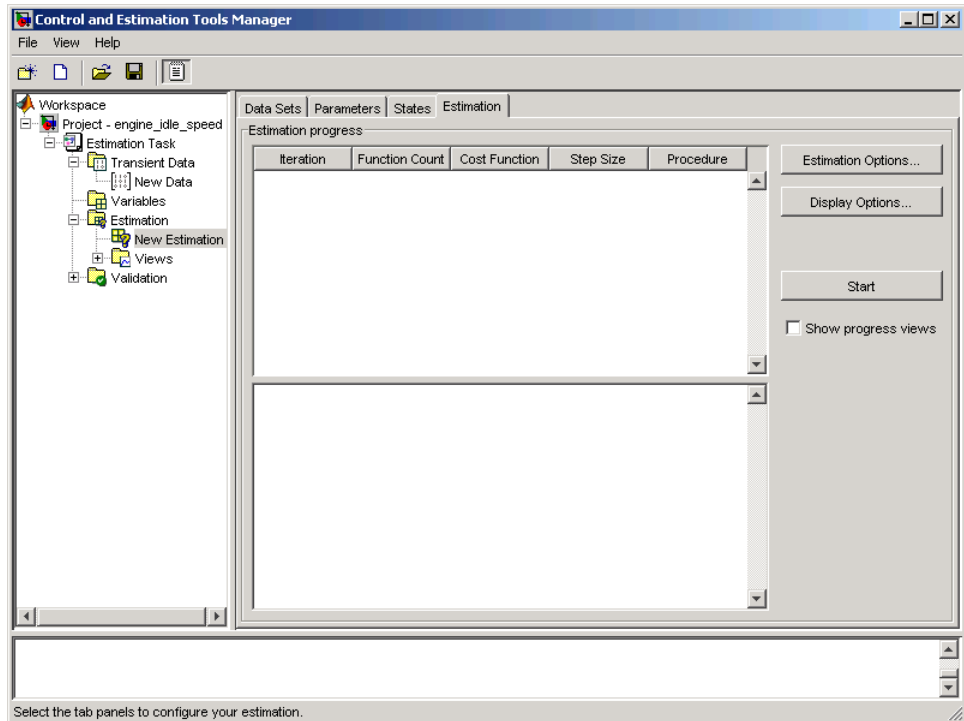
When you perform the estimation, the plot updates automatically.

Specifying Estimation Options

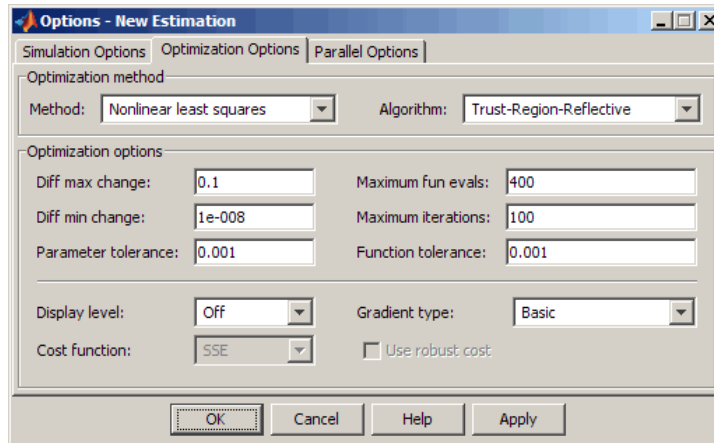
- “Accessing Estimation Options” on page 2-24
- “Supported Estimation Methods” on page 2-25
- “Selecting Optimization Termination Options” on page 2-27
- “Selecting Additional Optimization Options” on page 2-27
- “Specifying Goodness of Fit Criteria (Cost Function)” on page 2-28
- “How to Specify Estimation Options in the GUI” on page 2-28

Accessing Estimation Options

In the New **Estimation** node in the **Workspace** tree, click the **Estimation** tab.



Click **Estimation Options**. This action opens the Options- New Estimation dialog box where you can specify the estimation method, algorithm options and cost function for the estimation.

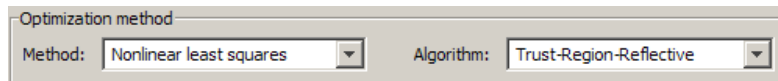


The following sections describe the estimation method settings and cost function:

- “Supported Estimation Methods” on page 2-25
- “Selecting Optimization Termination Options” on page 2-27
- “Selecting Additional Optimization Options” on page 2-27
- “Specifying Goodness of Fit Criteria (Cost Function)” on page 2-28

Supported Estimation Methods

Both the **Method** and **Algorithm** options define the optimization method. Use the **Optimization method** area of the Options dialog box to set the estimation method and its algorithm.



For the **Method** option, the four choices are:

- **Nonlinear least squares** (default) — Uses the Optimization Toolbox™ nonlinear least squares function `lsqnonlin`.
- **Gradient descent** — Uses the Optimization Toolbox function `fmincon`.

- **Pattern search** — Uses the pattern search method `patternsearch`. This option requires Global Optimization Toolbox software.
- **Simplex search** — Uses the Optimization Toolbox function `fminsearch`, which is a direct search method. **Simplex search** is most useful for simple problems and is sometimes faster than `fmincon` for models that contain discontinuities.

The following table summarizes the **Algorithm** options for the Nonlinear least squares and Gradient descent estimation methods:

Method	Algorithm Option	Learn More
Nonlinear least squares	<ul style="list-style-type: none"> • Trust-Region-Reflective (default) • Levenberg-Marquardt 	In the Optimization Toolbox documentation, see: <ul style="list-style-type: none"> • “Large Scale Trust-Region Reflective Least Squares” • “Levenberg-Marquardt Method”
Gradient descent	<ul style="list-style-type: none"> • Active-Set (default) • Interior-Point • Trust-Region-Reflective 	In the Optimization Toolbox documentation, see: <ul style="list-style-type: none"> • “fmincon Active Set Algorithm” • “fmincon Interior Point Algorithm” • “fmincon Trust Region Reflective Algorithm”

Selecting Optimization Termination Options

Specify termination options in the **Optimization options** area.

The screenshot shows a dialog box titled "Optimization options" with six input fields arranged in two columns. The left column contains "Diff max change" (0.1), "Diff min change" (1e-008), and "Parameter tolerance" (1e-006). The right column contains "Maximum fun evals" (400), "Maximum iterations" (100), and "Function tolerance" (1e-006).

Option	Value
Diff max change:	0.1
Diff min change:	1e-008
Parameter tolerance:	1e-006
Maximum fun evals:	400
Maximum iterations:	100
Function tolerance:	1e-006

Several options define when the optimization terminates:

- **Diff max change** — The maximum allowable change in variables for finite-difference derivatives. See `fmincon` in the Optimization Toolbox documentation for details.
- **Diff min change** — The minimum allowable change in variables for finite-difference derivatives. See `fmincon` in the Optimization Toolbox documentation for details.
- **Parameter tolerance** — Optimization terminates when successive parameter values change by less than this number.
- **Maximum fun evals** — The maximum number of cost function evaluations allowed. The optimization terminates when the number of function evaluations exceeds this value.
- **Maximum iterations** — The maximum number of iterations allowed. The optimization terminates when the number of iterations exceeds this value.
- **Function tolerance** — The optimization terminates when successive function values are less than this value.

By varying these parameters, you can force the optimization to continue searching for a solution or to continue searching for a more accurate solution.

Selecting Additional Optimization Options

At the bottom of the **Optimization options** pane is a group of additional optimization options.

The screenshot shows two dropdown menus at the bottom of the dialog box. The first is labeled "Display level:" and is set to "None". The second is labeled "Gradient type:" and is set to "Basic".

Option	Value
Display level:	None
Gradient type:	Basic

Additional options for optimization include:

- **Display level** — Specifies the form of the output that appears in the MATLAB command window. The options are *Iteration*, which displays information after each iteration, *None*, which turns off all output, *Notify*, which displays output only if the function does not converge, and *Final*, which only displays the final output. Refer to the Optimization Toolbox documentation for more information on what type of iterative output each method displays.
- **Gradient type** — When using Gradient Descent or Nonlinear least squares as the **Method**, the gradients are calculated based on finite difference methods. The *Refined* method offers a more robust and less noisy gradient calculation method than *Basic*, although it does take longer to run optimizations using the *Refined* method.

Specifying Goodness of Fit Criteria (Cost Function)

The *cost function* is a function that estimation methods attempt to minimize. You can specify the cost function at the bottom of the **Optimization options** area.



You have the following options when selecting a cost function:

- **Cost function** — The default is SSE (sum of squared errors), which uses a least-squares approach. You can also use SAE, the sum of absolute errors.
- **Use robust cost** — Makes the optimizer use a robust cost function instead of the default least-squares cost. This is useful if the experimental data has many outliers, or if your data is noisy.

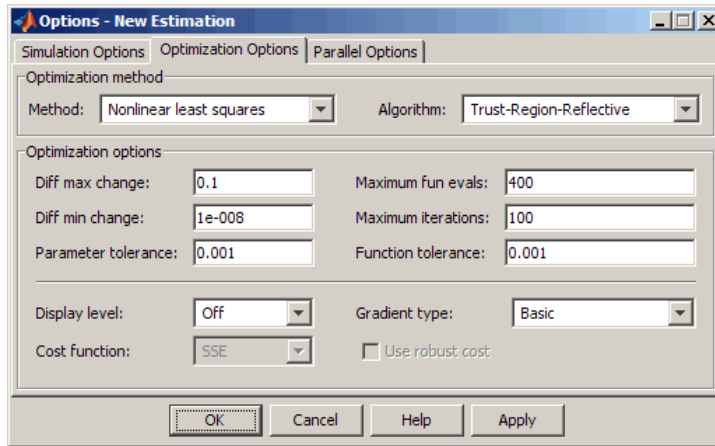
How to Specify Estimation Options in the GUI

You can set several options to tune the results of the estimation. These options include the optimization methods and their tolerances.

To set options for estimation:

- 1 Select the **New Estimation** node in the **Workspace** tree.

- 2 Click the **Estimation** tab.
- 3 Click **Estimation Options** to open the Options dialog box.
- 4 Click the **Optimization Options** tab and specify the options.



Specifying Simulation Options

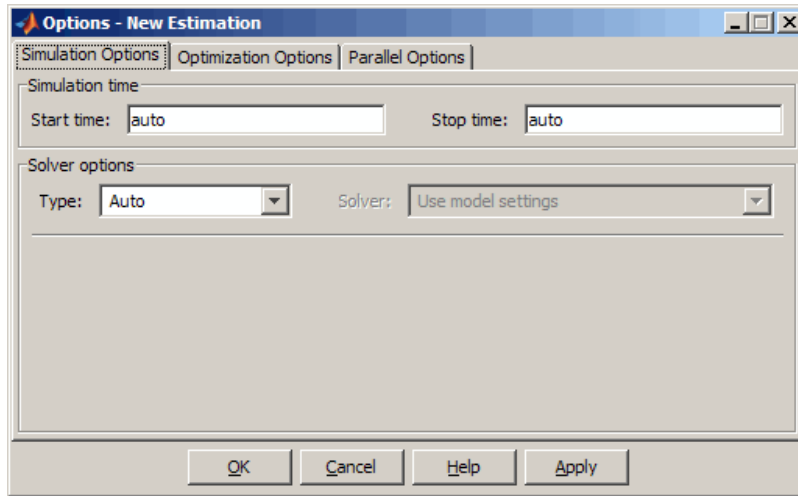
- “Accessing Simulation Options” on page 2-29
- “Selecting Simulation Time” on page 2-30
- “Selecting Solvers” on page 2-32

Accessing Simulation Options

To estimate parameters of a model, Simulink Design Optimization software runs simulations of the model.

To set options for simulation:

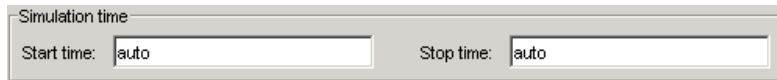
- 1 Select the **New Estimation** node in the **Workspace** tree.
- 2 Click the **Estimation** tab.
- 3 Click **Estimation Options** to open the Options dialog box.



- 4** Click the **Simulation Options** tab and specify the options, as described in the following sections:
- “Selecting Simulation Time” on page 2-30
 - “Selecting Solvers” on page 2-32

Selecting Simulation Time

You can specify the simulation start and stop times in the **Simulation time** area of the **Simulation Options** tab.

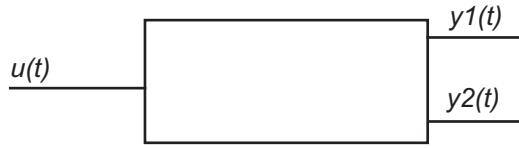


By default, **Start time** and **Stop time** are automatically computed based on the start and stop times specified in the Simulink model.

To set alternative start and stop times for the optimization, enter the new times under **Simulation time**. This action overwrites the simulation start and stop times specified in the Simulink model.

Simulation Time for Data Sets with Different Time Lengths. Simulink Design Optimization software can simulate models containing empirical data sets of different time lengths. You can use experimental data sets for estimation that contain I/O samples collected at different time points.

The following example shows a single-input, two-output model for which you want to estimate the parameters.



The model uses two output data sets containing transient data samples for parameter estimation:

- Output $y1(t)$ at time points $t1 = \{t_1^1, t_2^1, \dots, t_n^1\}$.
- Output $y2(t)$ at time points $t2 = \{t_1^2, t_2^2, \dots, t_m^2\}$.

The simulation time t is computed as:

$$t = t1 \cup t2 = \{t_1^1, t_1^2, t_2^1, t_2^2, \dots, t_n^1, t_m^2\}$$

This new set ranges from $tmin$ to $tmax$. The values $tmin$ and $tmax$ represent the minimum and maximum time points in t respectively.

When you run the estimation, the model is simulated over the time range t . Simulink extracts the simulated data for each output based on the following criteria:

- **Start time** — Typically, the start time in the Simulink model is set to 0. For a nonzero start time, the simulated data corresponding to time points before t_1^1 for $y1(t)$ and t_1^2 for $y2(t)$ are discarded.

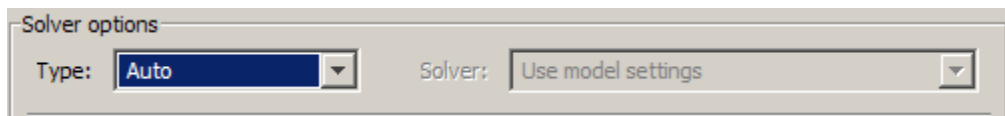
- **Stop time** — If the stop time $t_{stop} \geq t_{max}$, the simulated data corresponding to time points in $t1$ are extracted for $y1(t)$. Similarly, the simulated data for time points in $t2$ are extracted for $y2(t)$.

If the stop time $t_{stop} < t_{max}$, the data spanning time points $> t_{stop}$ are discarded for both $y1(t)$ and $y2(t)$.

Selecting Solvers

When running the estimation, the software solves the dynamic system using one of several Simulink solvers.

Specify the solver type and its options in the **Solver options** area of the **Simulation Options** tab of the Options dialog box.



The solver can be one of the following **Type**:

- **Auto (default)** — Uses the simulation settings specified in the Simulink model.
- **Variable-step** — Variable-step solvers keep the error within specified tolerances by adjusting the step size the solver uses. For example, if the states of your model are likely to vary rapidly, you can use a variable-step solver for faster simulation. For more information on the variable-step solver options, see “Variable-Step Solver Options” on page 2-33.
- **Fixed-step** — Fixed-step solvers use a constant step-size. For more information on the fixed-step solver options, see “Fixed-Step Solver Options” on page 2-34.

See “Choosing a Solver” in the Simulink documentation for information about solvers.

Note To obtain faster simulations during estimation, you can change the solver **Type** to **Variable-step** or **Fixed-step**. However, the estimated parameter values apply only for the chosen solver type, and may differ from values you obtain using settings specified in the Simulink model.

Variable-Step Solver Options. When you select **Variable-step** as the solver **Type**, you can choose one of the following as the **Solver**:

- Discrete (no continuous states)
- ode45 (Dormand-Prince)
- ode23 (Bogacki-Shampine)
- ode113 (Adams)
- ode15s (stiff/NDF)
- ode23s (stiff/Mod. Rosenbrock)
- ode23t (Mod. stiff/Trapezoidal)
- ode23tb (stiff/TR-BDF2)

Solver options

Type: Variable-step Solver: ode45 (Dormand-Prince)

Maximum step size: auto Relative tolerance: 1e-3

Minimum step size: auto Absolute tolerance: auto

Initial step size: auto Zero crossing control: On

You can also specify the following parameters that affect the step-size of the simulation:

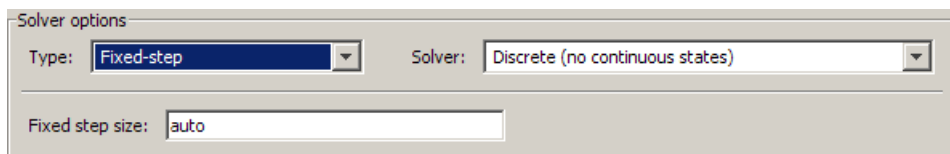
- **Maximum step size** — The largest step-size the solver can use during a simulation.
- **Minimum step size** — The smallest step-size the solver can use during a simulation.
- **Initial step size** — The step-size the solver uses to begin the simulation.

- **Relative tolerance** — The largest allowable relative error at any step in the simulation.
- **Absolute tolerance** — The largest allowable absolute error at any step in the simulation.
- **Zero crossing control** — Set to on for the solver to compute exactly where the signal crosses the x -axis. This option is useful when using functions that are nonsmooth and the output depends on when a signal crosses the x -axis, such as absolute values.

By default, the software automatically chooses the values for these options. To specify your own values, enter them in the appropriate fields. For more information, see “Solver Pane” in the Simulink documentation.

Fixed-Step Solver Options. When you select **Fixed-step** as the solver **Type**, you can choose one of the following as the **Solver**:

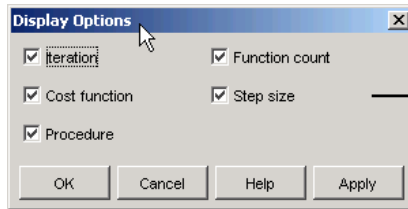
- Discrete (no continuous states)
- ode5 (Dormand-Prince)
- ode4 (Runge-Kutta)
- ode3 (Bogacki-Shampine)
- ode2 (Heun)
- ode1 (Euler)



You can also specify the **Fixed step size** value, which determines the step size the solver uses during the simulation. By default, the software automatically chooses a value for this option. For more information, see “Fixed-step size (fundamental sample time)” in the Simulink documentation.

Specifying Display Options

You can specify the display options by clicking **Display Options** in the **Estimation** tab in the Control and Estimation tools Manager. This opens the following dialog box.



By default, all boxes are checked. Uncheck any feature that you don't want to view during the estimation process.

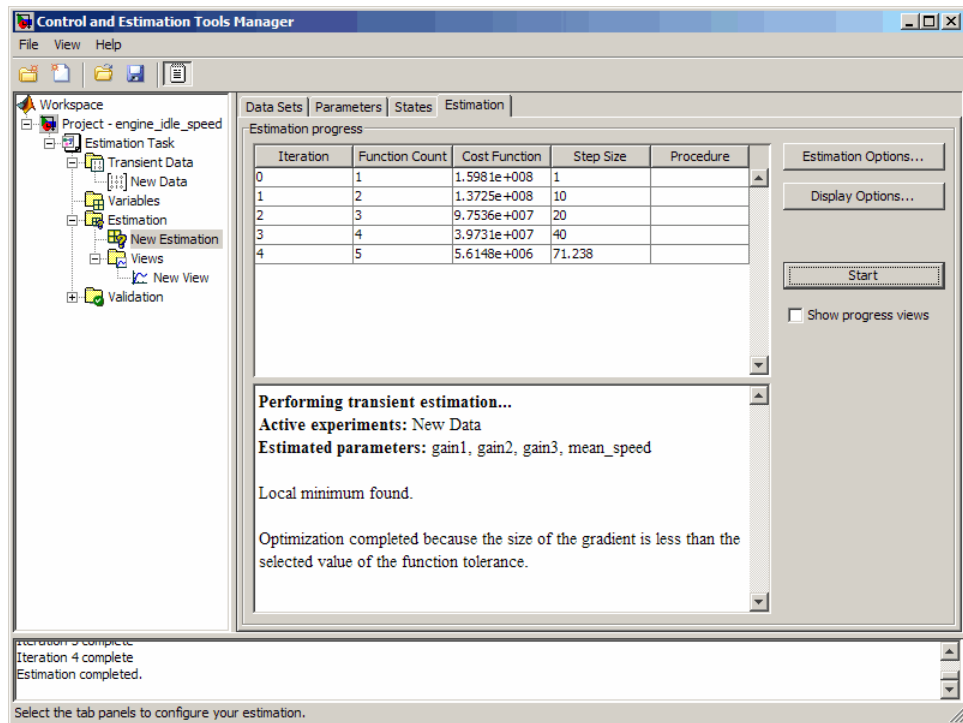
Clearing a check box implies that feature will not appear in the display table as the estimation progresses. To learn more about the display table, see “Displaying Iterative Output” in the Optimization Toolbox documentation.

Estimating Parameters in the GUI

Before you begin estimating the parameters, you must have configured the estimation data and parameters, and specified estimation and simulation options, as described in “Configuring Parameter Estimation in the GUI” on page 2-3.

To start the estimation, select the **New Estimation** node in the Control and Estimation Tools Manager and select the **Estimation** tab.

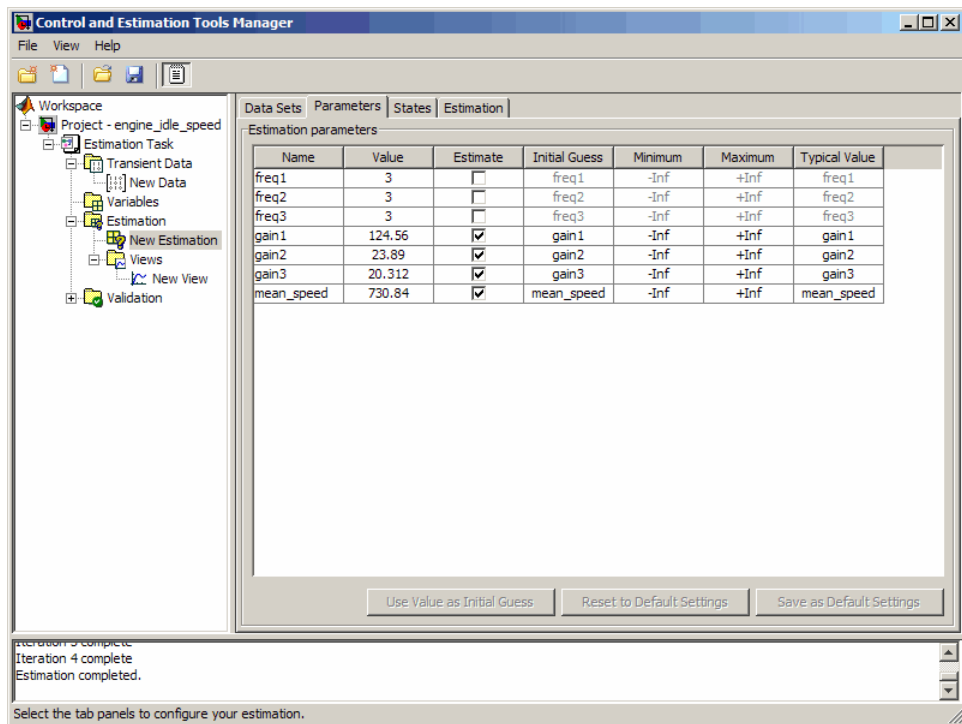
Click **Start** to begin the estimation process. At the end of the iterations, the window should resemble the following:



Usually, a lower cost function value indicates a successful estimation, meaning that the experimental data matches the model simulation with the estimated parameters.

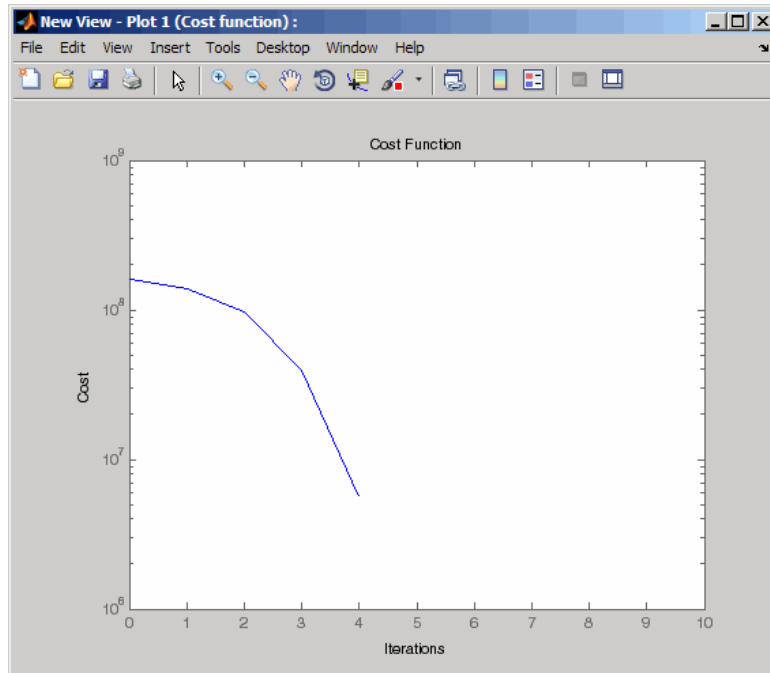
Note For information on types of problems you may encounter using optimization solvers, see “Steps to Take After Running a Solver” in the Optimization Toolbox documentation.

The **Estimation** pane displays each iteration of the optimization methods. To see the final values for the parameters, click the **Parameters** tab.



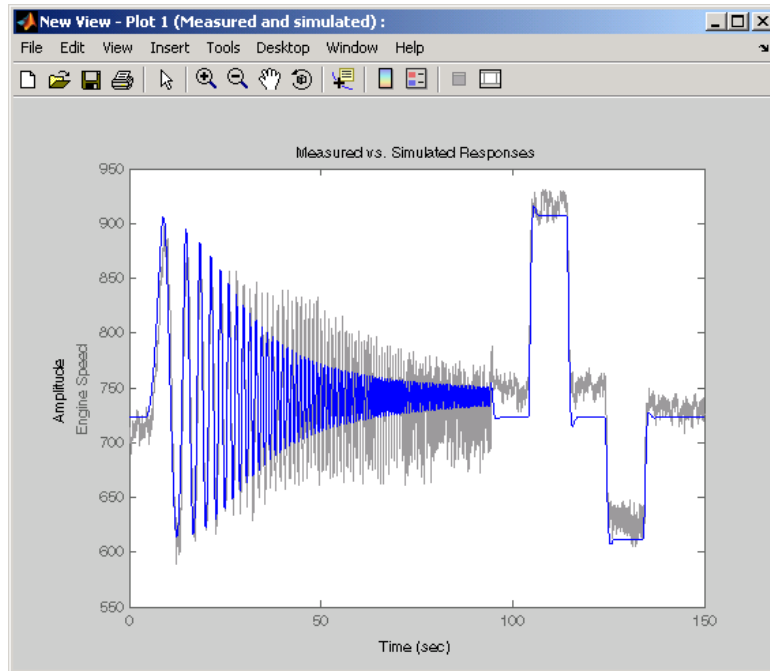
The values of these parameters are also updated in the MATLAB workspace. If you specify the variable name in the **Initial Guess** column, you can restart the estimation from where you left off at the end of a previous estimation.

After the estimation process completes, the cost function minimization plot appears as shown in the following figure.



If the optimization went well, you should see your cost function converge on a minimum value. The lower the cost, the more successful is the estimation.

You can also examine the measured versus simulated data plot to see how closely the simulated data matches the measured estimation data. The next figure shows the measured versus simulated data plot generated by running the estimation of the engine_idle_speed model.



Validating Parameters in the GUI

In this section...
“Basic Steps for Model Validation” on page 2-40
“Loading and Importing the Validation Data” on page 2-41
“Performing Validation” on page 2-43
“Comparing Residuals” on page 2-47

Basic Steps for Model Validation

After you complete estimating the parameters, as described in “Estimating Parameters in the GUI” on page 2-36, you must validate the results against another set of data.

The steps to validate a model using the Control and Estimation Tools Manager are:

- 1** Import the validation data set to the **Transient Data** node.
- 2** Add a new validation task in the **Validation** node in the **Workspace** tree.
- 3** Configure the validation settings by selecting the plot types and the validation data set from the **Validation Setup** pane.
- 4** Click **Show Plots** in the **Validation Setup** pane and view the results in the plot window.
- 5** Compare the validation plots to the corresponding view plots to see if they match.

The basic difference between the validation and views features is that you can run validation after the estimation is complete. All views should be set up before an estimation, and you can watch the views update in real time. Validations can use other validation data sets for comparison with the model response. Also, validations appear after you have completed an estimation and do not update.

You can validate your data by comparing measured vs. simulated data for your estimation data and validation data sets. Also, it is often useful to compare residuals in the same way.

Loading and Importing the Validation Data

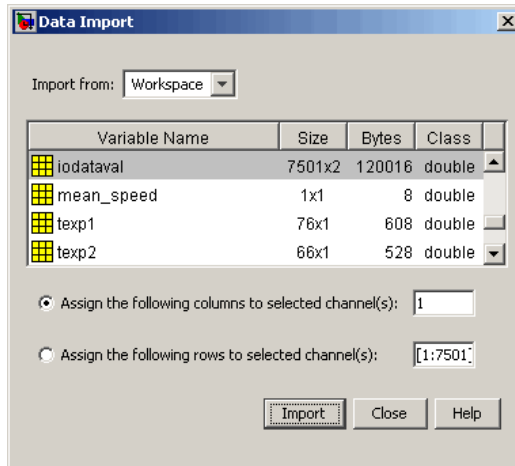
To validate the estimated parameters computed in “Estimating Parameters in the GUI” on page 2-36, you must first import the data into the Control and Estimation Tools Manager GUI.

To load the validation data, type

```
load iodataval
```

at the MATLAB prompt. This loads the data into the MATLAB workspace. The next step is to import this data into the Control and Estimation Tools Manager. See “Importing Data into the GUI” on page 1-5 for information on importing data, but the quickest way is to follow these steps:

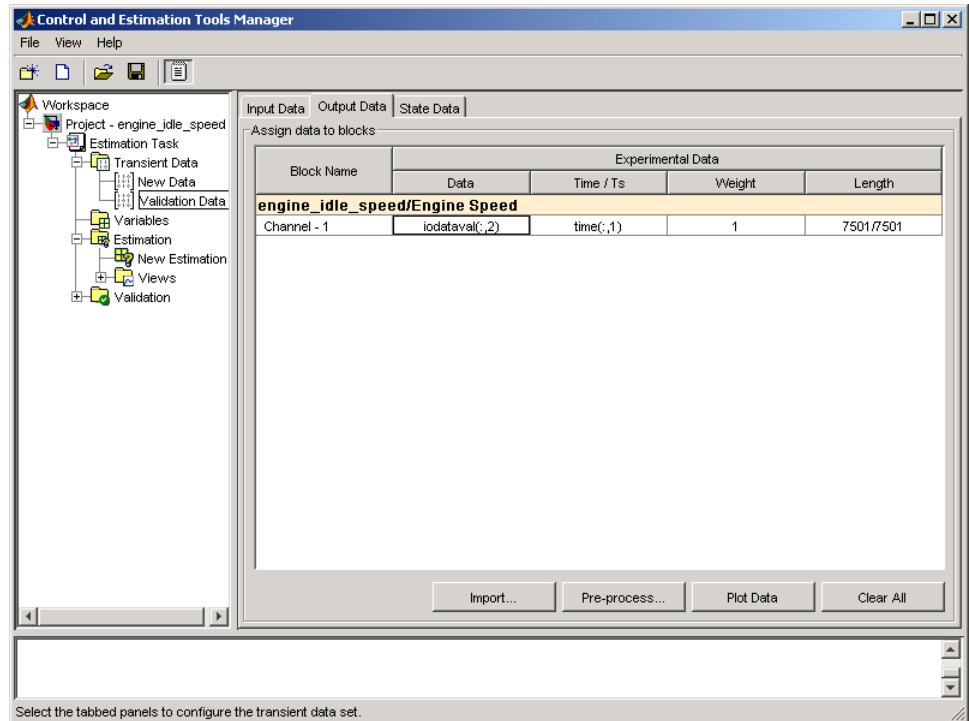
- 1** Right-click the **Transient Data** node in the **Workspace** tree in the Control and Estimation Tools Manager and select **New**.
- 2** Select **New Data (2)** from the **Transient Data** pane.
- 3** Right-click the **New Data (2)** node in the **Workspace** tree and select **Rename**. Change the name of the data to **Validation Data**.
- 4** In the **Input Data** pane, select the **Data** cell associated with Channel 1 - 1 and click **Import**. In the Data Import dialog box, select `iodataval` and assign column 1 to the selected channel by entering 1 in the **Assign columns** field. Click **Import** to import the input data.



- 5 Select the **Time/Ts** cell and import time using the Data Import dialog box.
- 6 Similarly, in the **Output Data** pane, select **Time/Ts** and import time.
- 7 In the **Output Data** pane, select the **Data** cell associated with Channel 1 - 1 and click **Import**. Import the second column of data in **iodataval** by

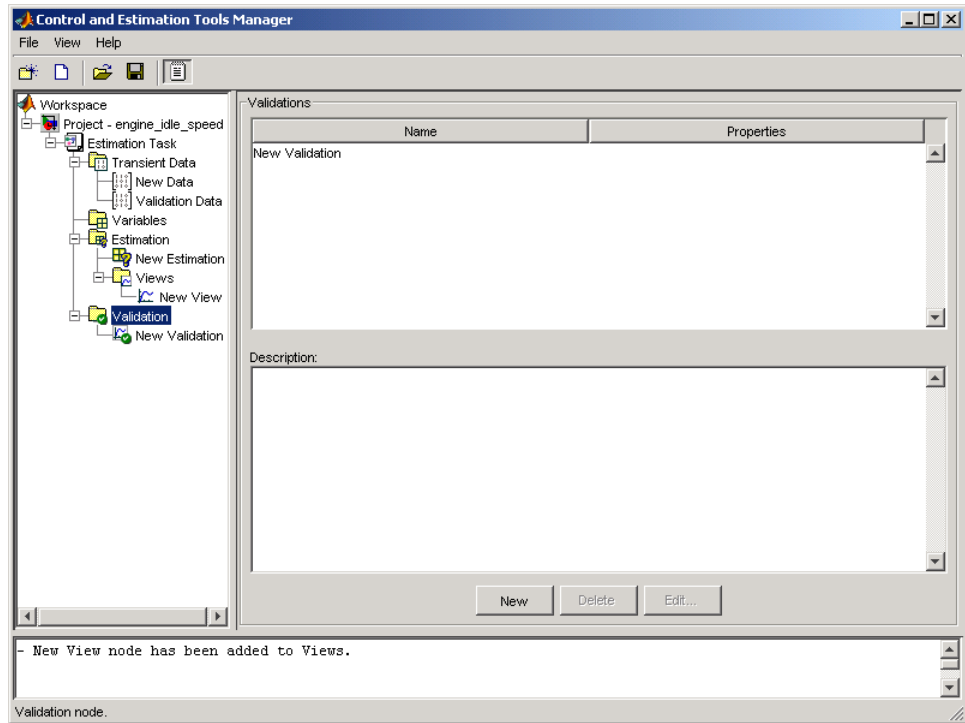
selecting it from the list in the Import Data dialog box and entering 2 in the **Assign columns** field. Click **Import** to import the output data.

The Control and Estimation Tools Manager should resemble the next figure.



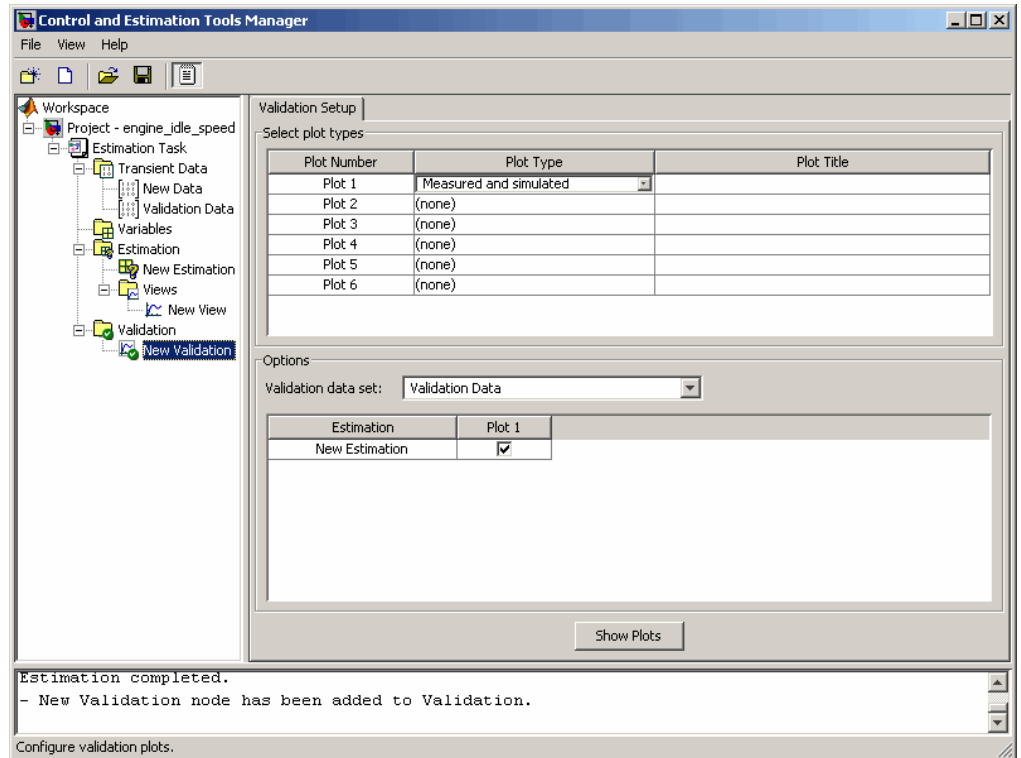
Performing Validation

After you import the validation data, as described in “Loading and Importing the Validation Data” on page 2-41, right-click the **Validation** node and select **New**. This creates a **New Validation** node in the Control and Estimation Tools Manager.

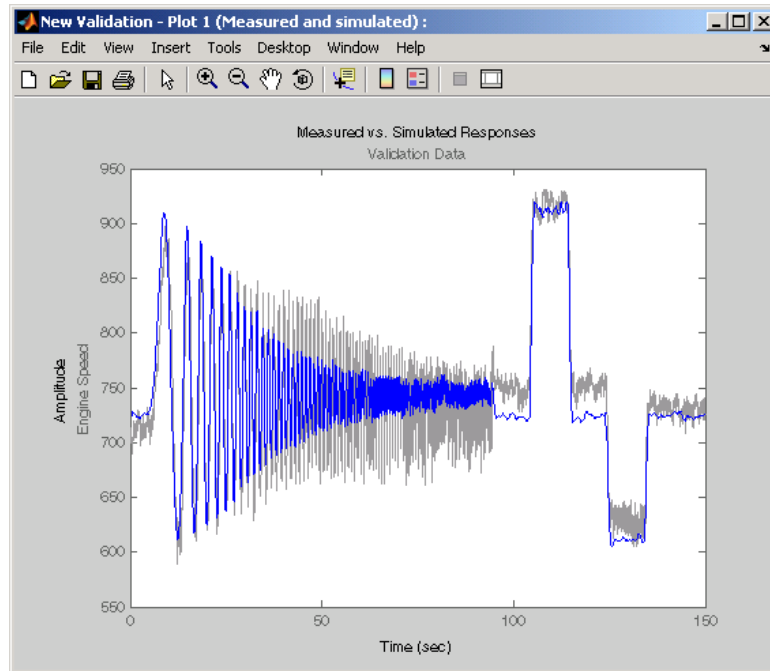


To perform the validation:

- 1 Select the **New Validation** node in the **Workspace** tree to open the **Validation Setup** pane.

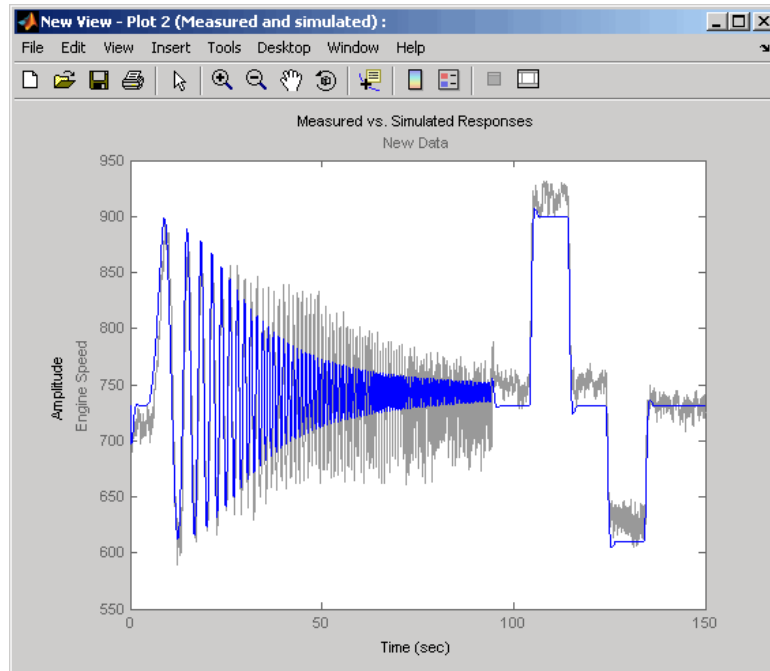


- 2** Click the **Plot Type** cell for Plot 1 and select Measured and simulated from the drop-down menu.
- 3** In the **Options** area, select Validation Data in the **Validation data set** drop-down list.
- 4** Click **Show Plots** to open a plot figure window as shown next.



Measured Versus Simulated Data Plot for Validation Data

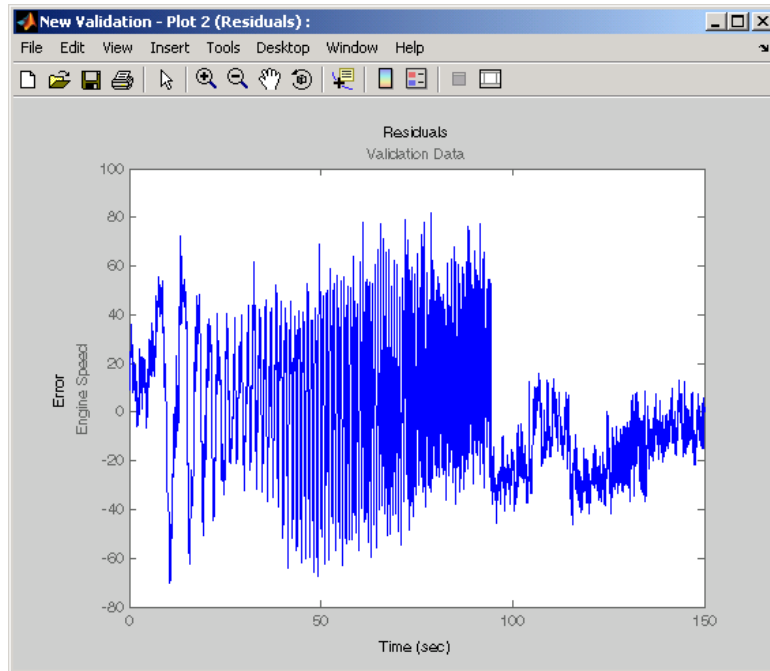
- 5 Compare this plot with the plot of Measured and simulated data for the validation data. For more information on how to create this plot, see “Selecting Views for Plotting” on page 2-19.



Measured and Simulated Data Views Plot

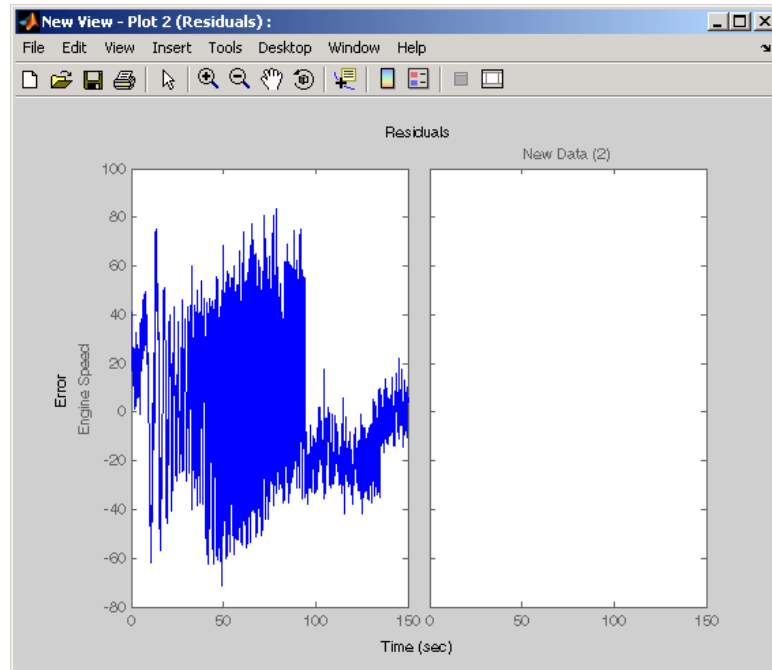
Comparing Residuals

To look at the residuals, select **Residuals** as the **Plot Type** for **Plot 2** in the **New Validation** pane. In the **Options** area, select the **Plot 2** check box and click **Show Plots**. The following figure shows the resulting residuals plot.



Plot of Residuals Using the Validation Data

Compare the validation data residuals with the original data set residuals from the **Views** node in the **Workspace** tree. To create the plot of residuals for the original data set, select the **New View** node and choose **Residuals** as the **Plot Type**.



Plot of Residuals Using the Test Data

The plot on the left agrees with the plot of the residuals for the validation data. The right side has no plot because residuals were not calculated for the validation data during the original estimation process.

Accelerating Model Simulations During Estimation

In this section...

“About Accelerating Model Simulations During Estimation” on page 2-50

“Limitations” on page 2-50

“Setting the Accelerator Mode for Parameter Estimation” on page 2-50

About Accelerating Model Simulations During Estimation

You can accelerate the parameter estimation computations by changing the simulation mode of your Simulink model. Simulink Design Optimization software supports `Normal` and `Accelerator` simulation modes. For more information about these modes, see “Accelerating Models” in the Simulink documentation.

The default simulation mode is `Normal`. In this mode, Simulink software uses interpreted code, rather than compiled C code during simulations.

In the `Accelerator` mode, Simulink Design Optimization software runs simulations during estimation with compiled C code. Using compiled C code speeds up the simulations and reduces the time to estimate parameters.

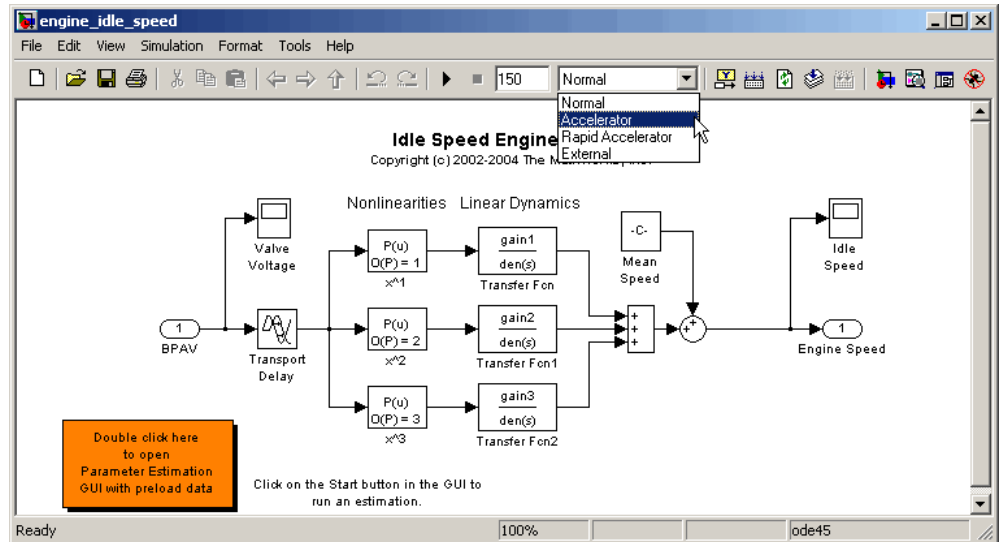
Limitations

You cannot use the `Accelerator` mode if your model contains algebraic loops. If the model contains MATLAB function blocks, you must either remove them or replace them with Fcn blocks.

Setting the Accelerator Mode for Parameter Estimation

To set the simulation mode to `Accelerator`, open the Simulink model window and perform one of the following actions:

- Select **Simulation > Accelerator**.
- Choose `Accelerator` from the drop-down list as shown in the next figure.



Tip To obtain the maximum performance from the Accelerator mode, close all Scope blocks in your model.

Speeding Up Parameter Estimation Using Parallel Computing

In this section...

“When to Use Parallel Computing for Estimating Model Parameters” on page 2-52

“How Parallel Computing Speeds Up Parameter Estimation” on page 2-53

“Specifying Model Dependencies” on page 2-56

“Configuring Your System for Parallel Computing” on page 2-56

“How to Use Parallel Computing in the GUI” on page 2-58

“Troubleshooting” on page 2-62

When to Use Parallel Computing for Estimating Model Parameters

You can use Simulink Design Optimization software with Parallel Computing Toolbox™ software to speed up parameter estimation of Simulink models. Using parallel computing may reduce the estimation time in the following cases:

- The model contains a large number parameters to estimate, and the Nonlinear least squares or Gradient descent is selected as the estimation method.
- The Pattern search method is selected as the estimation method.
- The model is complex and takes a long time to simulate.

When you use parallel computing, Simulink Design Optimization software distributes independent simulations to run them in parallel on multiple MATLAB sessions, also known as *workers*. The time required to simulate the model dominates the total estimation time. Therefore, distributing the simulations significantly reduces the estimation time. For more information on the expected speedup, see “How Parallel Computing Speeds Up Parameter Estimation” on page 2-53.

The following sections describe how to configure your system, and use parallel computing:

- “Configuring Your System for Parallel Computing” on page 2-56
- “How to Use Parallel Computing in the GUI” on page 2-58
- “How to Use Parallel Computing at the Command Line” on page 2-116

How Parallel Computing Speeds Up Parameter Estimation

You can enable parallel computing with the `Nonlinear least squares`, `Gradient descent` and `Pattern search` estimation methods in the Simulink Design Optimization software. The following sections describe how parallel computing speeds up the estimation:

- “Parallel Computing with Nonlinear least squares and Gradient descent Methods” on page 2-53
- “Parallel Computing with the Pattern search Method” on page 2-54

Parallel Computing with Nonlinear least squares and Gradient descent Methods

When you select `Gradient descent` as the estimation method, the model is simulated during the following computations:

- Objective value computation — One simulation per iteration
- Objective gradient computations — Two simulations for every tuned parameter per iteration
- Line search computations — Multiple simulations per iteration

The total time, T_{total} , taken per iteration to perform these simulations is given by the following equation:

$$T_{total} = T + (N_p \times (2 \times T)) + (N_{ls} \times T) = T \times (1 + (2 \times N_p) + N_{ls})$$

where T is the time taken to simulate the model and is assumed to be equal for all simulations, N_p is the number of parameters to estimate, and N_{ls} is the number of line searches.

When you use parallel computing, Simulink Design Optimization software distributes the simulations required for objective gradient computations. The simulation time taken per iteration when the gradient computations are performed in parallel, T_{totalP} , is approximately given by the following equation:

$$T_{totalP} = T + \left(\text{ceil} \left(\frac{N_p}{N_w} \right) \times 2 \times T \right) + (N_{ls} \times T) = T \times \left(1 + 2 \times \text{ceil} \left(\frac{N_p}{N_w} \right) + N_{ls} \right)$$

where N_w is the number of MATLAB workers.

Note The equation does not include the time overheads associated with configuring the system for parallel computing and loading Simulink software on the remote MATLAB workers.

The expected reduction of the total estimation time is given by the following equation:

$$\frac{T_{totalP}}{T_{total}} = \frac{1 + 2 \times \text{ceil} \left(\frac{N_p}{N_w} \right) + N_{ls}}{1 + (2 \times N_p) + N_{ls}}$$

For example, for a model with $N_p=3$, $N_w=4$, and $N_{ls}=3$, the expected reduction of

the total estimation time equals $\frac{1 + 2 \times \text{ceil} \left(\frac{3}{4} \right) + 3}{1 + (2 \times 3) + 3} = 0.6$.

Parallel Computing with the Pattern search Method

The Pattern search method uses search and poll sets to create and compute a set of candidate solutions at each estimation iteration.

The total time, T_{total} , taken per iteration to perform these simulations, is given by the following equation:

$$T_{total} = (T \times N_p \times N_{ss}) + (T \times N_p \times N_{ps}) = T \times N_p \times (N_{ss} + N_{ps})$$

where T is the time taken to simulate the model and is assumed to be equal for all simulations, N_p is the number of parameters to estimate, N_{ss} is a factor for the search set size, and N_{ps} is a factor for the poll set size.

When you use parallel computing, Simulink Design Optimization software distributes the simulations required for the search and poll set computations, which are evaluated in separate parfor loops. The simulation time taken per iteration when the search and poll sets are computed in parallel, T_{totalP} , is given by the following equation:

$$\begin{aligned} T_{totalP} &= (T \times \text{ceil}(N_p \times \frac{N_{ss}}{N_w})) + (T \times \text{ceil}(N_p \times \frac{N_{ps}}{N_w})) \\ &= T \times (\text{ceil}(N_p \times \frac{N_{ss}}{N_w}) + \text{ceil}(N_p \times \frac{N_{ps}}{N_w})) \end{aligned}$$

where N_w is the number of MATLAB workers.

Note The equation does not include the time overheads associated with configuring the system for parallel computing and loading Simulink software on the remote MATLAB workers.

The expected speed up for the total estimation time is given by the following equation:

$$\frac{T_{totalP}}{T_{total}} = \frac{\text{ceil}(N_p \times \frac{N_{ss}}{N_w}) + \text{ceil}(N_p \times \frac{N_{ps}}{N_w})}{N_p \times (N_{ss} + N_{ps})}$$

For example, for a model with $N_p=3$, $N_w=4$, $N_{ss}=15$, and $N_{ps}=2$, the expected

$$\text{speedup equals } \frac{\text{ceil}(3 \times \frac{15}{4}) + \text{ceil}(3 \times \frac{2}{4})}{3 \times (15 + 2)} = 0.27 .$$

Using the `Pattern search` method with parallel computing may not speed up the estimation time. When you do not use parallel computing, the method stops searching for a candidate solution at each iteration as soon as it finds a solution better than the current solution. When you use parallel computing, the candidate solution search is more comprehensive. Although the number of iterations may be larger, the estimation without using parallel computing may be faster.

Specifying Model Dependencies

Model dependencies are files, such as referenced models, data files and S-functions, without which a model cannot run. When you use parallel computing, Simulink Design Optimization software helps you identify model path dependencies. To do so, the software uses the Simulink Manifest Tools. The dependency analysis may not find all the files required by your model. For example, folder paths that contain code for your model or block callback. To learn more, see the “Scope of Dependency Analysis” in the Simulink documentation.

If your model has dependencies that the software cannot detect automatically, you must add the dependencies before you start the estimation using parallel computing:

- 1 Add the path dependencies, as described “How to Use Parallel Computing in the GUI” on page 2-58 and “How to Use Parallel Computing at the Command Line” on page 2-116.
- 2 Add the file dependencies, as described in “Configuring Parallel Computing on Multiprocessor Networks” on page 2-57.

Note When you use parallel computing, verify that the remote MATLAB workers can access all the model dependencies. The optimization errors out if all the remote workers cannot access all the model dependencies.

Configuring Your System for Parallel Computing

You can use parallel computing on multi-core processors or multi-processor networks. To configure your system for parallel computing, see the following sections:

- “Configuring Parallel Computing on Multicore Processors” on page 2-57
- “Configuring Parallel Computing on Multiprocessor Networks” on page 2-57

After you configure your system for parallel computing, you can use the GUI or the command-line functions to estimate the model parameters.

Configuring Parallel Computing on Multicore Processors

With a basic Parallel Computing Toolbox license, you can establish a pool of up to four parallel MATLAB sessions in addition to the MATLAB client.

To start a pool of four MATLAB sessions in local configuration, type the following at the MATLAB prompt:

```
matlabpool open local
```

To learn more, see the `matlabpool` reference page in the Parallel Computing Toolbox documentation.

Configuring Parallel Computing on Multiprocessor Networks

To use parallel computing on a multiprocessor network, you must have the Parallel Computing Toolbox software and the MATLAB® Distributed Computing Server™ software. To learn more, see the Parallel Computing Toolbox and MATLAB Distributed Computing Server documentation.

To configure a multiprocessor network for parallel computing:

- 1** Create a user configuration file to include any model file dependencies, as described in “Defining Configurations” and FileDependencies reference page in the Parallel Computing Toolbox documentation.
- 2** Open the pool of MATLAB workers using the user configuration file, as described in “Applying Configurations in Client Code” in the Parallel Computing Toolbox documentation.

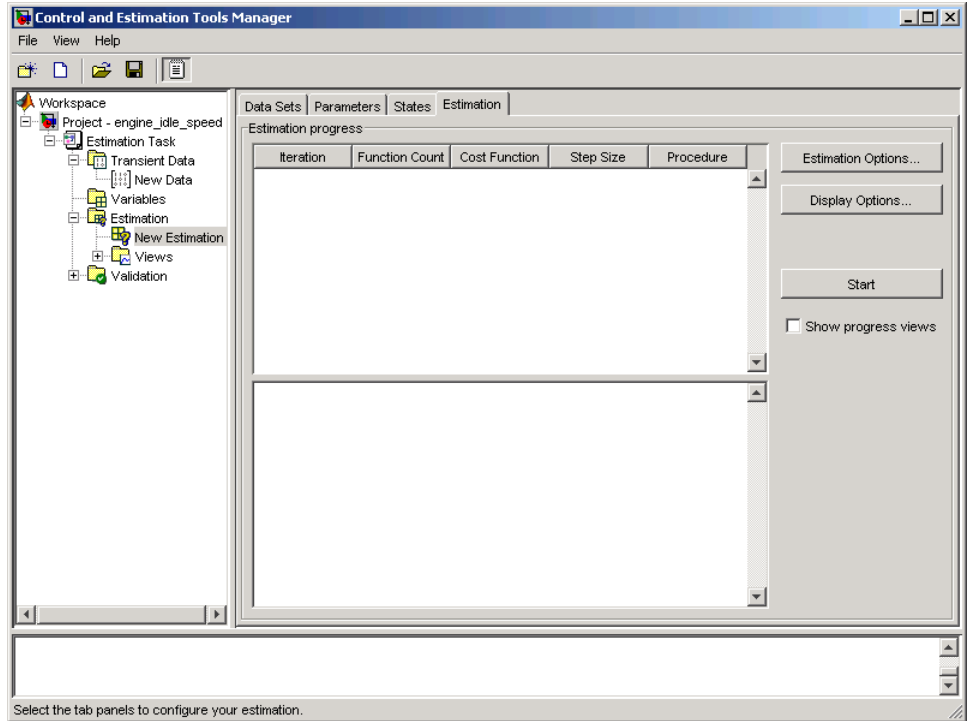
Opening the pool allows the remote workers to access the file dependencies included in the user configuration file.

How to Use Parallel Computing in the GUI

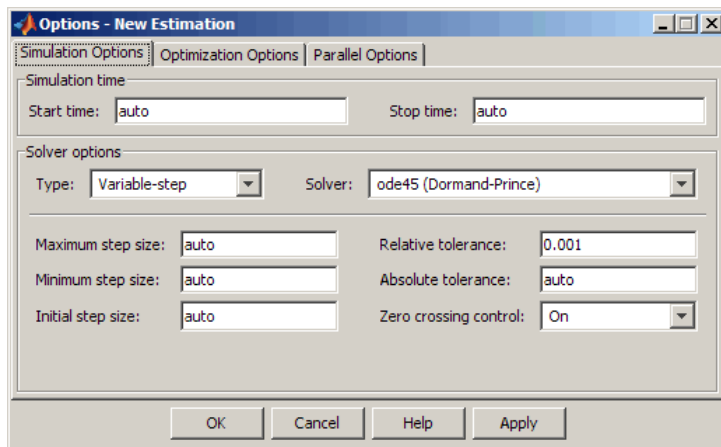
After you configure your system for parallel computing, as described in “Configuring Your System for Parallel Computing” on page 2-56, you can use the GUI to estimate the model parameters.

Tip If you want to use functions to estimate parameters using parallel computing, see “How to Use Parallel Computing at the Command Line” on page 2-116.

- 1** Open the Simulink model by typing the model name at the MATLAB prompt.
- 2** Configure the model for parameter estimation, as described in “Configuring Parameter Estimation in the GUI” on page 2-3.
- 3** In the **Estimation** tab of the **New Estimation** node, click **Estimation Options**.



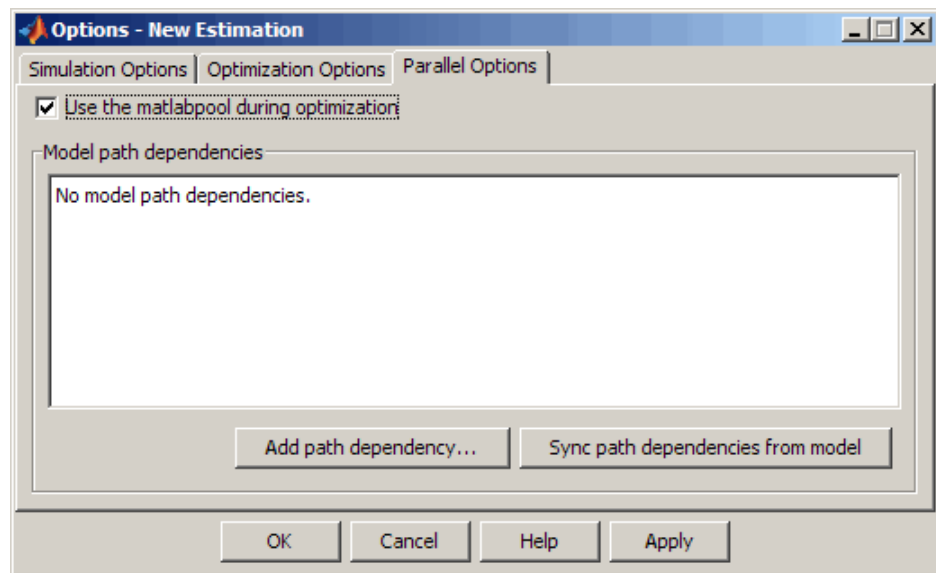
This action opens the Options - New Estimation dialog box.



- 4 In the **Parallel Options** tab, select the **Use the matlabpool during optimization** option.

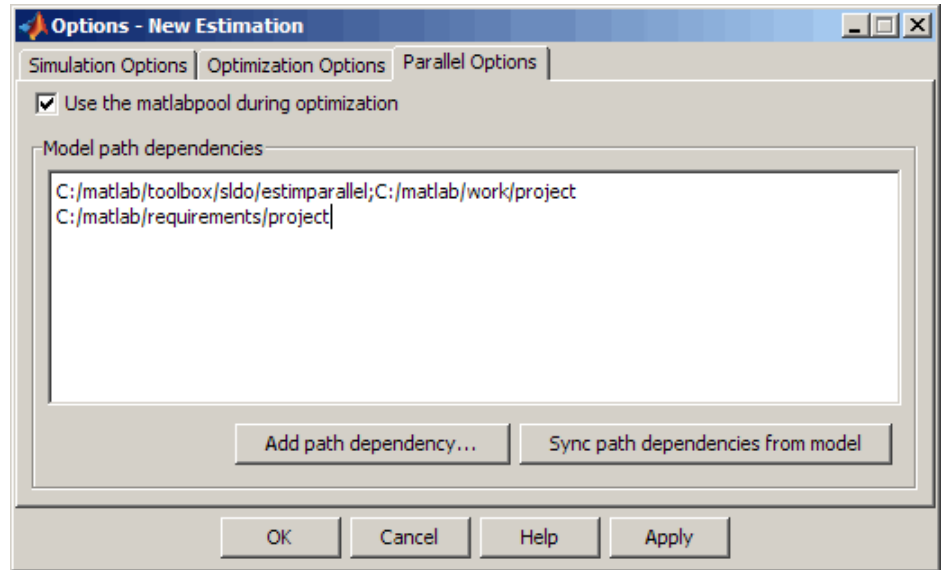
This action checks for model path dependencies in your Simulink model and displays the path dependencies in the **Model path dependencies** list box.

Note As described in “Specifying Model Dependencies” on page 2-56, the automatic path dependencies check may not detect all the path dependencies in your model.



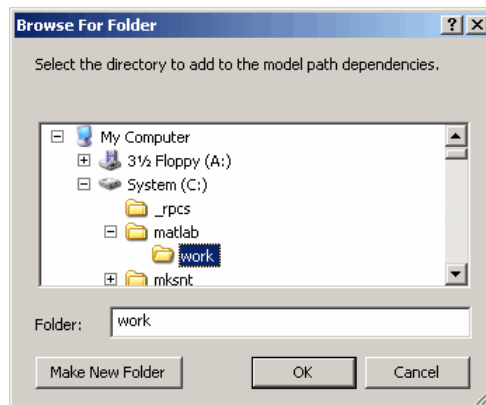
- 5 (Optional) Add the path dependencies that the automatic check does not detect.
 - a Specify the paths in the **Model path dependencies** list box.

You can specify the paths separated with a semicolon, or on a new line.



b Click **Apply** to include the new paths.

Alternatively, you can click **Add path dependency** to open a Browse For Folder dialog box where you can select the folder to add.



6 (Optional) If you modify the Simulink model such that it introduces a new path dependency, then you must resync the path dependencies. Click **Sync**

path dependencies from model in the **Parallel Options** tab to rerun the automatic dependency check for your model.

This action updates the **Model path dependencies** list box with any new path dependency found in the model.

7 Click **OK**.

8 In the **Estimation** tab, click **Start** to estimate the model parameters using parallel computing.

9 Examine the values of the estimated parameters in the **Value** column of the **Parameters** tab.

For more information on how to troubleshoot estimation results you obtained using parallel computing, see “Troubleshooting” on page 2-62.

Troubleshooting

- “Why are the estimation results with and without using parallel computing different?” on page 2-62
- “Why do I not see the estimation speedup I expected using parallel computing?” on page 2-63
- “Why does the estimation using parallel computing not make any progress?” on page 2-64
- “Why do I receive an error "Cannot save model tpe5468c55_910c_4275_94ef_305e2eeeeef4"?” on page 2-64
- “Why does the estimation using parallel computing not stop when I click **Stop**?” on page 2-64

Why are the estimation results with and without using parallel computing different?

The values of the estimated parameters obtained using parallel computing may differ from the values obtained without using parallel computing. The results can be different under the following conditions:

- Different numerical precision on the client and worker machines can produce marginally different simulation results. Thus, the estimation

method may take a completely different solution path and produce a different result.

Note Numerical precision can differ because of different operating systems or hardware on the client and worker machines.

- The state of the model on the client and the worker machines can differ, and thus lead to a different result. For example, the state can become different if you change a parameter value initialized by a callback function on the client machine after the workers have loaded the model. The model parameter values on the workers and the client are now out of sync, which can lead to a different result.

After you change the model parameter values initialized by a callback function, verify that the parameters exist in the model workspace or update the callback function so that the remote workers have access to the changed parameter values.

- When you use parallel computing with the `Pattern search` method, the method searches for a candidate solution more comprehensively than when you do not use parallel computing. This more comprehensive search can result in a different solution. To learn more, see “Parallel Computing with the `Pattern search Method`” on page 2-54.

Why do I not see the estimation speedup I expected using parallel computing?

- The resulting estimation time may not be faster when you estimate a small number of model parameters or when the model does not take long to simulate. In such cases, the overheads associated with creating and distributing the parallel tasks outweighs the benefits of running the simulations during estimation in parallel.
- Using `Pattern search` method with parallel computing may not speed up the estimation time. When you do not use parallel computing, the method stops searching for a candidate solution at each iteration as soon as it finds a solution better than the current solution. The candidate solution search is more comprehensive when you use parallel computing. Although the

number of iterations may be larger, the optimization without using parallel computing is faster.

To learn more about the expected speedup, see “Parallel Computing with the Pattern search Method” on page 2-54.

Why does the estimation using parallel computing not make any progress?

In some cases, the gradient computations on the remote worker machines may silently error out when you use parallel computing. In such cases, the **Estimation progress** table shows that the $f(x)$ values do not change, and the optimization terminates after two iterations.

To troubleshoot the problem:

- 1** Run the optimization for a few iterations without parallel computing to see if the optimization progresses.
- 2** Check if the remote workers have access to all model dependencies. To learn more, see “Specifying Model Dependencies” on page 2-56.

Why do I receive an error "Cannot save model tpe5468c55_910c_4275_94ef_305e2eeeeef4"?

When you select **Refined** as the **Gradient type**, the software may error out when it saves a temporary model to a nonwriteable folder, and then displays this error message. Change the **Gradient type** to **Basic** to clear this error. To learn more, see “Selecting Additional Optimization Options” on page 2-27.

Why does the estimation using parallel computing not stop when I click Stop?

When you use parallel computing, the software has to wait till the current iteration completes before it notifies the workers to stop the estimation. The estimation does not terminate immediately when you click **Stop**, and appears to continue to run.

Estimating Initial States

In this section...

“How to Estimate Initial States in the GUI” on page 2-65

“Estimating Initial Conditions for Blocks with External Initial Conditions” on page 2-66

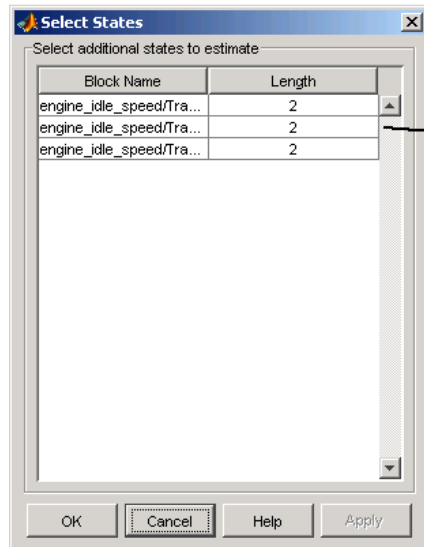
“Example — Estimating Initial States of a Mass-Spring-Damper System” on page 2-67

How to Estimate Initial States in the GUI

In general, you choose to estimate only those states that are not already in the model. Before you estimate initial conditions, you must have already imported the estimation data and specified the parameters to estimate, as described in “Specifying Data for Parameter Estimation” on page 2-4 and “Specifying Parameters to Estimate” on page 2-6, respectively.

To estimate initial conditions (or initial states) if they are not known:

- 1** In the Control and Estimation Tools Manager, select the **Variables** node in the **Workspace** tree.
- 2** Click the **Estimated States** tab.
- 3** Click **Add** to open the Select States dialog box.



By default, the Select States dialog box looks at all states of all blocks in the model.

List of states

Use your mouse to select data. To select adjacent states, hold down the **Shift** key while clicking the first and last state in the selection. To select nonadjacent states, hold down the **Ctrl** key while clicking each state.

4 Select the states to estimate and click **OK**.

The states selected for estimation are added to the **Estimated States** tab. For an example of estimating initial states, see “Example — Estimating Initial States of a Mass-Spring-Damper System” on page 2-67.

Estimating Initial Conditions for Blocks with External Initial Conditions

When an integrator block uses an initial-condition port, which you specify by an IC block feeding into the integrator block, you cannot estimate the initial conditions (ICs) of the integrator using Simulink Design Optimization software. This is because external ICs have priority over the ICs of a specific block to maintain the integrity of the model.

To tune the ICs of an integrator block with external ICs, you must modify the model to make the external signal into a tunable parameter. For example, you can set the IC block that feeds into the integrator to be a tunable variable and estimate it.

Example — Estimating Initial States of a Mass-Spring-Damper System

- “Loading the Example” on page 2-67
- “Model Parameters” on page 2-68
- “Setting Up the Estimation Project” on page 2-69
- “Importing Transient Data and Selecting Parameters for Estimation” on page 2-70
- “Selecting Parameters and Initial Conditions for Estimation” on page 2-72
- “Creating the Estimation Task” on page 2-73
- “Running the Estimation and Viewing Results” on page 2-75

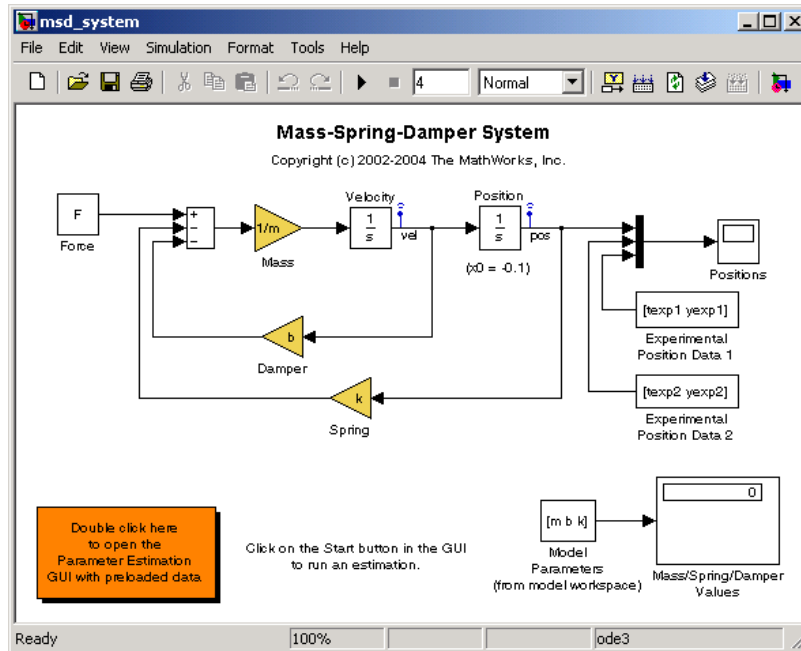
Loading the Example

To open the Simulink model of a mass-spring-damper system, type:

```
msd_system
```

at the MATLAB prompt.

This action also loads the two sets of measured data with differing initial conditions.




You can also run the Initial State Estimation demo that shows how to estimate parameters and initial states. This example goes beyond the demo by providing in-depth discussion of each task.

Model Parameters

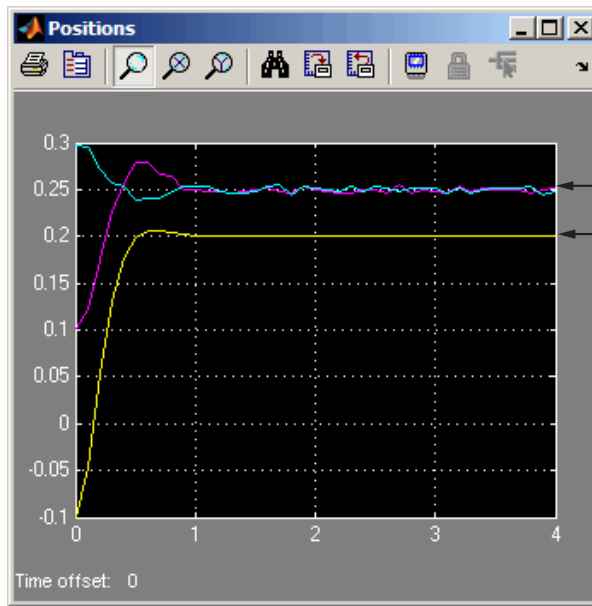
The output of the Simulink model, `msd_system`, is the position of the mass in a mass-spring-damper system. The model is subject to a constant force F , and an initial condition, x_0 , for the mass displacement. x_0 is the initial condition of the integrator block, Position.

The model parameters of interest are the mass, m , the viscous damping, b , and the spring constant, k . For more information about physical modeling of mass-spring-damper systems, see any book on mathematical modeling or on automatic control systems.

For estimating the model parameters m , b , and k , this model uses two experimental data sets. The data sets contain output data measured using two different initial positions, $x_0=0.1$ and $x_0=0.3$, and additive noise.

Click  to run a simulation. The simulation generates the following plots, as shown in the next figure:

- Simulated response of the model for $x_0 = -0.1$ and parameter values, $m=8$, $k=500$, and $b=100$
- Experimental data sets

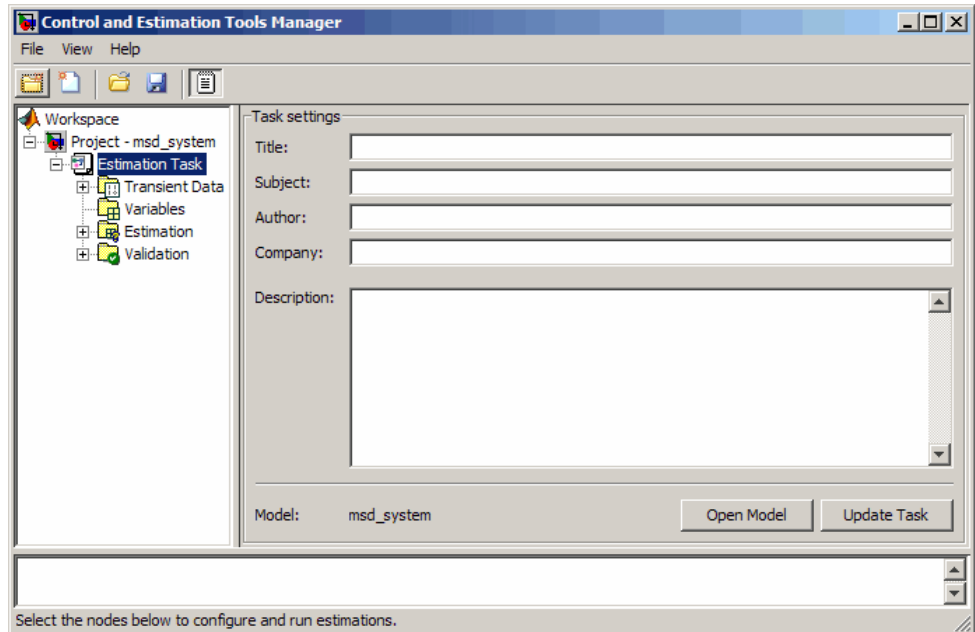


Magenta and cyan lines are experimental data sets with different initial conditions.

Yellow line is the response of the model to a constant force.

Setting Up the Estimation Project

To set up the estimation of initial conditions and then transient state space data, select **Tools > Parameter Estimation** in the `msd_system` model window.



Importing Transient Data and Selecting Parameters for Estimation

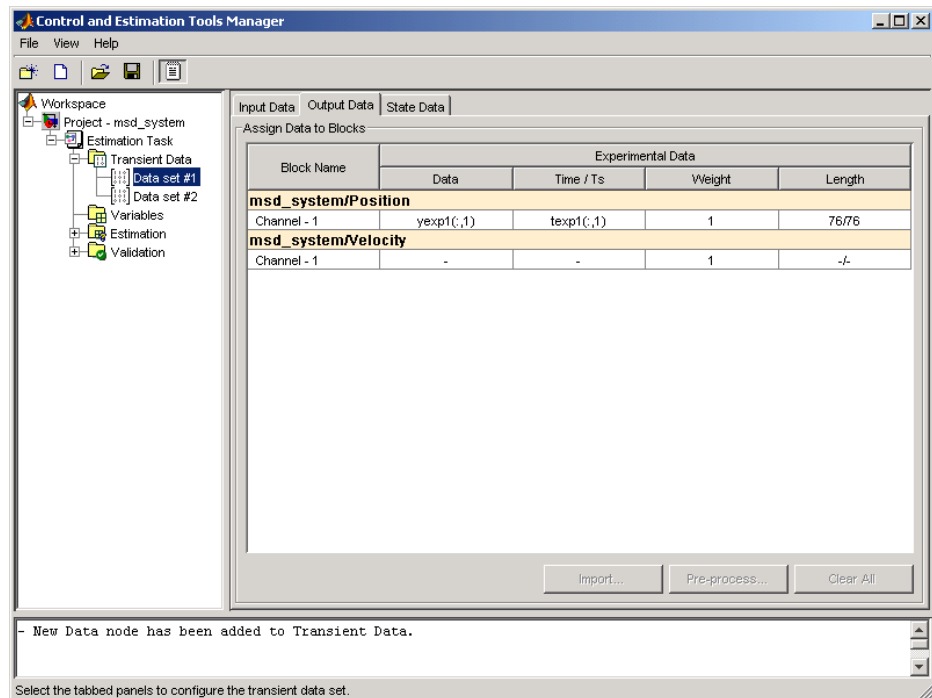
To import data for initial state estimation:

- 1 In the Control and Estimation Tools Manager, select **Transient Data** under the **Estimation Task** node.
- 2 Right-click **Transient Data**, and select **New** to create a **New Data** node.
- 3 Select **New Data** under the **Transient Data** node.
- 4 In the **Output Data** pane of the **New Data** node, select the **Data** column of the `msd_system/Position` state, and click **Import**. The Data Import dialog box opens.

Select `yexp1`, and click **Import** to assign the data `yexp1` to the model.

- 5 In the **Output Data** pane, select the **Time/Ts** column of the **msd_system/Position** state. In the Data Import dialog box, select **texp1**, and click **Import** to assign the time vector **texp1** to the model.
- 6 Right-click **New Data** in the **Workspace** tree, and rename it to **Data set #1**.
- 7 Repeat steps 2–5 to add a second data set, **yexp2** and **texp2**. Rename the data set to **Data set #2**.

The Control and Estimation Tools Manager GUI now resembles the next figure.



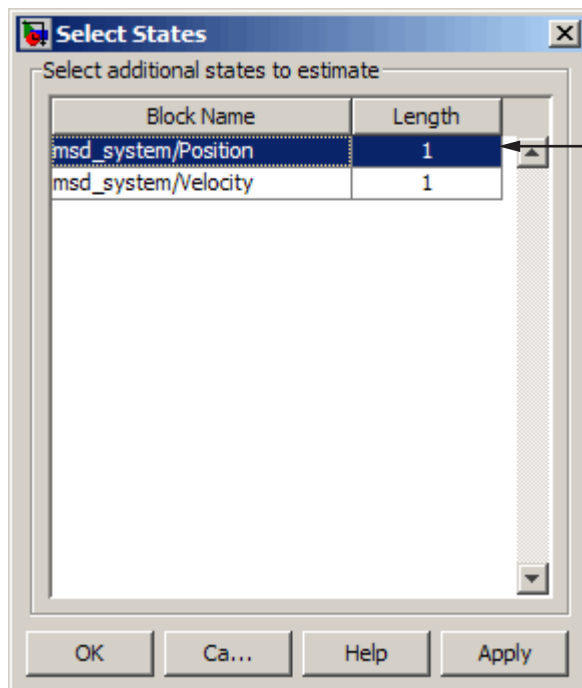
For more information on importing data, see “Importing Data into the GUI” on page 1-5.

Selecting Parameters and Initial Conditions for Estimation

To select the parameters and initial states you want to estimate for the Simulink model `msd_system`:

- 1 Select the **Variables** node in the **Workspace** tree of the Control and Estimation Tools Manager GUI.
- 2 In the **Estimated Parameters** pane, click **Add** to open the Select Parameters dialog box.
- 3 Select the parameters `b`, `k`, and `m`, and then click **OK**. The selected parameters now appear in the **Selected parameters** area of the **Estimated Parameters** pane.
- 4 In the **Estimated States** pane, click **Add**.

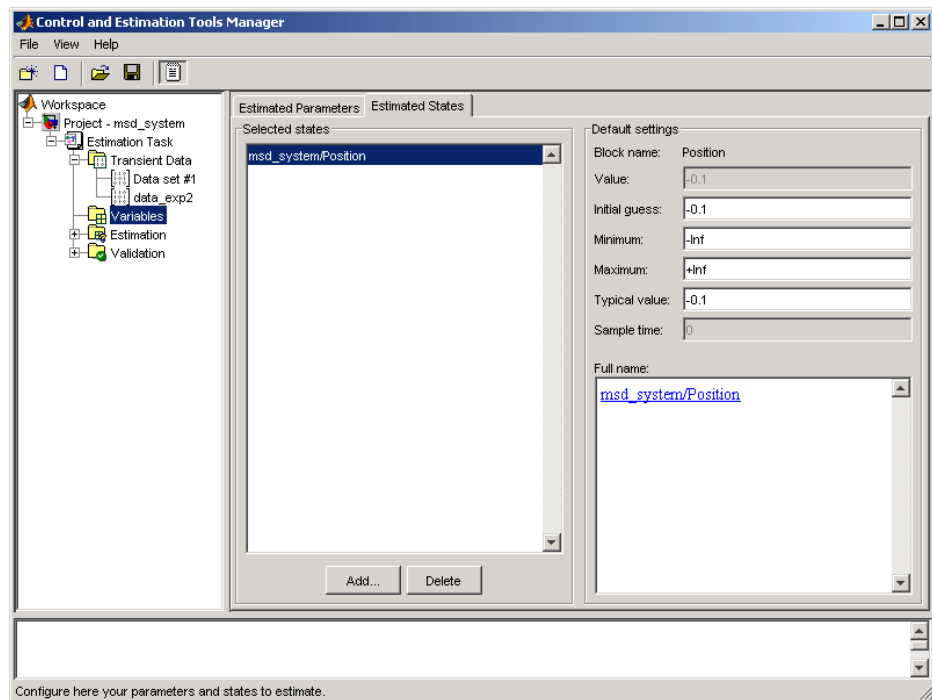
The Select States dialog box opens, which displays all the available states of the `msd_system` model. Select `msd_system/Position`, and click **OK**.



Select states with initial conditions that you want to estimate. Hold down Shift, and use your mouse to select groups of adjacent states. Hold down Ctrl, and use your mouse to select non-adjacent states.

Note For states that you do not estimate, the software uses the initial condition value specified in the Simulink model. In this example, the value of initial velocity, as specified in the model is 0.

The selected state appears in the **Selected states** area of the **Estimated States** pane, as shown in the next figure.



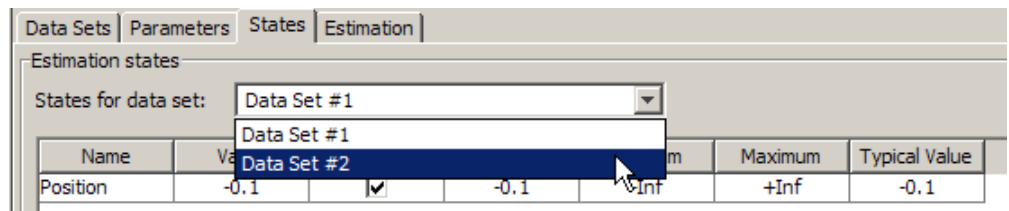
For more information on selecting parameters to estimate, see “Specifying Parameters to Estimate” on page 2-6.

Creating the Estimation Task

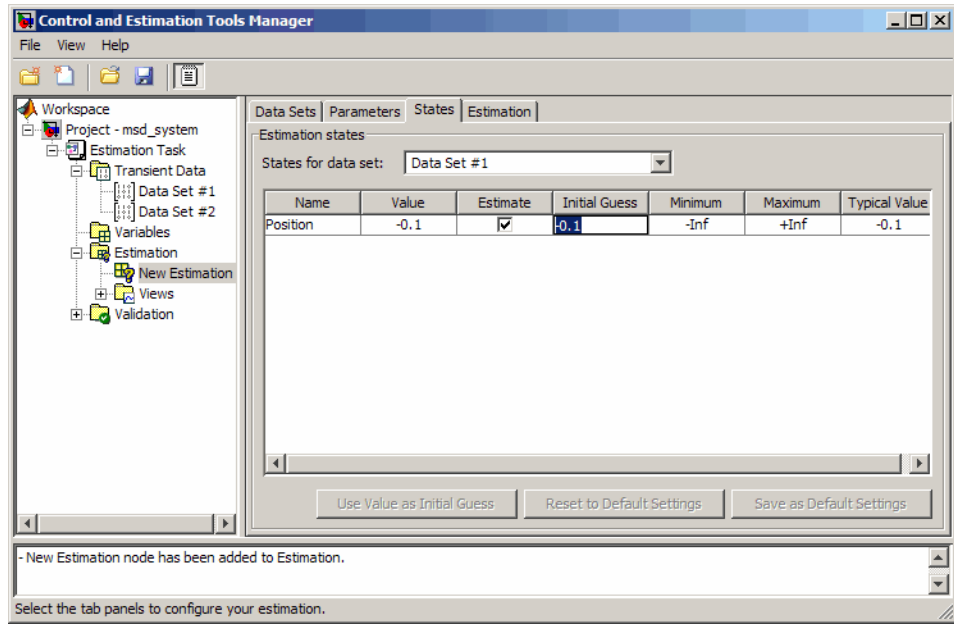
To create an estimation task in the Control and Estimation Tools Manager GUI, select the **Estimation** node in the **Workspace** tree, and click **New**. This action creates a **New Estimation** node.

In the **New Estimation** node, select the following check boxes:

- 1** Data Set #1 and Data Set #2 in the **Data Sets** pane.
- 2** **Estimate** for b , k , and m in the **Parameters** pane.
- 3** **Estimate** for Position in the **States** pane. Make sure to select this check box for both Data Set #1 and Data Set #2 to estimate the initial position for the spring.



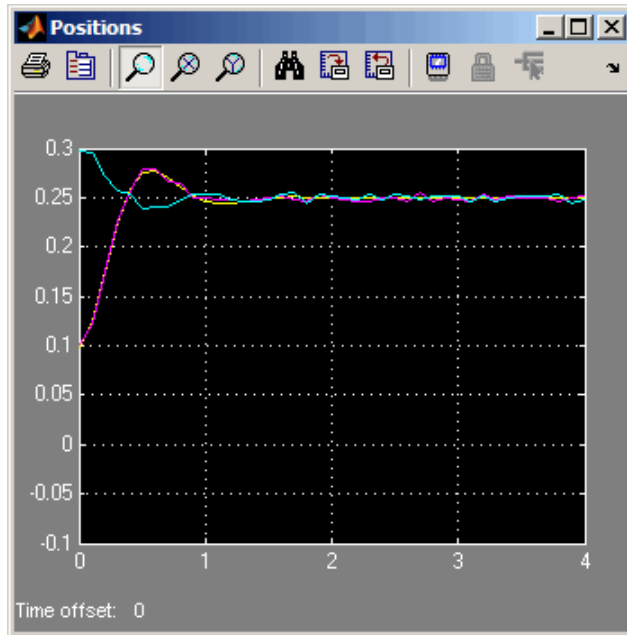
Although the initial positions for the two data sets differ, specify the initial state guesses for both data sets as -0.1 . The Control and Estimation Tools Manager GUI now resembles the following figure.



Running the Estimation and Viewing Results

Click **Start** in the **Estimation** pane to run the estimation.

As the estimation proceeds, the most current estimation of the position response (yellow curve) updates in the Scope. The curve toggles between the two experimental data sets because the software uses the two data sets successively to update the estimates of the parameter values. The software converges to the correct parameter values, within the scope of experimental noise and optimization options settings. The closeness of the estimated response (yellow) to the experimental data (magenta) indicates that simulated data is a good match to the measured data. .



View the initial position estimates for Data Set #1 and Data Set #2 in the **Value** column of the **States** tab. The estimated values match closely with the known values, 0.1 and 0.3 of initial position.

Estimation states

States for data set: Data Set #1

Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Value
Position	0.29945	<input checked="" type="checkbox"/>	-0.1	-Inf	+Inf	-0.1

View the estimated parameter values in the **Value** column of the **Parameters** tab.

Estimation parameters						
Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Value
b	57.967	<input checked="" type="checkbox"/>	b	-Inf	+Inf	b
k	400.27	<input checked="" type="checkbox"/>	k	-Inf	+Inf	k
m	9.7662	<input checked="" type="checkbox"/>	m	-Inf	+Inf	m

The estimation of initial states is important for obtaining the correct estimates of the model parameters. You do not set the initial states (x_0 in this case) as parameters because the initial states do not represent fixed physical properties of the system. For different experimental data or operating conditions, these states need not be unique.

In this example, you use two data sets with distinct initial positions together for a single estimation of model parameters. While the estimates of the model parameters are unique, the initial state (position) is different, and you estimate them individually for each data set.

Working with Estimation Projects

In this section...
“Structure of an Estimation Project” on page 2-78
“Managing Multiple Projects and Tasks” on page 2-79
“Adding, Deleting and Renaming an Estimation Project” on page 2-80
“Saving Control and Estimation Tools Manager Projects” on page 2-81
“Loading Control and Estimation Tools Manager Projects” on page 2-82

Structure of an Estimation Project

The Control and Estimation Tools Manager, which is a graphical user interface (GUI) for performing parameter estimation, stores and organizes all data from a given Simulink model inside a *project*. To open the Control and Estimation Tools Manager GUI, select **Tools > Parameter Estimation** in the Simulink model window.

When using the Control and Estimation Tools Manager for parameter estimation, you can

- Manage estimation projects.
- Select parameters and initial conditions to configure the estimation.
- Specify cost functions.
- Import experimental data (to be matched by the output of your Simulink model).
- Specify the initial conditions of your model.

Each estimation task can include

- One or more data sets
- Parameter information
- One or more sets of estimation settings, or configurations

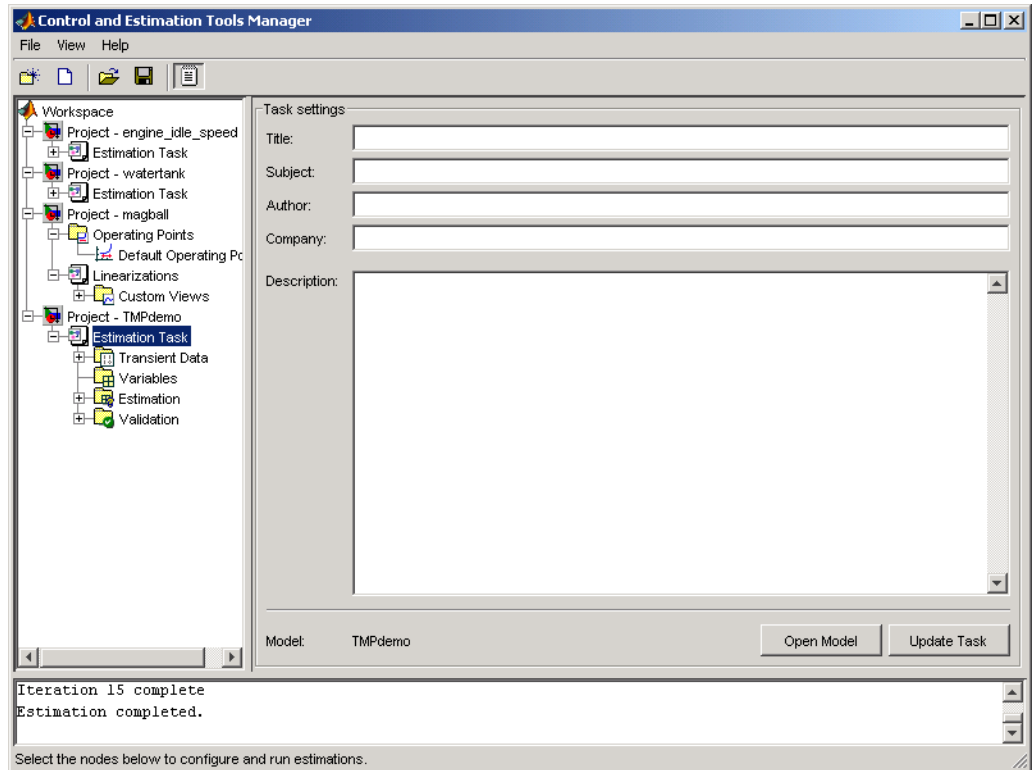
The default project name is the same as the Simulink model name. The project name is shown in the **Workspace** tree of the Control and Estimation Tools Manager.

You can also add tasks from Simulink® Control Design™ and Model Predictive Control Toolbox™ software to the current project, if these products are installed on your system.

Managing Multiple Projects and Tasks

The Control and Estimation Tools Manager works seamlessly with products in the Controls and Estimation family. In particular, if you have licenses for Simulink Control Design or Model Predictive Control Toolbox software, you can use these products to perform tasks on projects that you have created in Simulink Design Optimization software, and vice versa.

This figure shows a tools manager with multiple projects and multiple tasks.



You can save projects individually, or group multiple projects together in one saved file, as described in:

- “Saving Control and Estimation Tools Manager Projects” on page 2-81
- “Loading Control and Estimation Tools Manager Projects” on page 2-82

Adding, Deleting and Renaming an Estimation Project

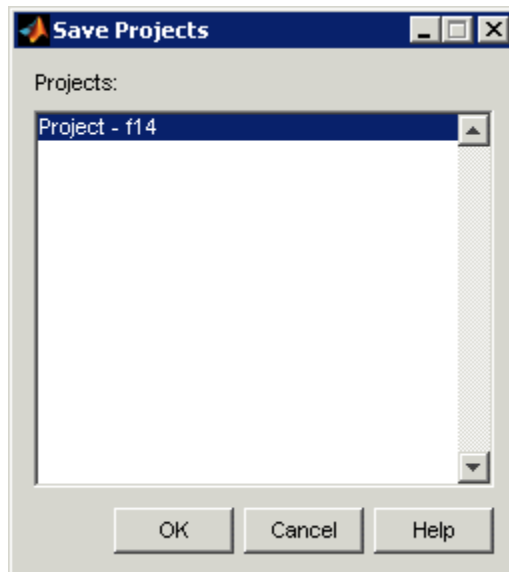
To add, delete, or rename the project or task:

- 1** Right-click the project or task node in the **Workspace** tree.
- 2** Select the appropriate command from the shortcut menu.

Saving Control and Estimation Tools Manager Projects

A Control and Estimation Tools Manager project can consist of tasks from products such as Simulink Control Design, Simulink Design Optimization, and Model Predictive Control Toolbox software. Each task contains data, objects, and results for the analysis of a particular model.

To save your project as a MAT-file, select **File** > **Save** in the Control and Estimation Tools Manager window.

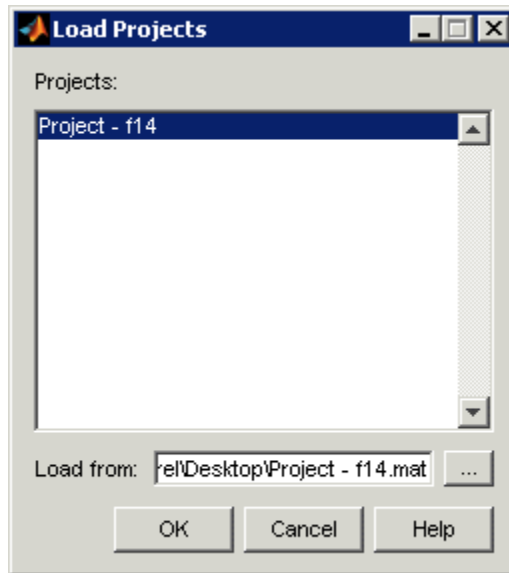


To save multiple projects within one file:

- 1** In the Save Projects dialog box, select the projects that you want to save.
- 2** Click **OK**.
- 3** Choose a folder and name for your project file by either browsing for a file or typing the full path and filename in the **Save as** field. Click **Save**.

Loading Control and Estimation Tools Manager Projects

To open previously saved projects, select **File** > **Load** in the Control and Estimation Tools Manager window.



In the Load Projects dialog box, choose a project file by either browsing for the folder and file, or by typing the full path and filename in the **Load from** field. Project files are always MAT-files. The projects within this file appear in the **Projects** list.

Select the projects that you want to load, then click **OK**. When a file contains multiple projects, you can choose to load them all or just a few.

Estimating Parameters at the Command Line

In this section...

“Workflow for Estimating Parameters at the Command Line” on page 2-83

“Example — Estimating Parameters and Initial States at the Command Line” on page 2-84

“Objects for Parameter Estimation” on page 2-95

“How to Use Parallel Computing at the Command Line” on page 2-116

Workflow for Estimating Parameters at the Command Line

In addition to the Control and Estimation Tools Manager GUI, you can also use Simulink Design Optimization functions to perform parameter and state estimation. These functions perform the same tasks as the tools manager, but have the advantages of command-line execution. When you perform a state or parameter estimation using the GUI, you create MATLAB objects for all the states and parameters of your model. If you have a large number of states or parameters, this can use up large amounts of memory and cause computational delays. With the command-line approach, only those states and parameters that you select are assigned MATLAB objects, which is more efficient.

In addition, the command-line approach is useful for batch jobs where you can estimate parameters for a large numbers of models.

Simulink Design Optimization software uses MATLAB objects to perform estimation tasks. To learn more about object-oriented programming, see the *Object-Oriented Programming* documentation for a description of object-oriented programming in MATLAB.

Simulink Design Optimization command-line interface requires a Simulink model as a starting point for analysis and estimation.

Note The Simulink model must contain an Inport or Outport block or logged signals to enable assigning data to the signals. For more information on how to configure a Simulink model for parameter estimation, see “Configuring a Model for Importing Data” on page 1-2.

After you configure your model for parameter estimation, as described in “Configuring a Model for Importing Data” on page 1-2, the estimation process at the command line consists of the following steps:

- 1 Defining experiments consisting of empirical data sets, and the operating conditions and/or initial conditions of your model.
- 2 Selecting the variables and states to be estimated.
- 3 Performing the estimation.
- 4 Reviewing the results and iterating as necessary.
- 5 Validating estimation results.

See “Objects for Parameter Estimation” on page 2-95

for more information on how to use methods and properties to perform parameter estimation.

Note The Simulink model must remain open to perform parameter estimation tasks.

For an example of estimating parameters using command-line functions, see “Example — Estimating Parameters and Initial States at the Command Line” on page 2-84.

Example — Estimating Parameters and Initial States at the Command Line

- “Loading the F14 Jet Model” on page 2-85

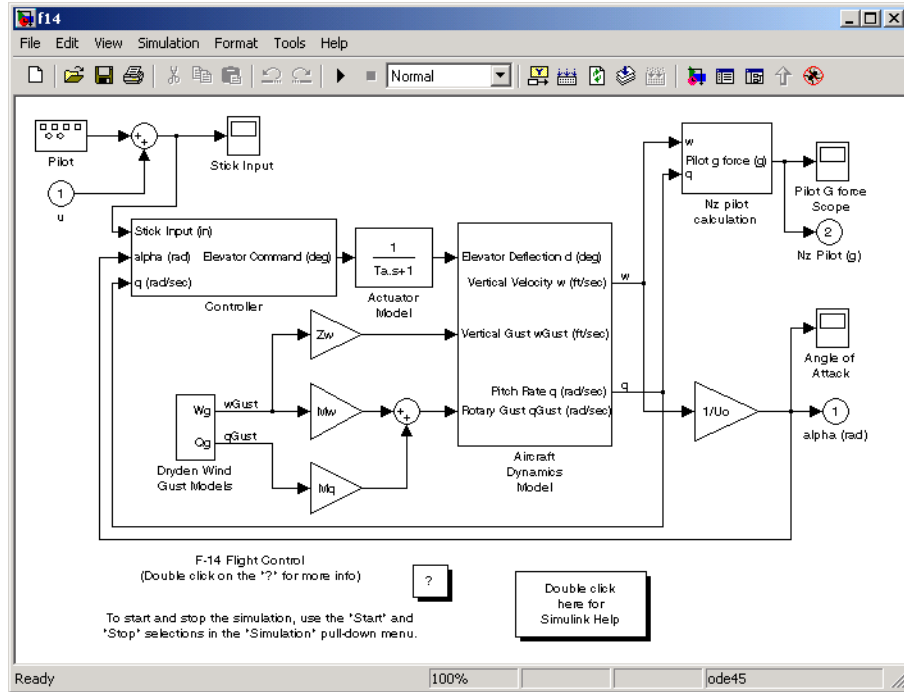
- “Baseline Simulation” on page 2-86
- “Creating a Transient Experiment Object” on page 2-88
- “Assigning Experimental Data to Inputs and Outputs of the Model” on page 2-89
- “Creating Parameter Objects for Estimation” on page 2-90
- “Creating an Estimation Object and Running the Estimation” on page 2-92

Loading the F14 Jet Model

To define an experiment, you must start with a Simulink model. For this example, type

```
f14
```

to load the F14 fighter jet model into the MATLAB workspace. The following figure shows the f14 model.



F14 Fighter Jet Model

This example outlines the basics of constructing an estimation project using object-oriented code. Only what you need to run the example is presented in this section. See “Objects for Parameter Estimation” on page 2-95 for details on all the properties and methods associated with parameter estimation.

Baseline Simulation

Before running an estimation, you need a baseline for data comparison. First, you must choose parameters and states’ initial conditions for estimation. This example uses T_a , the actuator time constant, and Z_d and M_d , the vertical velocity and pitch rate gains, respectively. Then use the following code to run the Simulink f14 model. Note that this is standard Simulink code and does not involve Simulink Design Optimization command-line interface in any way. See `sim` in the Simulink Reference documentation for information about running Simulink models from the MATLAB command line.

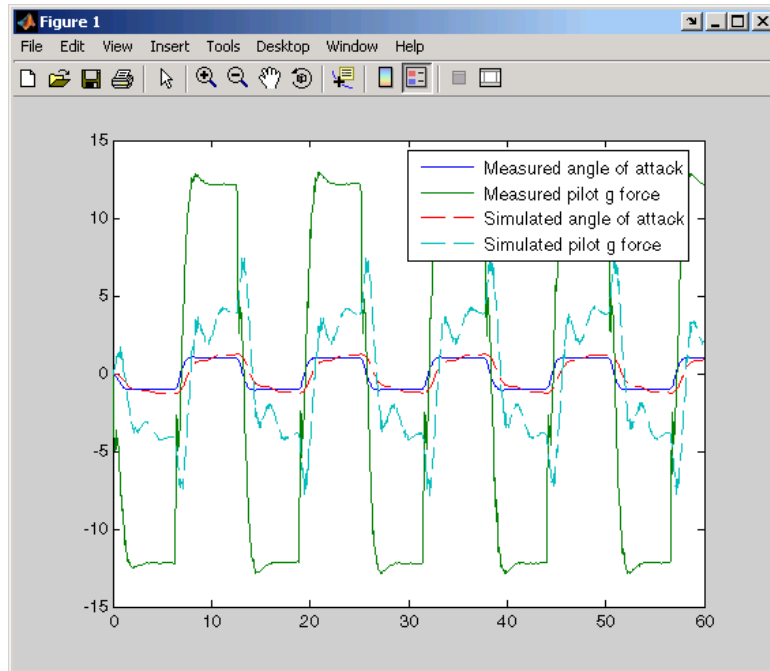

```
%% Open the model and load experimental data.
open_system('f14')
load f14_estim % Load empirical I/O data.

%% Set initialize unknown parameters
% Actuator time constant (ideal: Ta = 0.05)
Ta = 0.5;

% Aircraft dynamic model parameters (ideal: Md = -6.8847,
% Zd = -63.998)
Md = -1; Zd = -80;

%% Plot measured data and simulation results
[T,X,Y] = sim('f14', time, [], [time iodata(:,1)]);
plot(time, iodata(:,2:3), T, Y, '-');
legend( 'Measured angle of attack', 'Measured pilot g force', ...
        'Simulated angle of attack', 'Simulated pilot g force');
```

The following figure appears.



As you can see, the measured and simulated data are a poor match. The rest of this section describes how to estimate values for T_a , Z_d , and M_d that result in a better match of data sets.

Creating a Transient Experiment Object

After you have a model and identify the parameters you want to estimate, the next step is to create the objects required for an estimation. `ParameterEstimator` is both the name of the *class* and the *object* instantiated by that class. Classes are created by a *constructor*; objects are created by invoking the class name with parameters.

First, create an estimation project object. This is the constructor syntax:

```
hExp = ParameterEstimator.TransientExperiment('f14')
```

For more information, see “Transient Experiment Objects” on page 2-103.

This command returns the following information about the f14 model.

```

Experimental transient data set for the model 'f14':

Output Data
(1) f14/alpha (rad)
(2) f14/Nz Pilot (g)

Input Data
(1) f14/u

Initial States
(1) f14/Actuator Model
(2) f14/Aircraft Dynamics Model/Transfer Fcn.1
(3) f14/Aircraft Dynamics Model/Transfer Fcn.2
(4) f14/Controller/Alpha-sensor Low-pass Filter
(5) f14/Controller/Pitch Rate Lead Filter
(6) f14/Controller/Proportional plus integral compensator
(7) f14/Controller/Stick Prefilter
(8) f14/Dryden Wind Gust Models/Q-gust model
(9) f14/Dryden Wind Gust Models/W-gust model

```

Assigning Experimental Data to Inputs and Outputs of the Model

After you create a `ParameterEstimator` object, assign input and output experimental (i.e., empirical) data.

```

%% Create objects to represent the experimental data sets.
set(hExp.InputData(1), 'Data', iodata(:,1), 'Time', time);

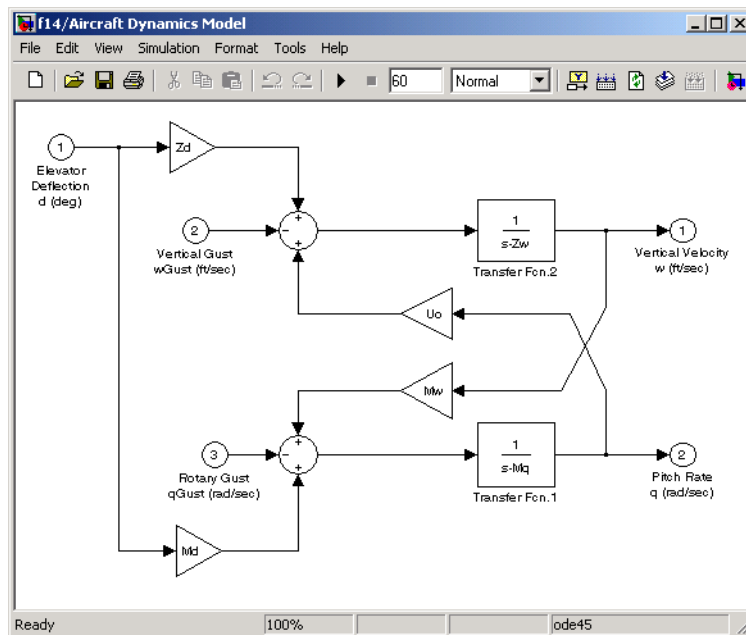
set(hExp.OutputData(1), 'Data', iodata(:,2), 'Time', ...
    time, 'Weight', 5);
set(hExp.OutputData(2), 'Data', iodata(:,3), 'Time', time);

```

Note In general, for models with multiple inputs and outputs, you must independently assign one data object to each input and output port. The data object you assign to a specific port can be a vector or a matrix that corresponds to that channel. You cannot use a single I/O port to represent multiple channels.

Creating Parameter Objects for Estimation

To activate parameters for estimation, you must create parameter objects for the parameters you want to estimate. For this example, use T_a , the actuator time constant, and Z_d and M_d , the vertical velocity and pitch rate gains, respectively. The Z_d and M_d gains are located in the F14 aircraft dynamics subsystem.



First, create `ParameterEstimator` objects for the parameters you want to estimate.

```
% Create objects to represent parameters.
hPar(1) = ParameterEstimator.Parameter('Ta');
set(hPar(1), 'Minimum', 0.01, 'Maximum', 1, 'Estimated', true)

hPar(2) = ParameterEstimator.Parameter('Md');
set(hPar(2), 'Minimum', -10, 'Maximum', 0, 'Estimated', true)

hPar(3) = ParameterEstimator.Parameter('Zd');
set(hPar(3), 'Minimum', -100, 'Maximum', 0, 'Estimated', true)

% Create objects to represent initial states.
hIc(1) = ParameterEstimator.State('f14/Actuator Model');
set(hIc(1), 'Minimum', 0, 'Estimated', false);
```

For more information, see “Parameter Objects” on page 2-106.

You can also use dot notation here. For example, instead of

```
set(hPar(2), 'Minimum', -10, 'Maximum', 0, 'Estimated', true)
```

you can write

```
hPar(2).Estimated=true;
hPar(2).Minimum=-10;
hPar(2).Maximum=0;
```

Creating an Estimation Object and Running the Estimation

Finally, create an estimation object and run the estimation, using `gcs` to get the full pathname to the Simulink model.

```
hEst = ParameterEstimator.Estimation(gcs, hPar, hExp);
hEst.States = hIc;

%% Setup estimation options
hEst.OptimOptions.Method      = 'lsqnonlin';
hEst.OptimOptions.GradientType = 'refined';
hEst.OptimOptions.Display     = 'iter';

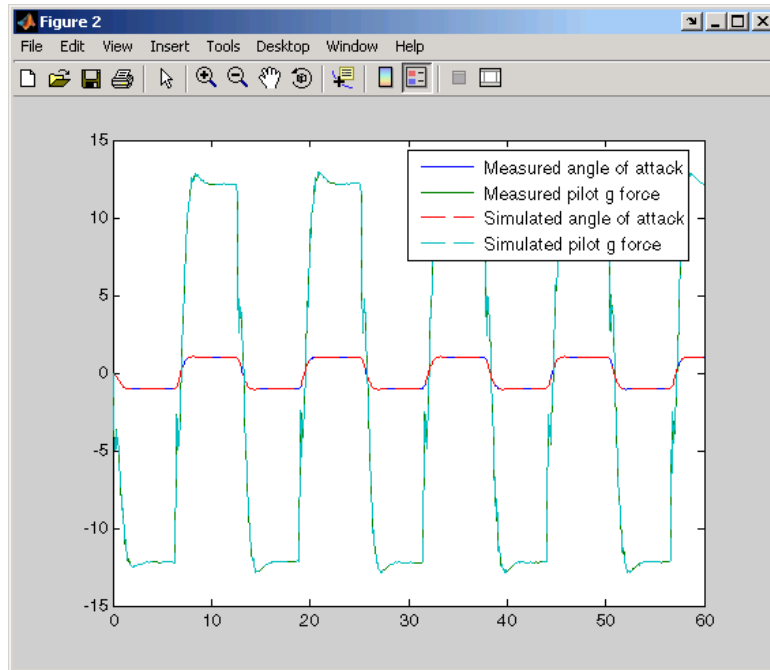
%% Run the estimation
estimate(hEst);
```

The estimation runs for a few iterations. After the estimation completes, plot the measured data and the simulated response.

```
%% Plot measured data and final simulation results
[T,X,Y] = sim('f14', time, [], [time iodata(:,1)]);
figure
plot(time, iodata(:,2:3), T, Y, '-');
legend( 'Measured angle of attack', 'Measured pilot g force', ...
        'Simulated angle of attack', 'Simulated pilot g force');
```

For more information, see “Estimation Objects” on page 2-113.

This figure shows the results of the estimation.



The measured and simulated outputs now appear to be a close match. Next, look at the estimated values to see how they compare with the default values of the f14 model.

```
%% Look at the estimated values
find(hEst.Parameters, 'Estimated', true)
```

This command returns the following result:

(1) Parameter data for 'Ta':

Parameter value : 0.05

Initial guess : 0.5

Estimated : true

Referenced by:

(2) Parameter data for 'Md':

Parameter value : -6.884

Initial guess : -1

Estimated : true

Referenced by:

(3) Parameter data for 'Zd':

Parameter value : -63.99

Initial guess : -80

Estimated : true

Referenced by:

Note You can use the `find` command to identify scalar, vector, or matrix parameters. The dimensions of the `Estimated` value you specify as the `find` argument must match the dimensions of the parameters you are trying to find. For example,

```
find(hEst.Parameters, 'Estimated', true)
```

finds only scalar estimated parameters. However,

```
find(hEst.Parameters, 'Estimated', [true;true])
```

finds only vector estimated parameters with dimensions 1-by-2 and excludes all scalar parameters.

You can verify that these values match the default values of the `f14` model by clearing your workspace, loading the model, and checking the values.

```
clear all
f14
whos
```

Objects for Parameter Estimation

The following sections describe in more detail how to create and modify transient data and estimation objects:

- “Transient Data Objects” on page 2-96
- “State Data Objects” on page 2-100
- “Transient Experiment Objects” on page 2-103
- “Parameter Objects” on page 2-106
- “State Objects” on page 2-109
- “Estimation Objects” on page 2-113

First, a quick look at terminology:

- *Objects* are instantiations of *classes*.
- *Classes* contain, or rather, define, *properties* and *methods*.
- You use a *constructor* to create an instance of an object, and use the `set` method or dot notation to modify the properties of your objects.

Transient Data Objects

- “What is a Transient Data Object” on page 2-96
- “Constructor” on page 2-96
- “Properties of Transient Data Objects” on page 2-97
- “Modifying Properties of Transient Data Objects” on page 2-99
- “Using Class Methods” on page 2-100

What is a Transient Data Object. The `@TransientData` object encapsulates the data measured at a single input or output of a physical system during an experiment. Transient data objects are associated with three types of Simulink blocks:

- Inport blocks
- Outport blocks
- Internal blocks used in conjunction with signal logging.

Each `@TransientData` object describes the time history of a signal at a Simulink port. A data set is identified by the `Block` property of this object corresponding to a block name in the Simulink model. A `PortNumber` value is also necessary for internal blocks to uniquely identify signals within the block diagram.

Constructor. Estimating parameters requires a transient data object, which you create using a constructor. The syntax to create a transient data object is

```
% I/O port block
h = ParameterEstimator.TransientData('block');
% Internal block
```

```

h = ParameterEstimator.TransientData('block',portnumber);

h = ParameterEstimator.TransientData('block',data,time);
h = ParameterEstimator.TransientData('block',data,Ts);
h = ParameterEstimator.TransientData('block',portnumber,data,time);
h = ParameterEstimator.TransientData('block',portnumber,data,Ts);

```

Properties of Transient Data Objects. Descriptions of properties of the transient data object and the associated input parameters are as follows.

Transient Data Object Properties

Property	Description
Block	Name of the Simulink block with which the data is associated. Must be a string.
PortType	The type of signal that this object represents is determined in the constructor from the Block property, which may be Inport, Outputport, or Signal.
PortNumber	For data associated with the outputs of regular blocks or subsystems, this property specifies the output port number of interest. The default value is 1.
Dimensions	Dimensions of the data required for this data set. It is computed from the CompiledPortDimensions property of the appropriate port of the block, and it defines the size of other properties. Currently, Simulink supports scalar, vector, or matrix signals, so Dimensions is either a scalar or a 1-by-2 array.

Transient Data Object Properties (Continued)

Property	Description
Data	<p>Actual experimental data. Its size must be consistent with the <code>Dimensions</code> property. To conform with Simulink conventions, the data is stored in the following formats:</p> <ul style="list-style-type: none"> • Scalar or vector-valued data. The data is of the form $Ns \times m$, where Ns is the number of data samples, and m is the number of channels in the signal. • Multidimensional data (matrix and higher dimensions). The data is of the form $m_1 \times \dots \times m_n \times Ns$, where Ns is the number of data samples, and m_i is the number of channels in the ith dimension of the signal. • For missing or unspecified data, NaNs are used.
Ts, Tstart, Tstop	<p>For uniformly sampled data, <code>Ts</code> is the sample time and <code>Tstart</code> is the start time of the signal. The stop time <code>Tstop</code> and the time vector <code>Time</code> are given by</p> $Tstop = Tstart + Ts * (Ns - 1)$ $Time = Tstart : Ts : Tstop$ <p>For nonuniform time data, <code>Ts</code> is set to NaN, and the start and stop times are calculated from the time vector.</p>
Time	<p>The time data in column vector format. The length of <code>Time</code> must be consistent with the number of samples in <code>Data</code>.</p> <p>For a nonuniformly spaced <code>Time</code> vector, its length should match the length of <code>Data</code>.</p> <p>Otherwise, <code>Time</code> is automatically adjusted based on the length of <code>Data</code>.</p> <p>Modifying <code>Ts</code> resets <code>Time</code> internally. In this case, <code>Time</code> is a virtual property whose value is computed from <code>Ts</code> and <code>Tstart</code> when you request it. The rules for setting time related properties are</p> <ul style="list-style-type: none"> • Modifying <code>Time</code> sets

Transient Data Object Properties (Continued)

Property	Description
	<pre>Ts = NaN Tstart = Time(1)</pre> <ul style="list-style-type: none"> • If the time vector is uniformly spaced, a sample time Ts is calculated. • Modifying Tstart translates time forward or backward. • Modifying Ts sets Time = [] internally and generates it when required by the simulation.
Weight	The weight associated with each channel of this data set. It is used to specify the relative importance of signals. The default value is 1.
InterSample	Interpolation method between samples can be zero-order hold (zoh) or first-order hold (foh). This property is used for data preprocessing.

Modifying Properties of Transient Data Objects. After a transient data object is created, you can modify its properties using this syntax:

```
in1.Data = rand(2,1,10); % 10 data values each of size [2 1]
in1.Time = 1:10; % Automatically converted to column vector
```

Some properties (e.g., Weight) support scalar expansion with respect to the value of the Dimensions property.

Example: Assigning Input Port Data

To assign data to an input port with 2-by-3 port dimensions, use

```
in1 = ParameterEstimator.TransientData(gcb, rand(2,3,100), 0.05)
```

This command returns the following result:

```
(1) Transient data for Inport block 'portdata_test_noSim/By//Pass
Air Valve Voltage':
Sampling interval: 0.05 sec.
Data set has 100 samples and 6 channels.
```

Using Class Methods. Descriptions of two important methods are given next:

- `select` — Extracts a portion of data. The result is returned in a new transient data object.

```
in2 = select(in1, 'Sample', 10:100); % 91 samples
in3 = select(in1, 'Range', [1 4]); % Samples for 1<t<4
% ... or an alternative
in3 = select(in1, 'Sample', find(in1.Time > 1 & in1.Time < 4));
```

To extract data from a subset of available channels, use

```
in4 = select(in1, 'Channel', [1 3 2]);
% channels 1,3,and 2 in this order
```

- `hiliteBlock` — Highlights the block associated with this object in the Simulink diagram.
- `update` — Updates the transient data object from the corresponding Simulink model block.

State Data Objects

- “What is a State Data Object” on page 2-101
- “Constructor” on page 2-101
- “Properties of the State Data Object” on page 2-101
- “Example: Initial Condition Data” on page 2-103
- “Modifying Properties” on page 2-103
- “Using Class Methods” on page 2-103

What is a State Data Object. The `ParameterEstimator.StateData` object defines the known states of a dynamic Simulink block. It is used in a transient estimation context to define known initial conditions of a block diagram model, and in a steady-state estimation context to define the known states of the model.

For example, the Simulink model of a simple mass-spring-damper system has two integrator blocks to generate velocity and position signals from acceleration and velocity values, respectively, during simulation. If the corresponding physical system is known to be at rest at the beginning of an experiment, the initial states (velocity and position) of these integrators are zero. So, two `@StateData` objects can be created to describe these known initial conditions.

Constructor. The syntax for creating this object is

```
h = ParameterEstimator.StateData('block');
h = ParameterEstimator.StateData('block', data);
```

In the first constructor, the state vector is initialized from the model containing the block.

Properties of the State Data Object. Descriptions of some important properties are given in the following table .

State Data Object Properties

Property	Description
Block	Name of the Simulink block whose states are defined by this object.
Dimensions	Scalar value to store the number of states of the relevant block.

State Data Object Properties (Continued)

Property	Description
Data	<p>Column vector to store the initial value of the state for the block specified by this object. The length of this vector should be consistent with the <code>Dimensions</code> property. Since the underlying Simulink model also stores an initial state vector for all dynamic blocks, the following conventions are used to resolve the initial state values during estimations:</p> <ul style="list-style-type: none"> • If <code>Data</code> is not empty, use it when forming the state vector. • If <code>Data</code> is empty, get the state vector for this block from the model. This behavior is useful when using helper methods to create an experiment object that instantiates empty state data objects for all dynamic blocks in the Simulink model. • If there is no state data object for a dynamic block in the model, get the state vector of that block from the model. This behavior is useful for command-line users, when there are too many states in the model and only a few of them have to be set to different initial values.
Ts	<p>Sampling time of discrete blocks. Set to 0 for continuous blocks. This property is read only and is currently used for information only.</p>
Domain	<p>String to hold the physical domain of the block. Used for <code>SimMechanics</code> or <code>SimPowerSystems</code> blocks with states.</p>

Example: Initial Condition Data. To create an empty initial condition object for the engine_idle_speed/ TransferFcn2, use

```
st1 = ParameterEstimator.StateData ...
('engine_idle_speed/Transfer Fcn2', [1 2])
```

```
(1) State data for 'f14/Dryden Wind Gust Models/W-gust model'
block:
The block has 2 continuous state(s).
State value : [1;2]
```

Modifying Properties. After a state data object is created, you can modify its properties using this syntax:

```
st1.Data = [2 3]; % State vector of size 2
```

Some properties (e.g., Data) support scalar expansion with respect to the value of the Dimensions property.

Using Class Methods. Description of two important methods are given next:

- `hiliteBlock` — Highlights the block associated with this object in the Simulink diagram.
- `update` — Updates the object after the Simulink model has been modified. If the Dimensions property value changes, the other properties are reset to their default values.

Transient Experiment Objects

- “What is a Transient Experiment Object” on page 2-104
- “Constructor” on page 2-104
- “Properties of Transient Experiment Objects” on page 2-104
- “Example: Creating an F14 Experiment” on page 2-105
- “Example: Creating a Van der Pol Experiment from User Objects” on page 2-105
- “Modifying Properties” on page 2-105

- “Using Class Methods” on page 2-106

What is a Transient Experiment Object. The @TransientExperiment object encapsulates the data measured at the input and output ports of a system during a single experiment, as well as the system’s known initial states.

Constructor. The syntax to create a transient experiment object is

```
h = ParameterEstimator.TransientExperiment('model');
```

where model specifies the name of the Simulink model.

Properties of Transient Experiment Objects. Descriptions of some important properties are given in the following table.

Transient Experiment Object Properties

Property	Description
Model	Simulink model with which this experiment is associated.
InputData, OutputData	Transient data objects associated with appropriate I/O blocks in the model. Blocks with unassigned objects or objects with no data are not used in estimations, meaning: <ul style="list-style-type: none"> • For input ports, assign zeros to these ports/channels during simulation. • For output ports, don’t use these ports/channels in the cost function.
InitialStates	State data objects associated with appropriate dynamic blocks in the model.
InitFcn	Function to be executed to configure the model for this particular experiment.

Example: Creating an F14 Experiment. To create an empty transient experiment for the f14 model, use

```
exp1 = ParameterEstimator.TransientExperiment('f14')
Experimental (Transient) data set for the model 'f14':
Outputs
(1) f14/alpha (rad)
(2) f14/Nz Pilot (g)
Inputs
(1) f14/u
Initial States
(1) f14/Actuator Model
(2) f14/Aircraft Dynamics Model/Transfer Fcn.1
(3) f14/Aircraft Dynamics Model/Transfer Fcn.2
(4) f14/Controller/Alpha-sensor Low-pass Filter
(5) f14/Controller/Pitch Rate Lead Filter
(6) f14/Controller/Proportional plus integral compensator
(7) f14/Controller/Stick Prefilter
(8) f14/Dryden Wind Gust Models/Q-gust model
(9) f14/Dryden Wind Gust Models/W-gust model
```

Example: Creating a Van der Pol Experiment from User Objects. To create a transient experiment from user objects for I/Os and states, use

```
out1 = ParameterEstimator.TransientData('vdp/Out1');
ic1 = ParameterEstimator.StateData('vdp/x1');
exp1 = ParameterEstimator.TransientExperiment...
(gcs, [], out1, ic1);
Experimental (Transient) data set for the model 'vdp':
Outputs
(1) vdp/Out1
Inputs
(none)
Initial States
(1) vdp/x1
```

Modifying Properties. The objects referred in InputData, OutputData, and InitialStates properties can be modified or removed as necessary.

Using Class Methods. The description of one important method is given next:

`update` — Updates the object after the Simulink model has been modified. The object listed in the `InputData`, `OutputData`, and `InitialStates` properties are updated in turn.

Parameter Objects

- “What is a Parameter Object” on page 2-106
- “Constructor” on page 2-106
- “Properties of Parameter Objects” on page 2-107
- “Example: F14 Model” on page 2-108
- “Example: Gain Matrix” on page 2-108
- “Modifying Properties” on page 2-109
- “Using Class Methods” on page 2-109

What is a Parameter Object. The `@Parameter` object refers to the parameters of the Simulink model marked for estimation. Some of the Simulink model parameters are to be estimated and storage is required for the initial values, current values, ranges, etc. One `@Parameter` object corresponds to each parameter in the Simulink model to be potentially estimated. These objects represent estimation parameters of any type such as scalars, vectors, and multidimensional arrays.

Constructor. The syntax to create a parameter object is

```
h = ParameterEstimator.Parameter('Name');  
h = ParameterEstimator.Parameter('Name', Value);  
h = ParameterEstimator.Parameter('Name', Value, Minimum,  
    Maximum);
```

In the first case, `Name` is a workspace variable. In the other cases, `Name` does not need to exist in the workspace at the time of object creation. However, it is required at estimation time.

Properties of Parameter Objects. Descriptions of some important properties are given in the following table.

Parameter Object Properties

Property	Description
Name	Parameter name. The parameter can be a multidimensional array of any size.
Dimensions	Dimensions of the value of the parameter. This is the defining property for the size of other properties.
Value	The current or estimated value of the parameter. This is the defining property for size checking and scalar expansions.
Estimated	<p>A Boolean array of the same size as that of <code>Value</code>. Depending on the value of the elements of the <code>Estimated</code> property, the behavior of the corresponding elements of <code>Value</code> is as follows:</p> <ul style="list-style-type: none"> • The elements of <code>Value</code> is estimated if the corresponding elements in <code>Estimated</code> are set to true. The result is stored in the <code>Value</code> property. • The elements of <code>Value</code> are not estimated if the corresponding elements in <code>Estimated</code> are set to false. However, these elements are used to reset the corresponding workspace parameter during estimations. <p>This property is set to false by default, meaning that the parameter value is not estimated.</p>

Parameter Object Properties (Continued)

Property	Description
InitialGuess	<p>Separate properties are required to hold the initial and current values of the parameters. So, when the InitialGuess property is initialized with a value, both it and the Value property are assigned the same value. Depending on the value of the elements of the Estimated property, the behavior of the corresponding elements of InitialGuess is as follows:</p> <ul style="list-style-type: none"> • If any element in Estimated is set to true, then the corresponding element of InitialGuess is used to initialize the workspace parameter during estimations. • If any element in Estimated is set to false, then the corresponding element of InitialGuess is not used in any way.
Minimum, Maximum	Parameter range.
TypicalValue	The typical values of the parameters. This property is used in estimations for scaling purposes. The default value is 1.

Example: F14 Model. To create a parameter object for the parameter Ta in the f14 model, use

```
par1 = ParameterEstimator.Parameter('Ta')
(1) Parameter data for 'Ta':
Parameter value : 0.05
Initial value : 0.05
Estimated : false
Referenced by the blocks:
f14/Actuator Model
```

Example: Gain Matrix. To create a parameter object for a matrix parameter K of size 4-by-1, use

```
par1 = ParameterEstimator.Parameter('K', [1 2 3 4])
(1) Parameter data for 'K':
Parameter value : [1;2;3;4]
Initial value : [1;2;3;4]
Estimated elements : [false;false;false;false]
Referenced by the blocks:
```

Modifying Properties. After a parameter object is created, you can modify its properties using this syntax:

```
par1.Estimated = true; % Estimate this parameter
```

Most of the properties, for example, `Estimated` and `TypicalValue` support scalar expansion with respect to the size of `Value`.

Using Class Methods. Descriptions of two important methods are given next:

- `hiliteBlock` — Highlights the referenced blocks associated with parameter objects in the Simulink diagram.
- `update` — Updates the parameter object after the Simulink model has been modified. If the size of the `Value` property changes, then the other properties are reset to their default values.

State Objects

- “What is a State Object” on page 2-110
- “Constructor” on page 2-110
- “Properties of State Objects” on page 2-110
- “Example: F14 Model” on page 2-112
- “Modifying Properties” on page 2-112
- “Using Class Methods” on page 2-113

What is a State Object. The @State object is similar to the @Parameter object, and refers to the states of the Simulink model marked for estimation. One @State object corresponds to each Simulink block with states in the model.

Constructor. The syntax to create a state object is

```
h = ParameterEstimator.State('block');  
h = ParameterEstimator.State('block', Value);  
h = ParameterEstimator.State('block', Value, Minimum,  
    Maximum);
```

In the first case, the state vector is initialized from the model containing the block. In the other cases, `block` does not need to exist in the workspace at the time of object creation. However, it is required at estimation time.

Properties of State Objects. Descriptions of some important properties of state objects are given in the following table.

State Object Properties

Property	Description
Block	Name of the Simulink block whose states are defined by this object.
Dimensions	Scalar value to store the number of states of the relevant block.
Value	Column vector to store the value of the state for the block specified by this object. The length of this vector should be consistent with the Dimensions property.

State Object Properties (Continued)

Property	Description
Estimated	<p>A Boolean array of the same size as that of <code>Value</code>. Depending on the value of the elements of the <code>Estimated</code> property, the behavior of the corresponding elements of <code>Value</code> is as follows:</p> <ul style="list-style-type: none"> • The elements of <code>Value</code> are estimated if the corresponding elements in <code>Estimated</code> are set to true. The result is stored in the <code>Value</code> property. • The elements of <code>Value</code> are not estimated if the corresponding elements in <code>Estimated</code> are set to false. However, these elements are used to reset the corresponding states during estimations. <p>This property is set to false by default, meaning that the state value is not estimated.</p>
InitialGuess	<p>Separate properties are required to hold the initial and current values of the states. So, when the <code>InitialGuess</code> property is initialized with a value, both it and the <code>Value</code> property are assigned the same value. Depending on the value of the elements of the <code>Estimated</code> property, the behavior of the corresponding elements of <code>InitialGuess</code> is as follows:</p> <ul style="list-style-type: none"> • If any element in <code>Estimated</code> is set to true, then the corresponding element of <code>InitialGuess</code> is used to initialize the state during estimations. • If any element in <code>Estimated</code> is set to false, then the corresponding element of <code>InitialGuess</code> is not used in any way.
Minimum, Maximum	State vector range.
TypicalValue	The typical values of the states. This property is used in estimations for scaling purposes. The default value is 1.

State Object Properties (Continued)

Property	Description
Ts	Sampling time of discrete blocks. Set to zero for continuous blocks. This property is read-only and is currently used for information only.
Domain	String to hold the physical domain of the block. Used for SimMechanics™ or SimPowerSystems™ blocks with states.

Example: F14 Model. To create a state object for the f14/Actuator Model block in the f14 model, use

```
st1 = ParameterEstimator.State(gcb)
```

This command returns the following result:

(1) State data for f14/Actuator Model block:

The block has 1 continuous state(s).

```
State value : 0
Initial guess : 0
Estimated : false
```

Modifying Properties. After a state object is created, you can modify its properties using this syntax:

```
ic1.Estimated = true; % Estimate this state
```

Most of the properties, for example, Estimated and TypicalValue, support scalar expansion with respect to the size of Value.

Using Class Methods. Description of two important methods are given next:

- `hiliteBlock` — Highlights the referenced blocks associated with state objects in the Simulink diagram.
- `update` — Updates the state object after the Simulink model has been modified. If the size of `Value` property changes, then the other properties are reset to their default values.

Estimation Objects

- “What is an Estimation Object” on page 2-113
- “Constructor” on page 2-113
- “Properties of Estimation Objects” on page 2-113
- “Example: F14 Model” on page 2-115
- “Modifying Properties” on page 2-115
- “Using Class Methods” on page 2-115

What is an Estimation Object. The `@Estimation` object defines the estimation problem, and is the coordinator between the model, experiments, parameter objects, and state objects.

Constructor. The `@Estimation` object is the coordinator of the model, experiment, and parameter objects. The syntax to create an estimation object is

```
h = ParameterEstimator.Estimation('model');  
h = ParameterEstimator.Estimation('model', hParam);  
h = ParameterEstimator.Estimation('model', hParam, hExps);
```

Properties of Estimation Objects. Descriptions of some important properties of estimation objects are given in the following table.

Estimation Object Properties

Property	Description
Model	Name of the Simulink model with which this estimation is associated.
Experiments	Experiments to be used in estimations. For multiple experiments, the cost function uses a concatenation of the output error vectors obtained using each experimental data set.
Parameters	Parameter objects to be used in estimations.
States	State objects to be used in estimations. This is a handle matrix with as many columns as there are experiments, and as many rows as there are states in Model. The handle matrix is created automatically in the constructor. You can reorganize its rows to specify shared states between experiments, and set the Estimated flag of desired states. If state data is provided in an experiment, the state objects stored in the columns of this matrix are initialized from the experiments.
SimOptions	Same as simset structure. This property is initialized to simget(this.Model).
OptimOptions	Same as optimset structure.
EstimInfo	This property is used to store estimation-related information at each iteration of the optimizer, and is initialized as <pre>this.EstimInfo = struct('Cost', [],... 'Covariance', [],... 'FCount', [],... 'FirstOrd', [],... 'Gradient', [],... 'Iteration', [],... 'Procedure', [],... 'StepSize', [],... 'Values', []);</pre>

Example: F14 Model. To create an estimation object for the f14 model to estimate the parameters Ta and Kf and two states, use

```
exp1 = ParameterEstimator.TransientExperiment(gcs);
par1 = ParameterEstimator.Parameter('Ta', 'Estimated', true);
par2 = ParameterEstimator.Parameter('Kf', 'Estimated', true);
est1 = ParameterEstimator.Estimation(gcs, [par1, par2], exp1);
est1.States(1,1).Estimated = true;
est1.States(6,1).Estimated = true;
est1
```

This command returns the following result:

```
Estimated variables for the model 'f14':
```

```
Estimated Parameters
```

```
Using Experiments
```

```
(1) f14 experiment
```

```
Estimated States for Experiment 'f14 experiment'
```

```
(1) f14/Actuator Model
```

```
(6) f14/Controller/Proportional plus integral compensator
```

Modifying Properties. After an estimation object is created, you can modify its properties using this syntax:

```
est.OptimOptions.Method = 'fmincon'; % Estimation method
est.OptimOptions.Display = 'iter'; % Show estimation information
...in workspace
est.Parameters(1).Estimated = false; % Do not estimate first
...parameter
est.States(2,3).Estimated = false; % Do not estimate second state
...of third expression
```

Using Class Methods. Descriptions of some of the important methods are given next:

- `compare` — Compares an experiment and a simulation.
- `simulate` — Simulates the model with current parameters and states.
- `estimate` — Runs an estimation.
- `update` — Updates the estimation object after the Simulink model has been modified.

How to Use Parallel Computing at the Command Line

After you configure your system for parallel computing, as described in “Configuring Your System for Parallel Computing” on page 2-56, you can estimate the model parameters using the command-line functions. To learn more about parameter estimation using parallel computing, see “When to Use Parallel Computing for Estimating Model Parameters” on page 2-52, and “How Parallel Computing Speeds Up Parameter Estimation” on page 2-53.

To use parallel computing for parameter estimation at the command line:

- 1** Open the Simulink model by typing the model name at the MATLAB prompt.
- 2** Configure an estimation project, as described in “Workflow for Estimating Parameters at the Command Line” on page 2-83.
- 3** Enable the parallel computing option in the estimation project by typing the following command:

```
hEst.OptimOptions.UseParallel='always';
```

To view that the `UseParallel` property has been set, type the following command:

```
hEst.OptimOptions
```

- 4** Find the model path dependencies by typing the following command:

```
dirs=hEst.finddepend;
```

This command returns the model path dependencies in your Simulink model in the `dirs` cell array.

Note As described in “Specifying Model Dependencies” on page 2-56, the `finddepend` command may not detect all the path dependencies in your model.

- 5** (Optional) Modify `dirs` to include the model path dependencies that `finddepend` does not detect by typing the following command.

```
dirs=vertcat(dirs; '\\hostname\C$\matlab\work')
```

- 6** Assign the path dependencies to the estimation project by typing the following command:

```
hEst.OptimOptions.ParallelPathDependencies=dirs;
```

- 7** Run the estimation by typing the following command:

```
estimate(hEst);
```

For more information on how to troubleshoot estimation results you obtained using parallel computing, see “Troubleshooting” on page 2-62.

Parameter Optimization

- “Overview of Optimizing Model Parameters” on page 3-2
- “Optimizing Parameters Using the GUI” on page 3-3
- “Optimizing Parameters for Model Robustness” on page 3-42
- “Accelerating Model Simulations During Optimization” on page 3-58
- “Speeding Up Response Optimization Using Parallel Computing” on page 3-60
- “Refining and Troubleshooting Optimization Results” on page 3-72
- “Saving and Loading Response Optimization Projects” on page 3-82
- “Optimizing Parameters at the Command Line” on page 3-87

Overview of Optimizing Model Parameters

When you optimize parameters of a Simulink model to meet time-domain design requirements, Simulink Design Optimization software automatically converts time-domain requirements into a constrained optimization problem, and then solves the problem using optimization techniques. The constrained optimization problem iteratively simulates the Simulink model, compares the results of the simulations with the constraint objectives, and uses gradient methods to adjust tuned parameters to better meet the objectives.

The process for optimizing model parameters to meet time-domain design requirements consists of the following task:

- 1** “Constraining Model Signals” on page 3-3
- 2** “Specifying Design Requirements” on page 3-5
- 3** “Specifying Parameters to Optimize” on page 3-19
- 4** “Specifying Optimization Options” on page 3-27
- 5** “Running the Optimization” on page 3-39

To learn how to optimize parameters at the command line, see “Optimizing Parameters at the Command Line” on page 3-87.

Optimizing Parameters Using the GUI

In this section...

“Constraining Model Signals” on page 3-3

“Specifying Design Requirements” on page 3-5

“Specifying Parameters to Optimize” on page 3-19

“Specifying Optimization Options” on page 3-27

“Specifying the Simulation Options” on page 3-32

“Plotting Responses in the Signal Constraint Window” on page 3-36

“Running the Optimization” on page 3-39

Constraining Model Signals

Simulink Design Optimization software works by adjusting parameters in a Simulink model so that chosen response signals within the system behave in a specified way. You choose the signals that you want to shape or constrain by attaching Signal Constraint blocks to them. The constraints on the behavior of the response signals and the tuned parameters are set within the Signal Constraint blocks.

The first step in the response optimization process is to choose which signals in your Simulink model you would like to constrain and to attach Signal Constraint blocks to these signals.

Once you have selected signals to constrain, you need to attach a Signal Constraint block to each of these signals. You can find the Signal Constraint block in the Simulink Design Optimization library in the Simulink Library Browser. Alternatively, you can open Simulink Design Optimization library by typing `sdolib` at the MATLAB prompt.

To attach a Signal Constraint block to a signal in your model, drag the block from the block library into the model and join the signal line to the import of the Signal Constraint block. A model can include multiple Signal Constraint blocks, and you can attach the Signal Constraint block to any signal, including signals within subsystems of your model.

Note The Signal Constraint block is not an output block of the system and does not interfere with a linearization of your model (as opposed to blocks in the Nonlinear Control Design Blockset, the previous name for this product, which were output blocks).

Double-click a Signal Constraint block to open the Signal Constraint window associated with it. Within this window you can specify the constraints imposed on the signal. For more information, see “Specifying Design Requirements” on page 3-5. You can also specify parameters to optimize and optimization settings in this block.

Although you must specify the constraints for each signal individually within each Signal Constraint block, you only need to set the remaining settings such as tuned parameters and optimization settings within one Signal Constraint window as they apply to the whole project.

Opening a Signal Constraint window, automatically creates a response optimization project. The project consists of the following information:

- Constraints on all signals that have Signal Constraint blocks attached
- Tuned parameters in the system and specifications for these parameters such as initial guesses and maximum and minimum values
- Uncertain parameters in the system and specifications for these parameters
- Optimization and simulation setup options

A response optimization project exists within a single model; there are no cross-model projects. Additionally, although you can create different sets of constraints and tuned parameters and save these as different response optimization projects, you can only associate one project with the model at any time.

The remaining steps involved in specifying the settings of a response optimization project are discussed in the following sections:

- “Specifying Design Requirements” on page 3-5
- “Specifying Parameters to Optimize” on page 3-19

To save the project for use in a later session, see “Saving and Loading Response Optimization Projects” on page 3-82.

Specifying Design Requirements

- “What Are Design requirements” on page 3-5
- “Enforcing Signal Bounds” on page 3-5
- “Moving Constraints” on page 3-6
- “Including Gridlines on the Axes” on page 3-8
- “Positioning Constraints Exactly” on page 3-8
- “Adjusting Constraint Weightings” on page 3-9
- “Edit Design Requirement Dialog Box” on page 3-10
- “Scaling Constraints” on page 3-14
- “Splitting and Joining Constraints” on page 3-14
- “Choosing Step Response Specifications” on page 3-15
- “Tracking Reference Signals” on page 3-18

What Are Design requirements

Design requirements include the positions of the constraint bound segments and reference signals specified in the Signal Constraint block. The constraints are used in a response optimization project to define the region in which the response signal must lie.

Enforcing Signal Bounds

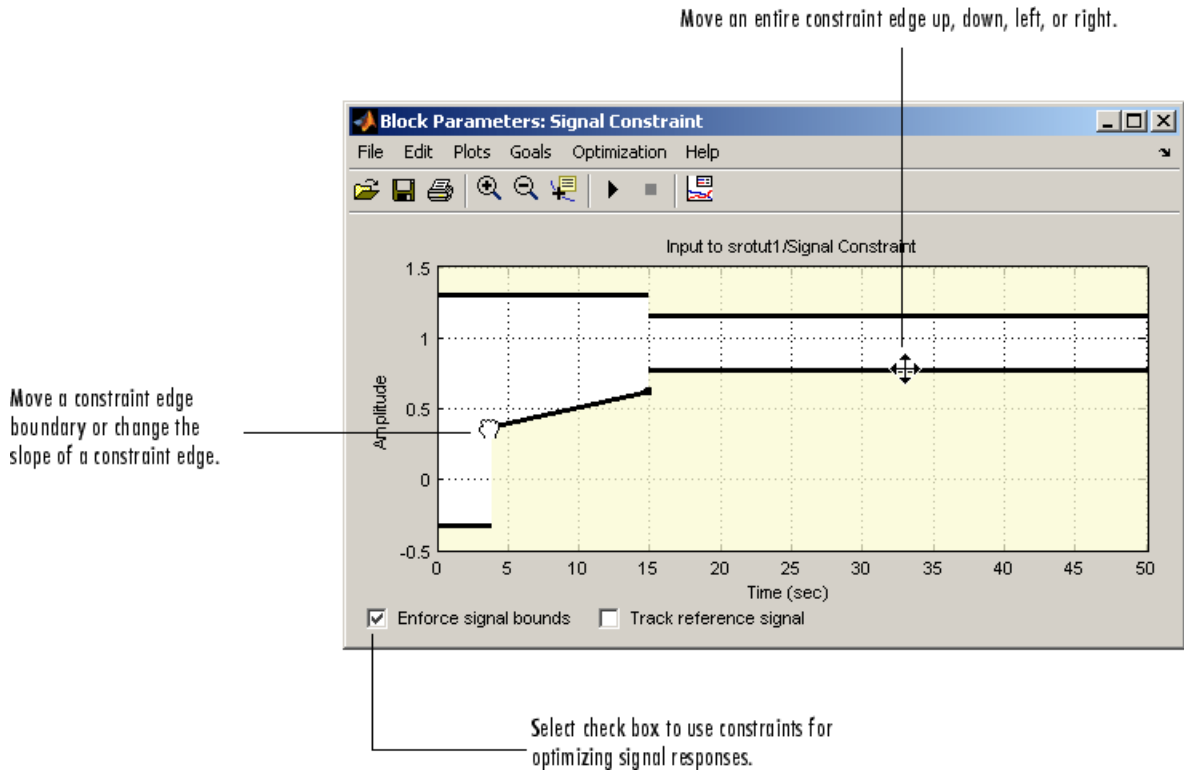
You can specify the desired response of a signal by enforcing signal bounds or by tracking a reference signal. To enforce signal bounds, select this option at the bottom of the Signal Constraint window, and then position time-domain-based constraint bound segments in the Signal Constraint window. To track a reference signal, select this option at the bottom of the Signal Constraint window, and then plot the signal in the Signal Constraint window. This section provides further details on both methods as well as instructions for editing the figure axes and plotting additional responses.

To specify the desired response signal using time-domain-based constraints, first select the **Enforce signal bounds** option at the bottom of the Signal Constraint window. Then, constrain the response signal by positioning the constraint bound segments within the figure axes using the following techniques.

When using a Signal Constraint block to directly optimize a Simulink model, by default, the start and stop time are inherited from the Simulink model. However, you can change them with the Simulation Options dialog box. Choose a stop time that captures enough of the desired response's characteristics. When you want the response to settle to a final value, use at least 10 to 20% of the simulation time for constraining the steady-state response. This ensures the proper weighting of requirements on the final value and overall stability.

Moving Constraints

Constraint-bound segments define the time-domain constraints you would like to place on a particular signal in your model. To position these segments, which appear as a yellow shaded region bordered by a black line, use the mouse to click and drag segments within the Signal Constraint window as shown in the following figure.



- To move a constraint segment boundary or to change the slope of a constraint segment, position the pointer over a constraint segment endpoint, and press and hold down the left mouse button. The pointer should change to a hand symbol. While still holding the button down, drag the pointer to the target location, and release the mouse button. Note that the segments on either side of the boundary might not maintain their slopes.
- To move an entire constraint segment up, down, left, or right, position the mouse pointer over the segment and press and hold down the left mouse button. The pointer should change to a four-way arrow. While still holding the button down, drag the pointer to the target location, and release the mouse button. Note that the segments on either side of the boundary might not maintain their slopes.

Tip To move a constraint segment to a perfectly horizontal or vertical position, hold down the **Shift** key while clicking and dragging the constraint segment. This causes the constraint segment to *snap* to a horizontal or vertical position.

To use these constraints to optimize signal responses, make sure that the **Enforce signal bounds** check box is selected at the bottom of the window.

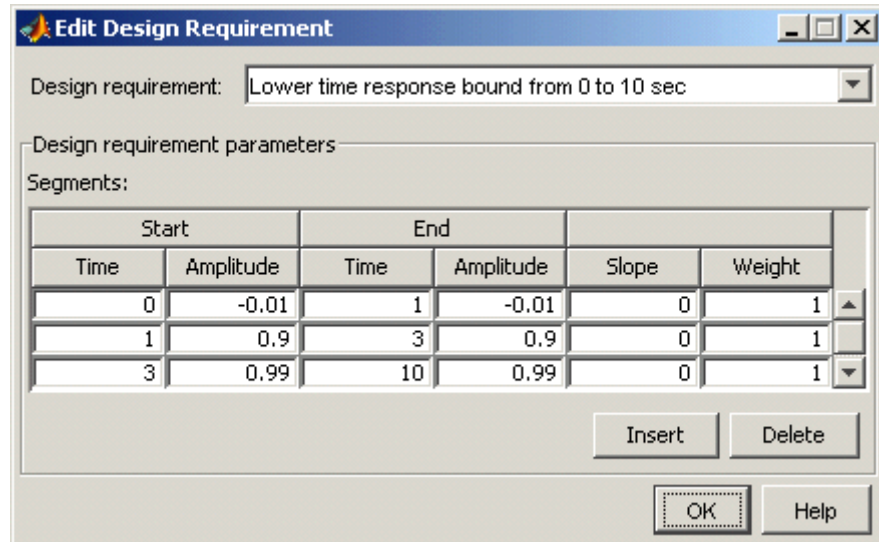
Note It is possible to move a lower bound constraint segment above an upper bound constraint segment, or vice versa, but this produces an error when you attempt to run the optimization.

Including Gridlines on the Axes

When moving constraint bound segments in the Signal Constraint window, it is sometimes helpful to display gridlines on the axes for careful alignment of the constraint bound segments. To turn the gridlines on or off, right-click within the axes of the Signal Constraint window and select **Grid**.

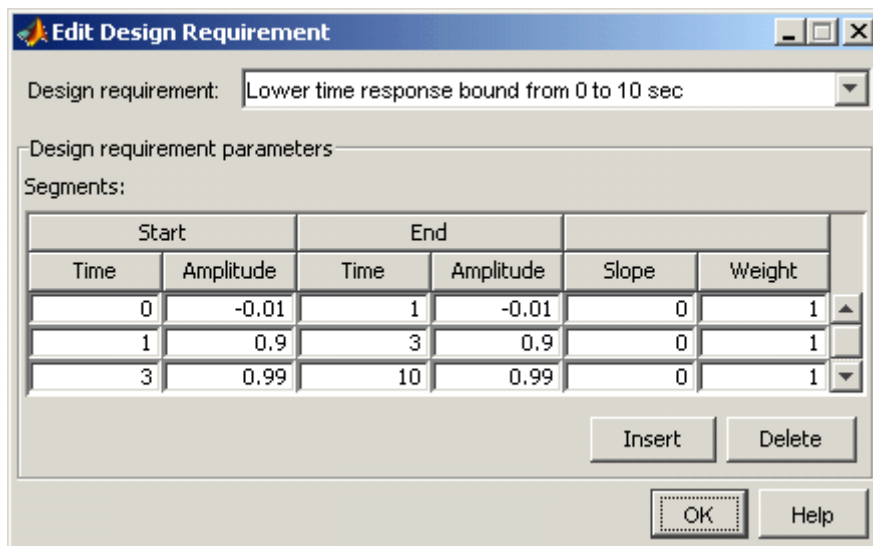
Positioning Constraints Exactly

To position a constraint segment exactly, position the pointer over the segment you want to move and press the right mouse button. Select **Edit** from the menu to open the Edit Design Requirement dialog box, shown next. For information on using the Edit Design Requirement dialog box, see “Edit Design Requirement Dialog Box” on page 3-10.



Adjusting Constraint Weightings

To change the weight of a constraint segment, position the pointer over the segment you want to weight and click the right mouse button. Select **Edit** from the menu to open the Edit Design Requirement dialog box, shown next. For information on using the Edit Design Requirement dialog box, see “Edit Design Requirement Dialog Box” on page 3-10.



Edit Design Requirement Dialog Box

The Edit Design Requirement dialog box allows you to exactly position constraint segments and to edit other properties of these constraints. The dialog box has two main components:

- An upper panel to specify the constraint you are editing
- A lower panel to edit the constraint parameters

The upper panel of the Edit Design Requirement dialog box resembles the image in the following figure.



In the context of the SISO Tool in Control System Toolbox™ software, **Design requirement** refers to both the particular editor within the SISO Tool that contains the requirement and the particular requirement within that editor. To edit other constraints within the SISO Tool, select another design requirement from the drop-down menu. In the context of the Signal Constraint block, the constraints are always time-bound constraints.

Edit Design Requirement Dialog Box Parameters. The particular parameters shown within the lower panel of the Edit Design Requirement dialog box depend on the type of constraint/requirement. In some cases, the lower panel contains a grid with one row for each segment and one column for each constraint parameter. The following table summarizes the various constraint parameters.

Edit Design Requirement Dialog Box Parameters

Parameter	Found in	Description
Time	Upper and lower time response bounds on step and impulse response plots	Defines the time range of a segment within a constraint/requirement.
Amplitude	Upper and lower time response bounds on step and impulse response plots	Defines the beginning and ending amplitude of a constraint segment.
Magnitude	SISO Tool Open-Loop Bode Editor, Prefilter Bode Editor	Defines the beginning and ending amplitude of a constraint segment.
Weight	Upper and lower time response bounds on step and impulse response plots, SISO Tool Open-Loop Bode Editor, Prefilter Bode Editor, Root Locus Editor, Open-Loop Nichols Editor	Defines the weight of a segment within a constraint/requirement. The weight is a measure of the relative importance of this constraint segment when used in a response optimization project. Weights can vary between 0 and 1, where 0 implies that the constraint segment is disabled and does not have to be satisfied, and 1 implies that the constraint segment must be satisfied. The weight of a constraint segment is graphically represented by the thickness of the black constraint line. An invisible constraint segment represents a weight of 0, and a thick constraint segment represents a weight of 1.

Edit Design Requirement Dialog Box Parameters (Continued)

Parameter	Found in	Description
Frequency	SISO Tool Open-Loop Bode Editor, Prefilter Bode Editor	Defines the frequency range of an edge within a constraint.
Slope (dB/decade)	SISO Tool Open-Loop Bode Editor, Prefilter Bode Editor	Defines the slope, in dB/decade, of a constraint segment. It is an alternative method of specifying the magnitude values. Entering a new Slope value changes any previously defined magnitude values.
Final value	Step response bounds	Defines the input level after the step occurs.
Rise time	Step response bounds	Defines a constraint segment for a particular rise time.
% Rise	Step response bounds	The percentage of the step's range used to describe the rise time.
Settling time <	SISO Tool Root Locus Editor	Defines a constraint segment for a particular settling time.
Settling time	Step response bounds	
% Settling	Step response bounds	The percentage of the final value that defines the settling region used to describe the settling time.
Percent overshoot <	SISO Tool Root Locus Editor	Defines the constraint segments for a particular percent overshoot.
% Overshoot	Step response bounds	
% Undershoot	Step response bounds	Defines the constraint segments for a particular percent undershoot.
Damping ratio >	SISO Tool Root Locus Editor	Defines the constraint segments for a particular damping ratio.

Edit Design Requirement Dialog Box Parameters (Continued)

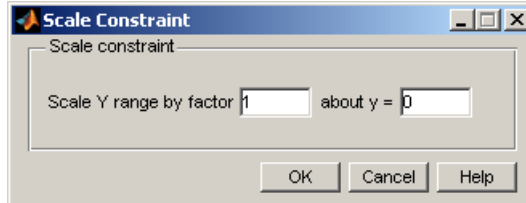
Parameter	Found in	Description
Natural frequency	SISO Tool Root Locus Editor	Defines a constraint segment for a particular natural frequency. To specify the constraint, choose at least or at most from the menu, and then specify the natural frequency of interest.
Real	SISO Tool Root Locus Editor	Defines the beginning and end of the real component of a pole-zero region constraint.
Imaginary	SISO Tool Root Locus Editor	Defines the beginning and end of the imaginary component of a pole-zero region constraint.
Phase margin >	SISO Tool Open-Loop Nichols Editor	Defines a constraint segment for a minimum phase margin. The phase margin specified should be a number greater than 0.
Located at	SISO Tool Open-Loop Nichols Editor	Defines the center, in degrees, of the constraint segment defining the phase margin, gain margin, or closed-loop peak gain. The location must be -180 plus a multiple of 360 degrees. If you enter an invalid location point, the closest valid location is selected.
Gain margin >	SISO Tool Open-Loop Nichols Editor	Defines a constraint segment for a particular gain margin.
Closed-Loop peak gain <	SISO Tool Open-Loop Nichols Editor	Defines a constraint segment for a particular closed-loop peak gain. The specified value can be positive or negative in dB. The constraint follows the curves of the Nichols plot grid, so we recommend that you have the grid on when using this feature.

Edit Design Requirement Dialog Box Parameters (Continued)

Parameter	Found in	Description
Open loop phase	SISO Tool Open-Loop Nichols Editor	Defines the beginning and end of the open loop phase component of a gain-phase constraint segment.
Open loop gain	SISO Tool Open-Loop Nichols Editor	Defines the beginning and end of the open loop gain component of a gain-phase constraint segment.

Scaling Constraints

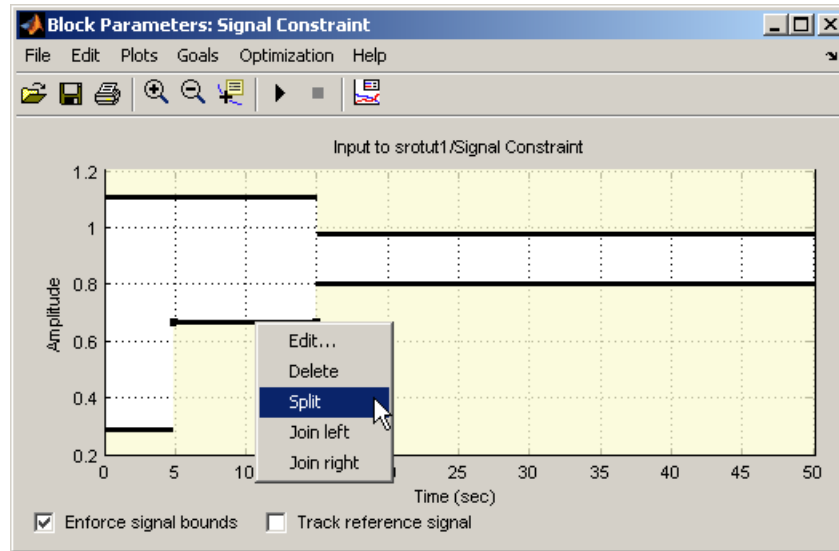
Instead of clicking and dragging the constraints to their new positions, you can scale the constraints. To scale the constraints, select **Edit > Scale Constraint** in the Signal Constraint window. This displays the Scale Constraint dialog box.



Enter the amount by which you want the constraints to scale and the point about which you want to scale them, and then click **OK**.

Splitting and Joining Constraints

To split a constraint segment, position the pointer over the segment to be split, and press the right mouse button. Select **Split** from the context menu. The segment splits in half. You can now manipulate each segment individually.

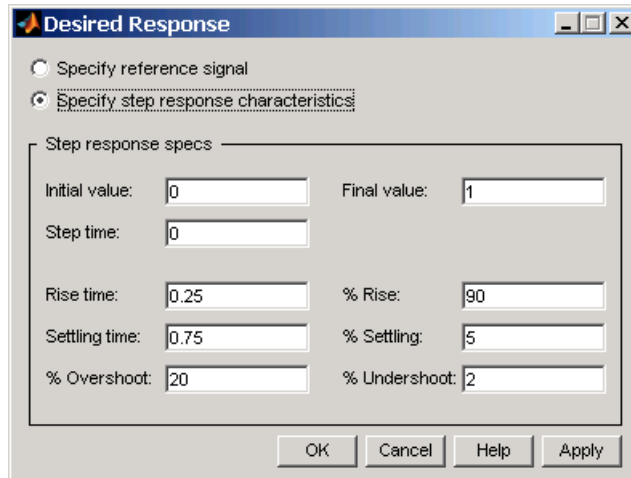


To join two neighboring constraint segments, position the pointer over one constraint segment, and press the right mouse button. Select **Join left** or **Join right** from the menu to join the segment to the left or right respectively.

Choosing Step Response Specifications

When you are optimizing the step response of your system, an alternative method of positioning the constraint bound segments is to specify the desired step response characteristics such as rise time, settling time, and overshoot.

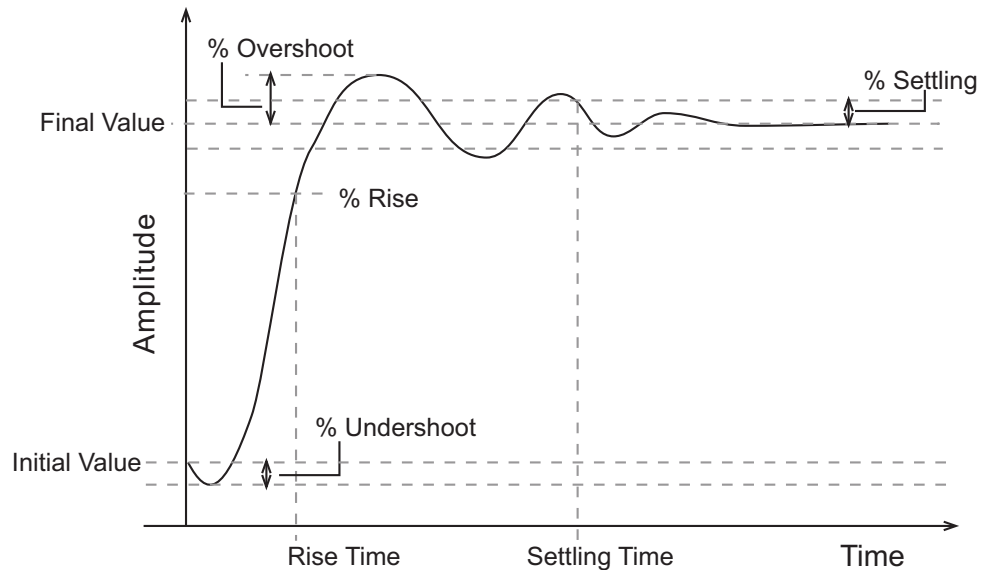
To specify step response characteristics, select **Goals > Desired Response** in the Signal Constraint window or right-click in the white space of the figure window and select **Desired Response** from the context menu. This displays the Desired Response dialog box. Select **Specify step response characteristics** to display the step response specifications as shown in the following figure.



The top three options specify the details of the step input:

- **Initial value:** Input level before the step occurs
- **Step time:** Time at which the step takes place
- **Final value:** Input level after the step occurs

The remaining options specify the characteristics of the response signal. Each of the step response characteristics is illustrated in the following figure.



- **Rise time:** The time taken for the response signal to reach a specified percentage of the step's range. The step's range is the difference between the final and initial values.
- **% Rise:** The percentage used in the rise time.
- **Settling time:** The time taken until the response signal settles within a specified region around the final value. This settling region is defined as the final step value plus or minus the specified percentage of the final value.
- **% Settling:** The percentage used in the settling time.
- **% Overshoot:** The amount by which the response signal can exceed the final value. This amount is specified as a percentage of the step's range. The step's range is the difference between the final and initial values.
- **% Undershoot:** The amount by which the response signal can undershoot the initial value. This amount is specified as a percentage of the step's range. The step's range is the difference between the final and initial values.

Enter values for the response specifications in the Response Specifications dialog box, based on the requirements of your model, and then click **OK**. The constraint segments now reflect the constraints specified.

Tracking Reference Signals

You can specify the desired response as an ideal or *reference* trajectory. You can use this reference signal in addition to, or instead of, enforcing signal bounds.

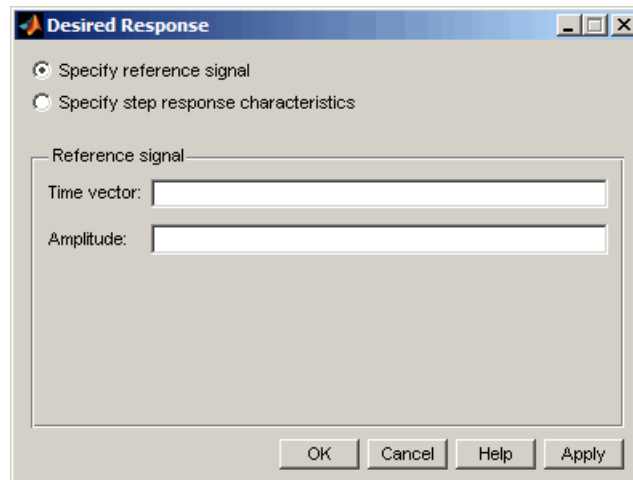
To specify a reference signal:

- 1 Select the **Track reference signal** check box at the bottom of the Signal Constraint window.



- 2 Specify a reference signal using one of the following techniques:
 - Selecting **Goals > Desired Response** in the Signal Constraint window.
 - Right-clicking in the white space of the figure window and selecting **Desired Response** from the context menu.

This action displays the Desired Response dialog box. Select the radio button labeled **Specify reference signal** to display the reference signal setup as shown in the following figure.



- 3 Define the reference signal by entering vectors, or variables from the workspace, for the time and amplitude of the signal, and then clicking **OK**.

This action plots the reference signal within the figure axes of the Signal Constraint block window.

To turn the reference signal plot on or off, right-click in the white space of the figure window, and select **Show > Reference Signal**.

Use the **Track reference signal** check box in the Signal Constraint window to enable or disable tracking the reference signal.

Note When tracking a reference signal, the software ignores the maximally feasible solution option. For more information on this option, see “Selecting Optimization Termination Options” on page 3-29.

Specifying Parameters to Optimize

- “Defining Tunable Parameters” on page 3-20
- “Adding Tuned Parameters” on page 3-21
- “Changing Tuned Parameter Specifications” on page 3-22

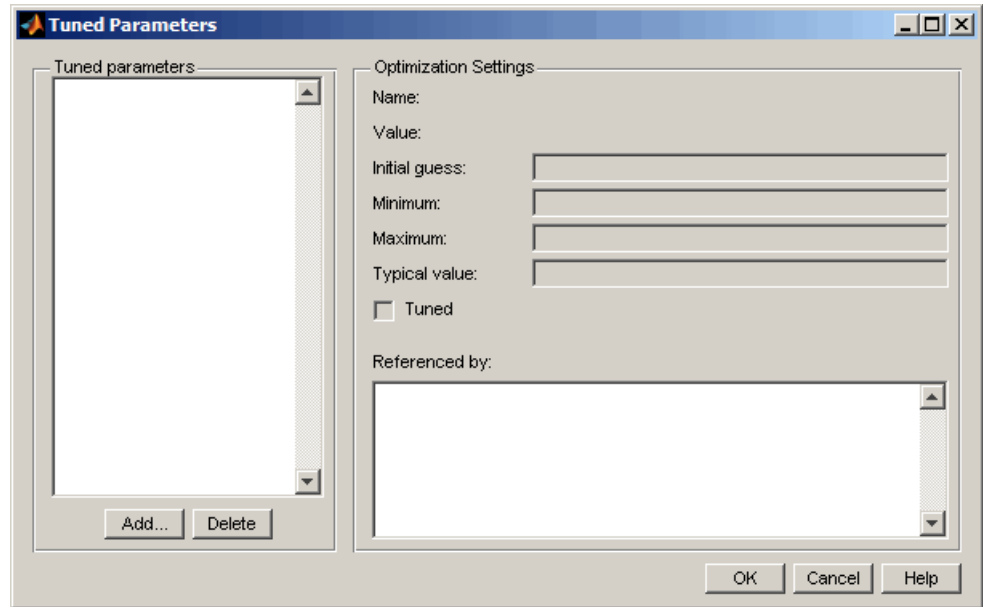
- “Specifying Independent Parameters” on page 3-23
- “Example — Specifying Independent Parameters for Optimization” on page 3-24

Defining Tunable Parameters

Before running the optimization, you must define which system parameters are tunable. By tuning these parameters, Simulink Design Optimization software makes the response signal meet the imposed constraints. In addition, you can define uncertain parameters to account for plant uncertainty in your response optimization project. The tunable and uncertain parameters can be scalar, vector, or matrix.

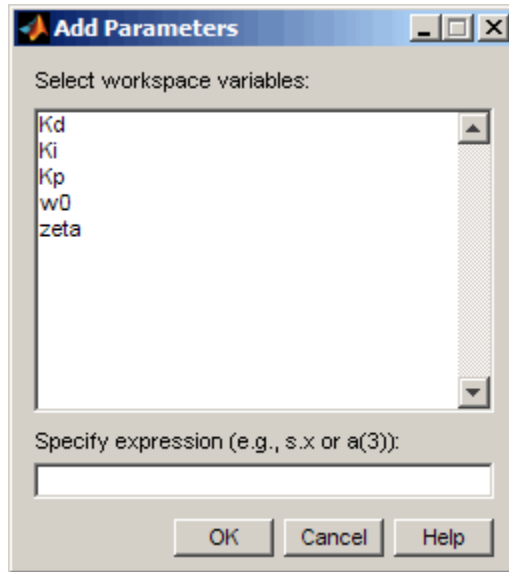
Simulink Design Optimization software optimizes the response signals of the model by varying the model's tuned parameters so that the response signals lie within the constraint bound segments or closely match a specified reference signal. You can specify these tuned parameters by selecting **Optimization > Tuned Parameters** in a Signal Constraint window.

Note When you have more than one Signal Constraint block in your model, you need to specify the tuned parameters in only one window as these settings apply to all constrained signals within the model.



Adding Tuned Parameters

Within the Tuned Parameters dialog box, the tuned parameters are shown in a list on the left. To add a tuned parameter to your response optimization project, click the **Add** button. This action opens the Add Parameters dialog box which lists all model parameters currently available in the MATLAB workspace.



Note If a parameter is already listed in the **Tuned parameters** list of the Tuned Parameters window, it does not appear in the Add Parameters dialog box.

Select the parameters that you want to tune, then click **OK** to add them to the **Tuned parameters** list. To delete a parameter from the **Tuned parameters** list, select the parameter you want to delete and click **Delete**.

Changing Tuned Parameter Specifications

To display the settings for a particular tuned parameter, select it within the **Tuned Parameters** list. Its settings appear on the right under **Optimization Settings**, as listed in the following table.

Setting	Description	Default
Name	The name of the parameter.	Not an editable field
Value	The current value of the parameter.	Not an editable field

Setting	Description	Default
Initial guess	The initial value used by the optimization method. A well-chosen initial guess can speed up the optimization and help keep the solution away from undesirable local minima. You can edit this field with numbers, variables, or expressions to provide an alternate initial guess.	The current value of the parameter
Minimum	The minimum value, or lower bound, that you would like the parameter to take. You can edit this field to provide an alternate minimum value.	- Inf
Maximum	The maximum value, or upper bound, that you would like the parameter to take. You can edit this field to provide an alternate maximum value.	Inf
Typical value	The tuned parameters are scaled, or normalized, by dividing their current value by a typical value. You can edit this field to provide an alternate scaling factor.	The initial value of the parameter
Tuned	This check box indicates whether this parameter is tunable. Select it if you want this parameter to be tuned during the optimization. Unselect if you do not want this parameter to be tuned during the optimization but you would like to keep it on the list of tuned parameters (for a subsequent optimization).	Selected
Referenced by	A list of all blocks this parameter appears in.	Not an editable field

After selecting the tuned parameters for the project and editing their optimization settings, click **OK** to save your changes and exit the Tuned Parameters dialog box.

Specifying Independent Parameters

Sometimes parameters in your model depend on independent parameters that do not appear in the model. The following steps give an overview of how

to tune and include uncertainty in these independent parameters. example follows in the next section:

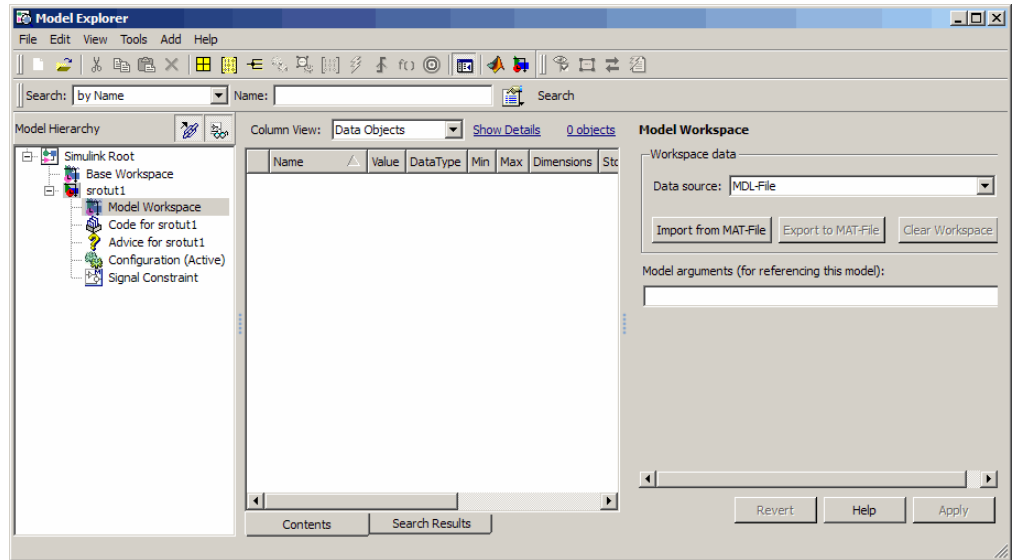
- 1 Add the independent parameters to the model workspace (along with initial values).
- 2 Define a Simulation Start function that runs before each simulation of the model. This Simulation Start function defines the relationship between the dependent parameters in the model and the independent parameters in the model workspace.
- 3 The independent parameters now appear in the Add Parameters dialog box when you select **Tuned parameters** or **Uncertain parameters**. Add these parameters to the list of tuned parameters to tune them during the response optimization.

Caution Avoid adding independent parameters together with their corresponding dependent parameters to the lists of tuned and uncertain parameters. Otherwise, the optimization could give incorrect results. For example, when a parameter x depends on the parameters a and b , avoid adding all three parameters to the lists of tuned and uncertain parameters.

Example – Specifying Independent Parameters for Optimization

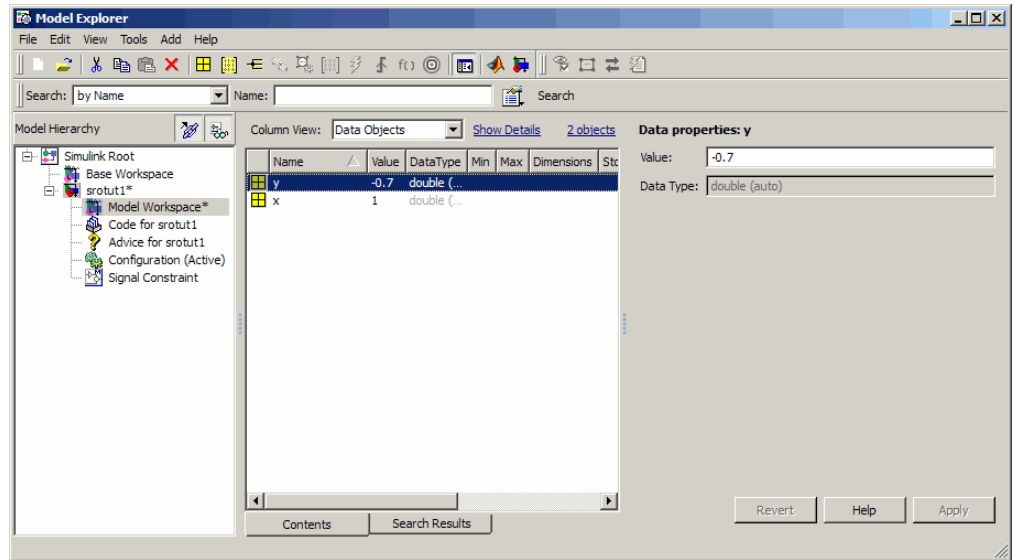
Assume that the parameter K_{int} in the model `srotut1` is related to the parameters x and y according to the relationship $K_{int}=x+y$. Also assume that the initial values of x and y are 1 and -0.7, respectively. To tune x and y instead of K_{int} , first define these parameters in the model workspace. To do this,

- 1 Select **View > Model Explorer** from the `srotut1` window to open the Model Explorer window.
- 2 In the Model Hierarchy tree, select `srotut1 > Model Workspace`.



- 3 Select **Add > MATLAB Variable** to add a new variable to the model workspace. A new variable with a default name **Var** appears in the **Name** column.
- 4 Double-click **Var** to make it editable and change the variable name to **x**. Edit the initial **Value** to **1**.
- 5 Repeat step 3 and 4 to add a variable **y** with an initial value of **-0.7**.

The Model Explorer window resembles the following figure.



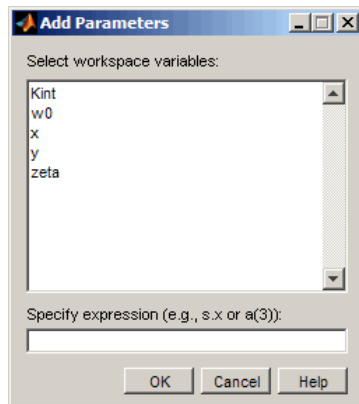
- 6 To add the Simulation Start function that defines the relationship between Kint and the independent parameters x and y, select **File > Model Properties** in the srotut1 model window.
- 7 In the Model Properties window, click the **Callbacks** tab.
- 8 To enter a Simulation start function, select **StartFcn***, and type the name of a new function. For example, srotut1_start in the **Simulation start function** panel. Then, click **OK**.
- 9 Create a MATLAB file named srotut1_start.

The content of the file defines the relationship between the parameters in the model and the parameters in the workspace. For this example, the content resembles the following:

```
wks = get_param(gcs, 'ModelWorkspace')
x = wks.evalin('x')
y = wks.evalin('y')
Kint = x+y;
```

Note You must first use the `get_param` function to get the variables `x` and `y` from the model workspace before you can use them to define `Kint`.

When you add a new tuned or uncertain parameter, `x` and `y` appear in the Add Parameters dialog box.



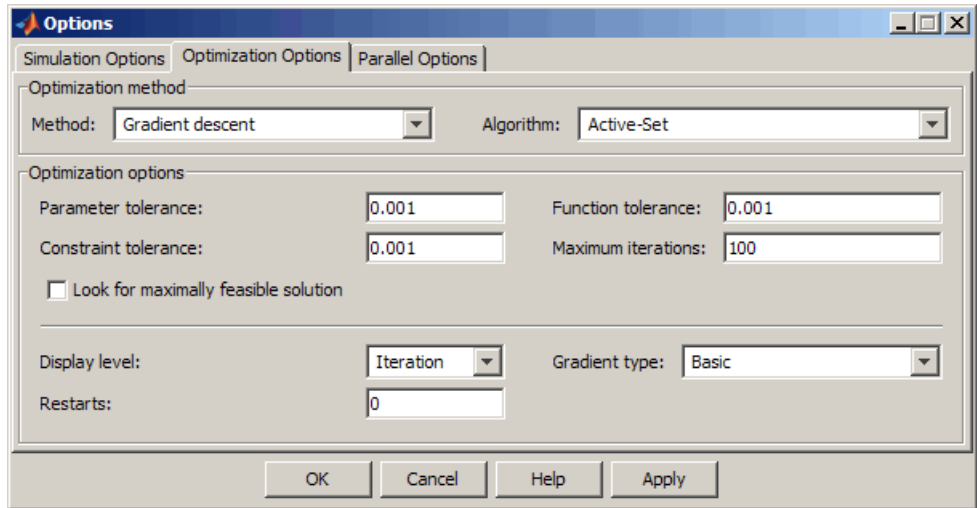
Specifying Optimization Options

- “Accessing Optimization Options” on page 3-27
- “Selecting Optimization Methods” on page 3-28
- “Selecting Optimization Termination Options” on page 3-29
- “Selecting Additional Optimization Options” on page 3-30

Accessing Optimization Options

Several options can be set to tune the results of optimization. These options include the optimization methods and the tolerances the methods use.

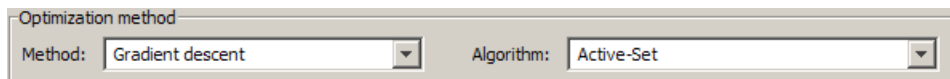
To set options for optimization, select **Optimization > Optimization Options** in the Signal Constraint window. This opens the Options dialog box.



Note If the optimization fails, a good first work-around is to change the **Gradient-type** to **Refined**. For more information on this option, refer to “Selecting Additional Optimization Options” on page 3-30.

Selecting Optimization Methods

Both the **Method** and **Algorithm** options define the optimization method. Use the **Optimization method** area of the Options dialog box to set the optimization method and its algorithm.



For the **Method** option, the three choices are:

- **Gradient descent** (default) — Uses the Optimization Toolbox function `fmincon` to optimize the response signal subject to the constraints.
- **Pattern search** — Uses the Global Optimization Toolbox function `patternsearch`, an advanced direct search method, to optimize the response. This option requires the Global Optimization Toolbox.

- **Simplex search** — Uses the Optimization Toolbox function `fminsearch`, a direct search method, to optimize the response. **Simplex search** is most useful for simple problems and is sometimes faster than Gradient descent for models that contain discontinuities.

The following table summarizes the **Algorithm** options for Gradient descent:

Algorithm Option	Learn More
Active-Set (default)	“fmincon Active Set Algorithm” in the Optimization Toolbox documentation.
Interior-Point	“fmincon Interior Point Algorithm” in the Optimization Toolbox documentation.
Trust-Region-Reflective	“fmincon Trust Region Reflective Algorithm” in the Optimization Toolbox documentation.

Selecting Optimization Termination Options

Use the **Optimization options** panel to specify when you want the optimization to terminate.

Optimization options

Parameter tolerance: Function tolerance:

Constraint tolerance: Maximum iterations:

Look for maximally feasible solution

- **Parameter tolerance:** The optimization terminates when successive parameter values change by less than this number. For more details, refer to the discussion of the parameter `TolX` in the reference page for the Optimization Toolbox function `fmincon`.

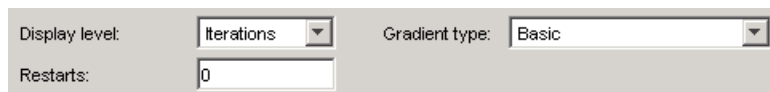
- **Constraint tolerance:** This number represents the maximum relative amount by which the constraints can be violated and still allow a successful convergence.
- **Function tolerance:** The optimization terminates when successive function values are less than this value. Changing the default **Function tolerance** value is only useful when you are tracking a reference signal or using the `Simplex` search method. For more details, refer to the discussion of the parameter `TolFun` in the reference page for the Optimization Toolbox function `fmincon`.
- **Maximum iterations:** The maximum number of iterations allowed. The optimization terminates when the number of iterations exceeds this number.
- **Look for maximally feasible solution:** When selected, the optimization continues after it has found an initial, feasible solution, until it finds a maximally feasible, optimal solution. When this option is unselected, the optimization terminates as soon as it finds a solution that satisfies the constraints and the resulting response signal sometimes lies very close to the constraint segment. In contrast, a maximally feasible solution is typically located further inside the constraint region.

Note If selected, the software ignores this option when tracking a reference signal.

By varying these parameters you can force the optimization to continue searching for a solution or to continue searching for a more accurate solution.

Selecting Additional Optimization Options

At the bottom of the **Optimization Options** panel is a group of additional optimization options.



The image shows a portion of the Optimization Options panel. It contains three controls: a dropdown menu for 'Display level' with 'Iterations' selected, a dropdown menu for 'Restart type' with 'Basic' selected, and a text input field for 'Restarts' containing the value '0'.

- “Display Level” on page 3-31

- “Restarts” on page 3-31
- “Gradient Type” on page 3-31

Display Level. The **Display level** option specifies the form of the output that appears in the Optimization Progress window. The options are **Iterations**, which displays information after each iteration, **None**, which turns off all output, **Notify**, which displays output only if the function does not converge, and **Termination**, which only displays the final output.

For more information on the type of iterative output that appears for the method you selected using the **Method** option, see the discussion of output for the corresponding function.

Method	Function	Output Information
Gradient descent	fmincon	fmincon section of “Function-Specific Output Headings” in the Optimization Toolbox documentation
Simplex search	fminsearch	fminsearch section of “Function-Specific Output Headings” in the Optimization Toolbox documentation
Pattern search	patternsearch	“Display to Command Window Options” in the Global Optimization Toolbox documentation

Restarts. In some optimizations the Hessian may become ill conditioned and the optimization does not converge. In these cases it is sometimes useful to restart the optimization after it stops, using the endpoint of the previous optimization as the starting point for the next one. To automatically restart the optimization, indicate the number of times you want to restart in this field.

Gradient Type. When using **Gradient descent** as the optimization method, Simulink Design Optimization software calculates gradients based on finite difference methods. The default method for computing the gradients is **Basic**. The **Refined** method offers a more robust and less noisy gradient calculation method than **Basic**, although it is sometimes more expensive and does not work with certain models such as SimPowerSystems models.

Tip If the optimization fails, a good first work-around, before changing solvers or adding parameter bounds, is to change **Gradient type** to Refined.

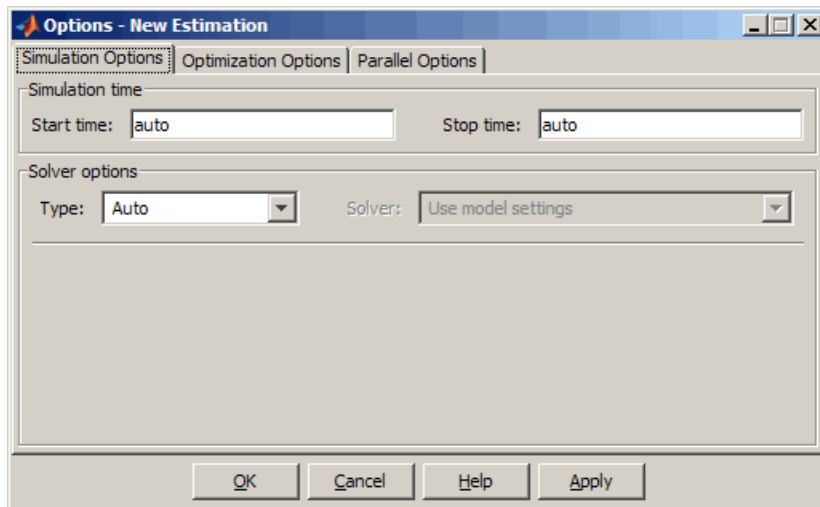
Specifying the Simulation Options

- “Accessing Simulation Options” on page 3-32
- “Selecting Simulation Time” on page 3-33
- “Selecting Solvers” on page 3-33

Accessing Simulation Options

To optimize the parameters of a model, Simulink Design Optimization software runs simulations of the model.

To set the simulation options, select **Optimization > Simulation Options** in the Signal Constraint window. This action opens the Options dialog box.



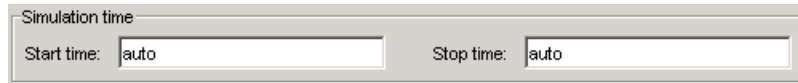
Specify the simulation options, as described in the following sections:

- “Selecting Simulation Time” on page 3-33

- “Selecting Solvers” on page 3-33

Selecting Simulation Time

By default, the **Start time** and **Stop time** are **auto**, which automatically uses the start and stop times specified in the model.

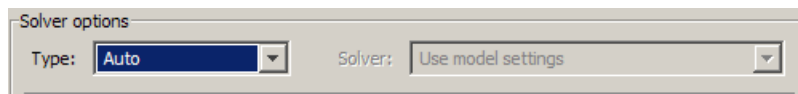


Note If the **Stop time** in the Simulink model is **inf**, the software automatically uses the largest time value in the constraints as the stop time value.

To specify alternative start and stop times for the response optimization project, enter the new times in the **Simulation time** area of the dialog box.

Selecting Solvers

When running the optimization, the software solves the dynamic system using one of the Simulink solvers. You can specify the solver type and its options in the **Solver options** area of the **Simulation Options** tab.



The solver can be one of the following **Type**:

- **Auto (default)** — Uses the simulation settings specified in the Simulink model.
- **Variable-step** — Variable-step solvers keep the error within specified tolerances by adjusting the step size the solver uses. For example, if the states of your model are likely to vary rapidly, you can use a variable-step solver for faster simulation. For more information on the variable-step solver options, see “Variable-Step Solver Options” on page 3-34.

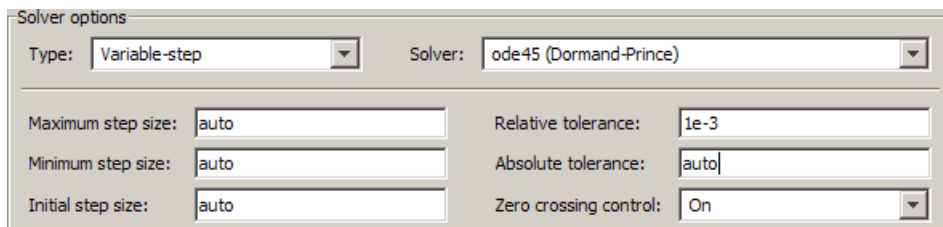
- **Fixed-step** — Fixed-step solvers use a constant step size. For more information on the fixed-step solver options, see “Fixed-Step Solver Options” on page 3-35.

See “Choosing a Solver” in the Simulink documentation for information about solvers.

Note To obtain faster simulations during optimization, you can change the solver **Type** to **Variable-step** or **Fixed-step**. However, the optimized parameter values apply only for the chosen solver type, and may differ from values you obtain using settings specified in the Simulink model.

Variable-Step Solver Options. When you select **Variable-step** as the solver **Type**, you can choose one of the following as the **Solver**:

- Discrete (no continuous states)
- ode45 (Dormand-Prince)
- ode23 (Bogacki-Shampine)
- ode113 (Adams)
- ode15s (stiff/NDF)
- ode23s (stiff/Mod. Rosenbrock)
- ode23t (Mod. stiff/Trapezoidal)
- ode23tb (stiff/TR-BDF2)



Solver options

Type: Variable-step Solver: ode45 (Dormand-Prince)

Maximum step size:	auto	Relative tolerance:	1e-3
Minimum step size:	auto	Absolute tolerance:	auto
Initial step size:	auto	Zero crossing control:	On

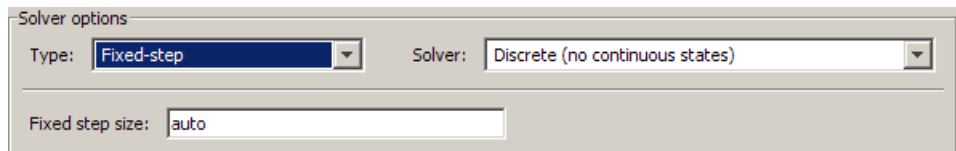
You can also specify the following parameters that affect the step-size of the simulation:

- **Maximum step size** — The largest step-size the solver can use during a simulation.
- **Minimum step size** — The smallest step-size the solver can use during a simulation.
- **Initial step size** — The step-size the solver uses to begin the simulation.
- **Relative tolerance** — The largest allowable relative error at any step in the simulation.
- **Absolute tolerance** — The largest allowable absolute error at any step in the simulation.
- **Zero crossing control** — Set to on for the solver to compute exactly where the signal crosses the x -axis. This option is useful when using functions that are nonsmooth and the output depends on when a signal crosses the x -axis, such as absolute values.

By default, the software automatically chooses the values for these options. To specify your own values, enter them in the appropriate fields. For more information, see “Solver Pane” in the Simulink documentation.

Fixed-Step Solver Options. When you select Fixed-step as the solver **Type**, you can choose one of the following as the **Solver**:

- Discrete (no continuous states)
- ode5 (Dormand-Prince)
- ode4 (Runge-Kutta)
- ode3 (Bogacki-Shampine)
- ode2 (Heun)
- ode1 (Euler)



You can also specify the **Fixed step size** value, which determines the step size the solver uses during the simulation. By default, the software automatically chooses a value for this option. For more information, see “Fixed-step size (fundamental sample time)” in the Simulink documentation.

Plotting Responses in the Signal Constraint Window

- “Types of Response Plots” on page 3-36
- “Reference Signals” on page 3-36
- “Current Response” on page 3-36
- “Initial Response” on page 3-36
- “Intermediate Steps” on page 3-37
- “Response Plots Property Editor” on page 3-37

Types of Response Plots

You can choose to plot several different signals in the Signal Constraint window, including reference signals, initial response signals, and response signals generated during the optimization.

Reference Signals

To plot a reference signal, use the methods in “Tracking Reference Signals” on page 3-18.

Current Response

To display the current response signal, based on the current parameter values, right-click within the white space of the Signal Constraint window and select **Plot Current Response**. The current response appears as a thick white line.

Initial Response

To turn the display of the initial response signal on or off, right-click within the white space of the Signal Constraint window and select **Show > Initial Response**. The initial response is the response of the signal based on parameter values in place before the optimization is run. The initial response appears as a blue line.

Intermediate Steps

To turn on, or off, the display of the response signal at intermediate steps during the optimization, right-click within the white space of the Signal Constraint window and select **Show > Intermediate Steps**. The response signal at an intermediate step is based on parameter values at an intermediate point in the optimization.

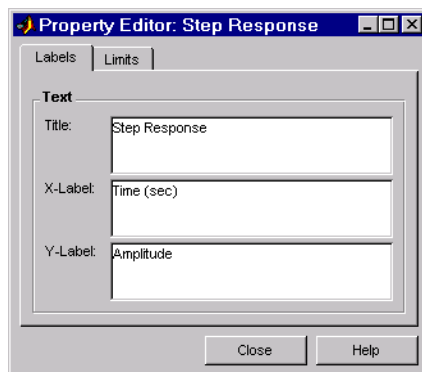
Response Plots Property Editor

- “Modifying Properties of Response Plots” on page 3-37
- “Labels Pane” on page 3-38
- “Limits Pane” on page 3-38

Modifying Properties of Response Plots. This section discusses how you can change the properties of response plots. Select **Edit > Axes Properties** in the Block Parameters: Signal Constraint window and select **Labels** to open the Property Editor dialog box.

Note Click the tabs to get help on panes in the Property Editor.

This figure shows the Property Editor dialog box for a step response.



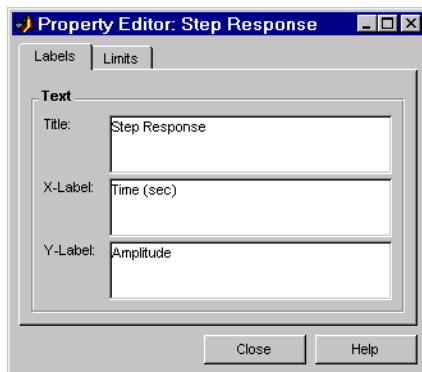
In general, you can change the following properties of response plots.

- **Labels** -- Titles and X- and Y-labels
- **Limits** -- Numerical ranges of the x - and y - axes

As you make changes in the Property Editor, they display immediately in the response plot. Conversely, if you make changes in a plot using right-click menus, the Property Editor for that plot automatically updates. The Property Editor and its associated plot are dynamically linked.

Labels Pane.

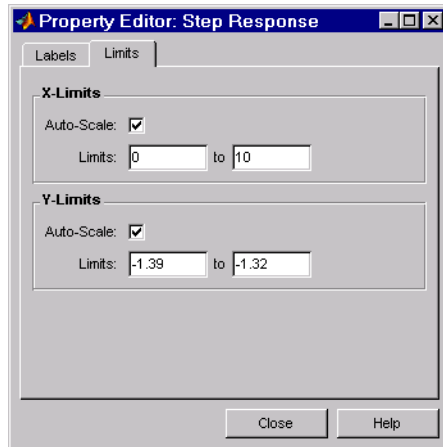
Note Click the tabs below to get help on the Property Editor.



To specify new text for plot titles and axis labels, type the new string in the field next to the label you want to change. The label changes immediately as you type, so you can see how the new text looks as you are typing.

Limits Pane.

Note Click the tabs to get help on the Property Editor.



Default values for the axes limits make sure that the maximum and minimum x and y values are displayed. If you want to override the default settings, change the values in the **Limits** pane fields. The **Auto-Scale** check box automatically clears if you click a different field. The new limits appear immediately in the response plot.

To reestablish the default values, select the **Auto-Scale** check box again.

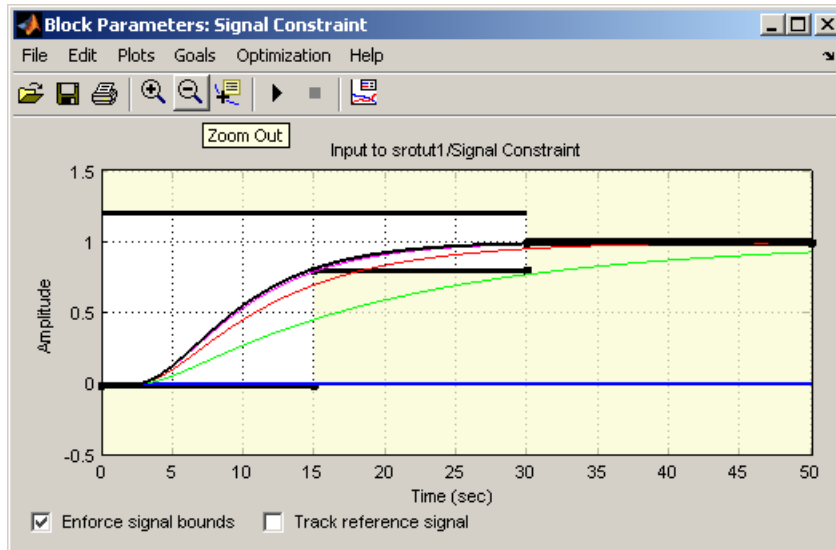
Running the Optimization

After you have specified constraints and the parameters to optimize, as described in “Specifying Design Requirements” on page 3-5 and “Specifying Parameters to Optimize” on page 3-19 respectively, you can run the optimization.

Run the optimization by selecting **Optimization > Start** in the Signal Constraint window, or click the **Start** button, which is the small triangle located on the control panel below the menus.

Simulink Design Optimization software uses optimization methods to find parameter values that allow a feasible solution, or best fit in the case of reference tracking, to the given constraints. Once the appropriate signals have been constrained with signal bounds or by tracking a reference signal, the tuned parameters set, and (optionally) any uncertain parameters and optimization settings specified, you are ready to run the optimization.

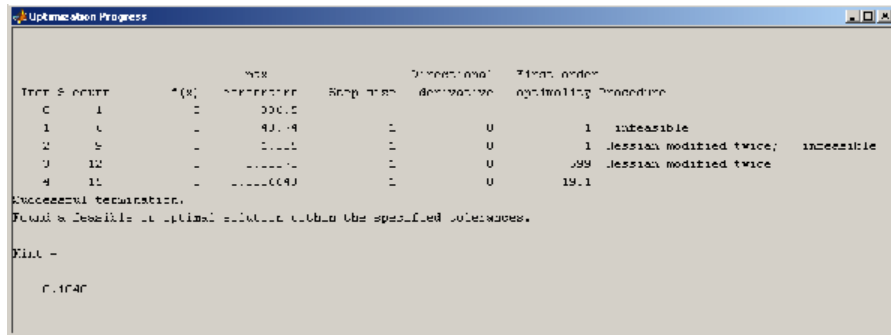
Simulink Design Optimization software begins by plotting the initial response in blue in the Signal Constraint window. During the optimization, intermediate responses are also plotted in various colors. The final response is plotted in black. If uncertainty is included in the optimization, the uncertain response signals are plotted as dashed lines, along with the nominal response as a solid line.



Simulink Design Optimization software changes the values of the tuned parameters within the MATLAB workspace and displays the final value in the Optimization Progress window. Alternatively, you can enter a parameter name at the MATLAB prompt to see its final value.

Note After the optimization, the values of the tuned parameters are changed to the new optimized values. This means that if you want to run another optimization, it uses these tuned values of the parameters as initial values, unless you specify alternative initial values in the Tuned Parameters dialog box. To revert to the unoptimized parameter values, select **Edit > Undo Optimize Parameters** from the Signal Constraint window.

The Optimization Progress window displays numerical output. The form of this output depends on the optimization method being used. To learn more, see “Selecting Optimization Methods” on page 3-28 and the discussion of **Display level** in “Selecting Additional Optimization Options” on page 3-30.



```

Optimization Progress
-----
Iter #   count   f(x)   step size   Hessian modified   Hessian modified   Hessian modified
1       1       -      93.14      1                 0                 1   infeasible
2       2       -      11.11      1                 0                 1   Hessian modified twice; infeasible
3       3       -      11.11      1                 0                 199  Hessian modified twice
4       4       -      11.11      1                 0                 19.1
Successful termination.
Found a feasible optimal solution within the specified tolerances.
Exit -
      0.1144
  
```

The Gradient descent optimization method may violate the bounds on parameter values when it cannot satisfy the signal constraints specified in the Signal Constraint block and the bounds on parameter values simultaneously. To learn how to troubleshoot this problem, see “Troubleshooting Optimization Results” on page 3-72

Note For more information on types of problems you may encounter using optimization solvers, see “Steps to Take After Running a Solver” in the Optimization Toolbox documentation.

If the optimization does not converge the first time, it often converges after adjusting the constraints or tuned parameter characteristics, or choosing different options. For more information, see “Troubleshooting Optimization Results” on page 3-72.

Optimizing Parameters for Model Robustness

In this section...

“What Is Model Robustness?” on page 3-42

“Sampling Methods for Computing Uncertain Parameter Values” on page 3-43

“How to Optimize Parameters for Model Robustness Using the GUI” on page 3-46

“Commands for Optimizing Parameters for Model Robustness” on page 3-49

“Example — Optimizing Parameters for Model Robustness Using the GUI” on page 3-49

What Is Model Robustness?

A model is *robust* when its response does not violate design requirements under parameter variations. When you optimize model parameters, your model may contain additional parameters whose values are not precisely known. Such parameters vary over a given range of values and are defined as *uncertain parameters*. You may know the nominal value and the range of values in which these uncertain parameters vary.

You can then use the Simulink Design Optimization software to incorporate the parameter uncertainty to test the robustness of your design. You can test and optimize parameters for model robustness in the following ways:

- **Before Optimization.** Specify the parameter uncertainty *before* you optimize the parameters to meet the design requirements. In this case, the optimization method optimizes the signals based on both nominal parameter values as well as the uncertain values. This mode requires more time.
- **After Optimization.** Specify the parameter uncertainty *after* you have optimized the model parameters to meet design requirements. You can then test the effect of the uncertain parameters by plotting the model's response. If the response violates the design requirements, you can optimize the parameters again by including the parameter uncertainty during the optimization.

To learn more, see “Example — Optimizing Parameters for Model Robustness Using the GUI” on page 3-49.

Note You cannot add uncertainty to controller or plant parameters when designing controllers using optimization-based methods in the SISO Design Tool.

Sampling Methods for Computing Uncertain Parameter Values

There are two sampling methods for computing uncertain parameter values. Both methods create several sample parameter values within the range of uncertainty, as described in the following sections:

- “**Random (Monte Carlo) Method**” on page 3-43
- “**Grid Method**” on page 3-45

To learn how to specify the sampling method, see the following sections:

- “How to Optimize Parameters for Model Robustness Using the GUI” on page 3-46
- `gridunc` and `randunc` function reference pages

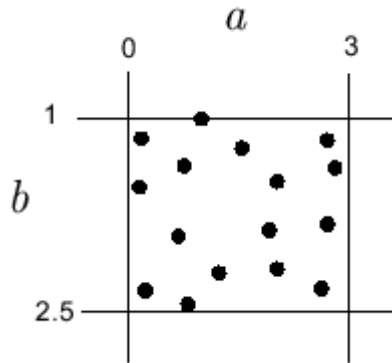
Random (Monte Carlo) Method

The **Random (Monte Carlo)** sampling method computes random values of uncertain parameters within a specified range. When you select the **Random (Monte Carlo)** method, you must also specify the following settings:

- Number of sample values
- Nominal parameter value
- Range of parameter values

When you specify more than one uncertain parameter, the sampling method creates random parameter values within a hypercube. This hypercube is defined by the minimum and maximum values of all uncertain parameters.

For example, the following figure shows two uncertain parameters, a and b , which range in value from 0 to 3 and 1 to 2.5 respectively. In the figure, the sample values appear as black dots and are scattered randomly within the rectangle.



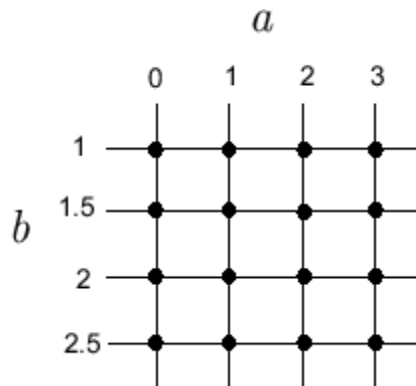
Grid Method

The **Grid** sampling method computes specified values of uncertain parameters within the range of uncertainty. When you select the **Grid** method, you must specify the following settings:

- Nominal parameter value
- Vector of sample parameter values

The sampling method uses the sample parameter values to compute the number of samples for the uncertain parameter.

When you specify more than one uncertain parameter, the sample values form a grid of combinations. For example, the following figure shows two uncertain parameters, a and b , with sample values [0 1 2 3] and [1 1.5 2 2.5]. In the figure, the sample values appear as black dots to form the grid.

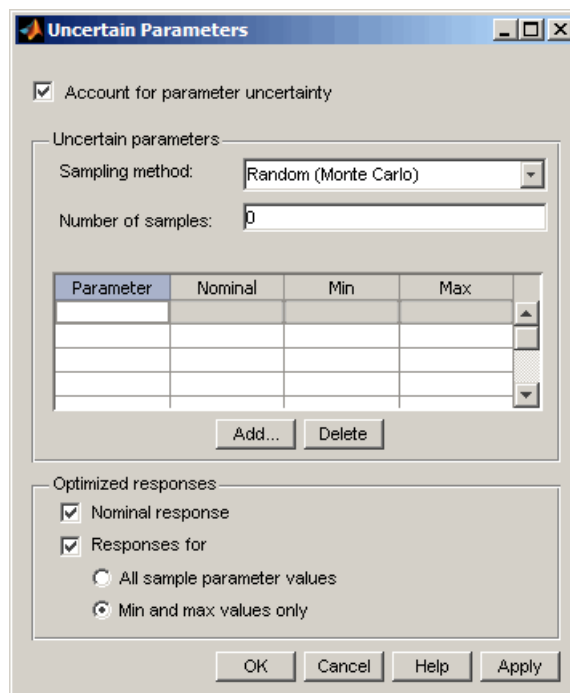


How to Optimize Parameters for Model Robustness Using the GUI

To optimize parameters for model robustness using the GUI:

- 1 In the Block Parameters: Signal Constraint window, select **Optimization > Uncertain Parameters**.

This action opens the Uncertain Parameters dialog box.



By default, the **Account for parameter uncertainty** check box is selected. This implies that the optimization method takes into account the parameter uncertainty during optimization. You can exclude the parameter uncertainty during optimization by clearing this option.

Note When you have more than one Signal Constraint block in your model, you only need to specify the uncertain parameters in one window. These settings apply to all constrained signals within the model.

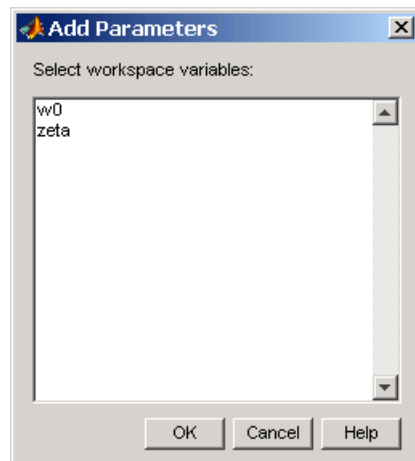
- 2 Select the sampling method from the **Sampling method** drop-down list.

To learn more about the sampling methods, see “Sampling Methods for Computing Uncertain Parameter Values” on page 3-43.

- 3 To add an uncertain parameter:

- a Click **Add** to open the Add Parameters dialog box.

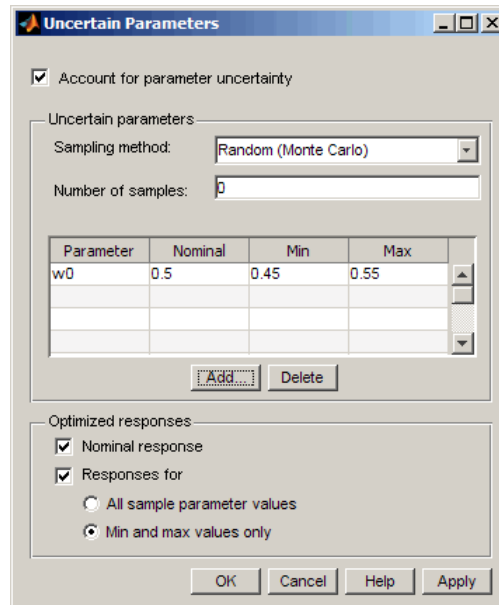
The dialog box lists all model parameters currently available in the MATLAB workspace.



Note Parameters that are already specified for optimization or as uncertain parameters do not appear in the Add Parameters dialog box.

- b** Select the parameters in the Add Parameters dialog box, and then click **OK**.

This action adds the parameters to the Uncertain Parameters dialog box.



For each parameter in the Uncertain Parameters dialog box, you can change the nominal, minimum and maximum values.

- 4** In the **Optimized responses** area of the GUI, configure the sample parameter values to use during optimization by selecting:
 - **Nominal response** check box to include the nominal values of the uncertain parameters
 - **All sample parameter values** check box to include all sample values of the uncertain parameters
 - **Min and max values only** check box to include only the minimum and maximum values of the uncertain parameters

Tip Using only the minimum and maximum values during optimization increases the computation speed.

- 5 Click **OK** to add the uncertain parameters to the response optimization project.

When you optimize the parameters for robustness, the optimization method uses the responses computed using all the uncertain parameter values to adjust the model parameters. For an example of testing and optimizing parameters for model robustness using the GUI, see “Example — Optimizing Parameters for Model Robustness Using the GUI” on page 3-49.

Commands for Optimizing Parameters for Model Robustness

You can also optimize parameters for model robustness by including parameter uncertainty at the command line. The following table summarizes the commands for model robustness. For detailed information about using each command, see the corresponding reference page.

Command	Purpose
setunc	Specify parameter uncertainty in response optimization project
gridunc	Sampling method for computing a grid of uncertain parameter values
randunc	Sampling method for computing random samples of uncertain parameter values

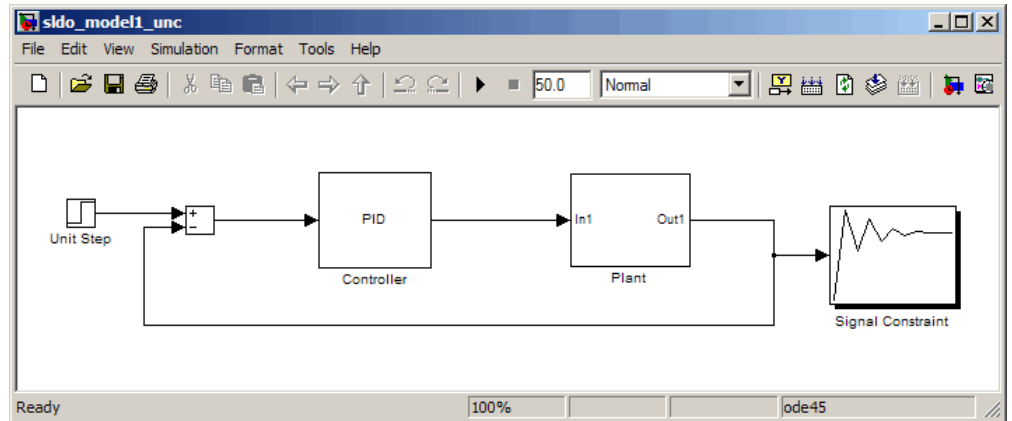
Example — Optimizing Parameters for Model Robustness Using the GUI

The following example shows how to optimize parameters for model robustness.

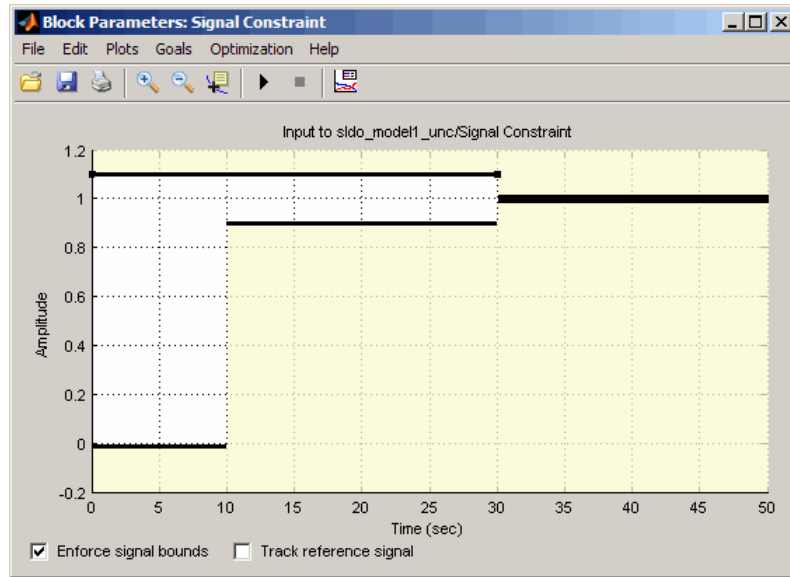
- 1 Open the Simulink model by typing the model name at the MATLAB prompt:

```
sldo_model1_desreq_optim
```

The following Simulink model opens.



The command also opens the Block Parameters: Signal Constraint window.



The Simulink model parameters have already been optimized to meet the following step response requirements:

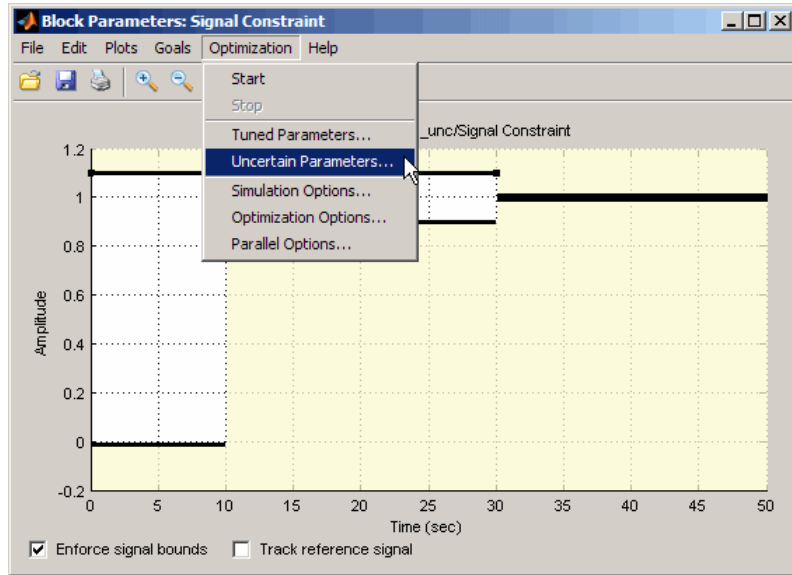
- Maximum overshoot of 10%
- Maximum rise time of 10 seconds
- Maximum settling time of 30 seconds

To learn how to optimize model parameters to meet design requirements, see “Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the GUI” in the *Simulink Design Optimization Getting Started Guide*.

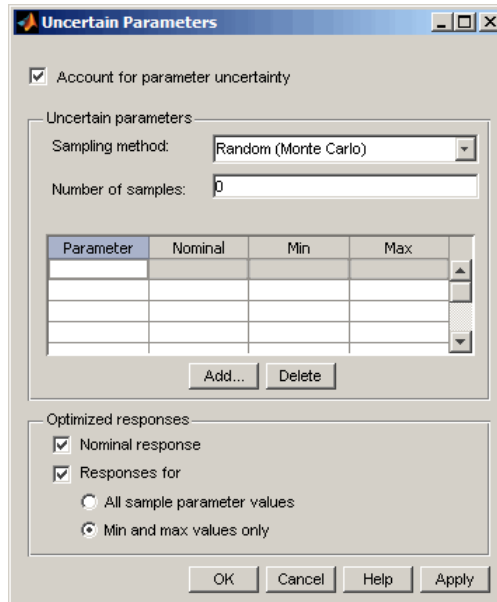
Tip To view the current response of the model, select **Plots > Plot Current Response** in the Block Parameters window.

2 To specify parameter uncertainty:

- In the Block Parameters window, select **Optimization > Uncertain Parameters**.

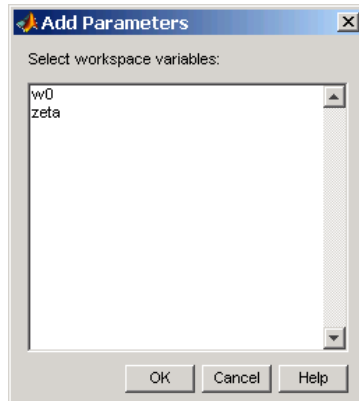


This action opens the Uncertain Parameters dialog box.



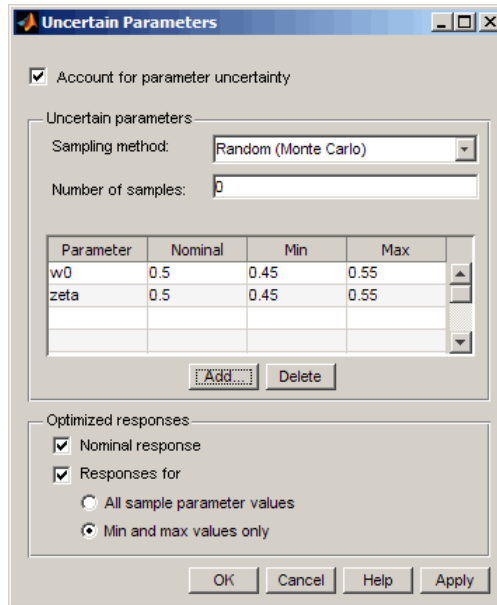
To learn more about the options in this dialog box, see “How to Optimize Parameters for Model Robustness Using the GUI” on page 3-46.

- b** Click **Add** to open the Add Parameters dialog box.



- c Select w_0 and zeta, and click **OK**.

This action adds the parameters to the Uncertain Parameters dialog box.

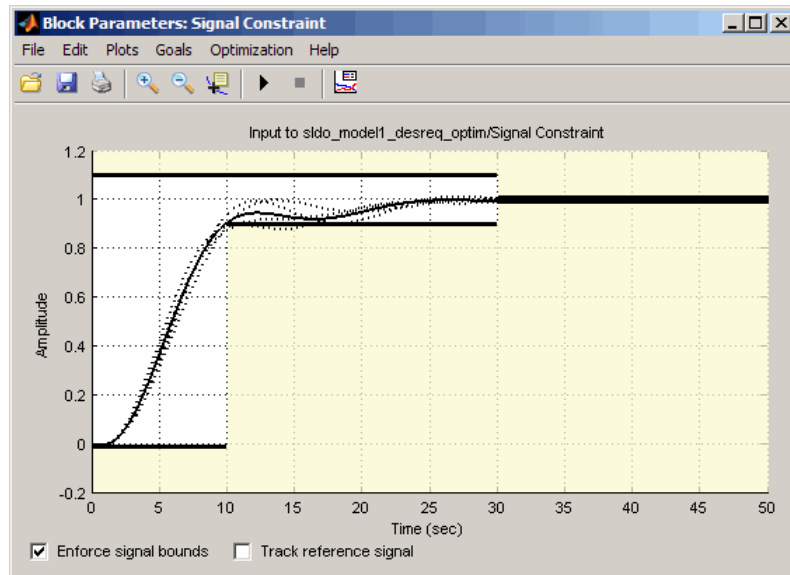


The **Nominal** column displays the nominal value of the parameters as specified in the Simulink model. The **Min** and **Max** columns specify the range in which the parameter can vary with respect to its nominal value. By default, the minimum and maximum parameter values vary by 10% of the nominal value.

- d Click **OK** to close the Uncertain Parameters dialog box.

- 3** To test the model robustness to the uncertain parameters, select **Plots > Plot Current Response** in the Signal Constraint block window.

The Block Parameters window updates, as shown in the following figure.



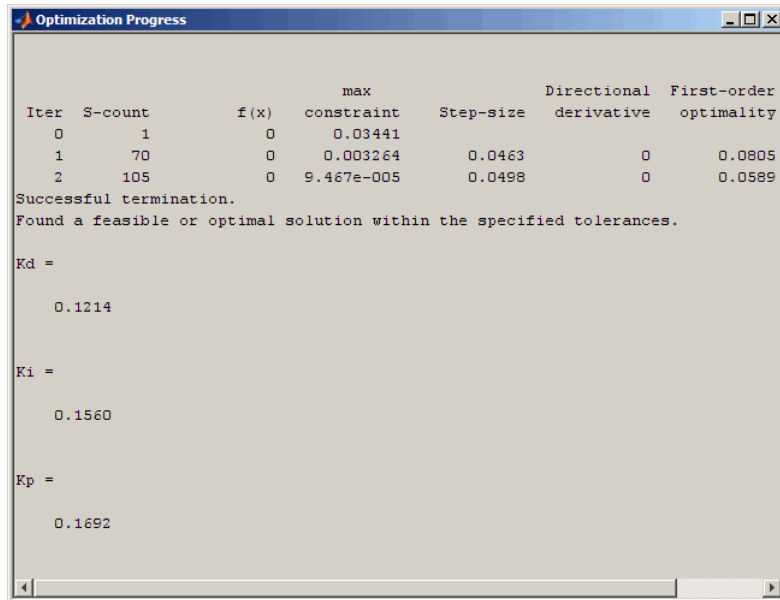
The window shows the following plot lines:

- The plot line shown as the solid black curve corresponds to the model's response computed using the optimized parameters and the nominal values of the uncertain parameter.
- The four plot lines shown as the dashed black curves correspond to the model's response with the minimum and maximum values of the uncertain parameters.

The dashed plot lines show that the model's response during the period of 10 to 15 seconds violates the design requirements.

- 4** To optimize the parameters for model robustness, select **Optimization > Start**.

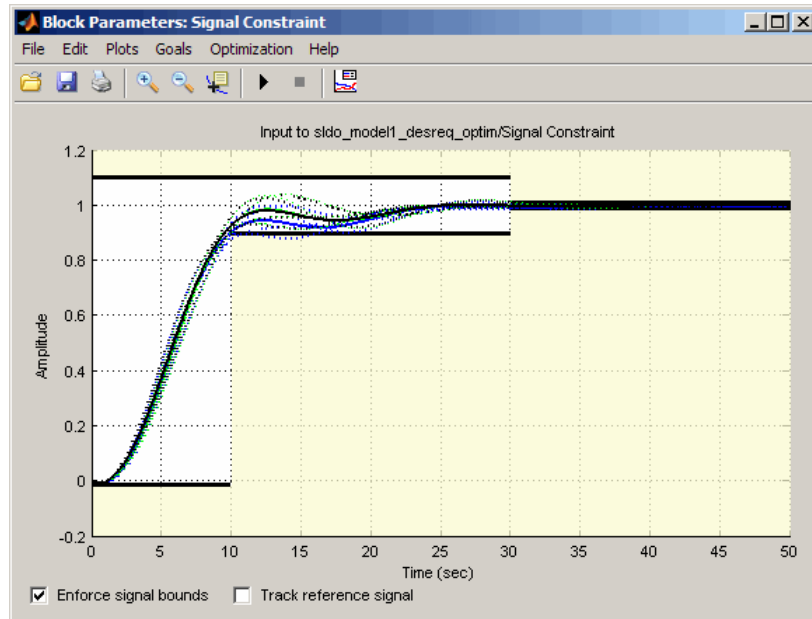
This action opens the Optimization Progress window, which displays the optimization iterations.



After the optimization completes, the message **Successful termination** indicates that the model's response meets all the specified design requirements. The Optimization Progress window also displays the optimized parameter values.

- 5** Examine the final response in the updated Block Parameters window.

The final response of the model appears as the solid black curve. The model's response with the uncertain parameter values now meets the design requirements.



Tip To view only the final response of the model, select **Plots > Clear Plots**. Then, select **Plots > Plot Current Response**.

Accelerating Model Simulations During Optimization

About Accelerating Optimization

You can accelerate the response optimization computations by changing the simulation mode of your Simulink model. Simulink Design Optimization software supports `Normal` and `Accelerator` simulation modes. For more information about these modes, see “Accelerating Models” in the Simulink documentation.

The default simulation mode is `Normal`. In this mode, Simulink uses interpreted code, rather than compiled C code during simulations.

In the `Accelerator` mode, Simulink Design Optimization software runs simulations during optimization with compiled C code. Using compiled C code speeds up the simulations and reduces the time to optimize the model response signals.

Limitations

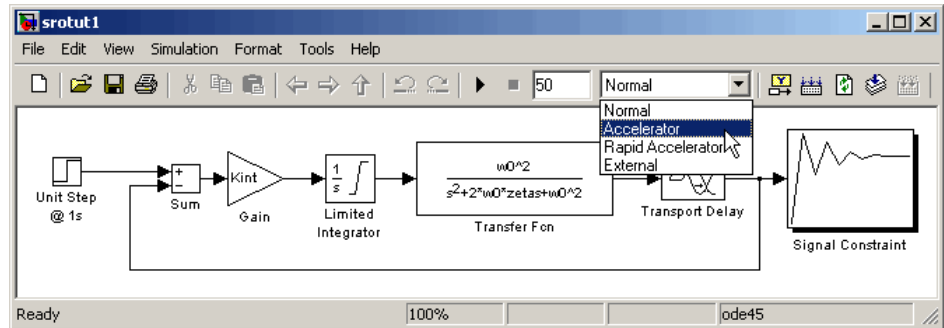
You cannot use the `Accelerator` mode if your model contains algebraic loops. If the model contains MATLAB function blocks, you must either remove them or replace them with Fcn blocks.

If the model structure changes during optimization, the model is compiled to regenerate the C code for each iteration. Using the `Accelerator` mode increases the computation time. To learn more about code regeneration, see “Code Regeneration in Accelerated Models” in the Simulink documentation.

Setting Accelerator Mode for Response Optimization

To set the simulation mode to `Accelerator`, open the Simulink model window and perform one of the following actions:

- Select **Simulation > Accelerator**.
- Choose `Accelerator` from the drop-down list as shown in the next figure.



Tip To obtain the maximum performance from the Accelerator mode, close all Scope blocks in your model.

Speeding Up Response Optimization Using Parallel Computing

In this section...

“When to Use Parallel Computing for Response Optimization” on page 3-60

“How Parallel Computing Speeds Up Optimization” on page 3-61

“Configuring Your System for Parallel Computing” on page 3-64

“Specifying Model Dependencies” on page 3-65

“How to Use Parallel Computing in the GUI” on page 3-66

“How to Use Parallel Computing at the Command Line” on page 3-70

When to Use Parallel Computing for Response Optimization

You can use Simulink Design Optimization software with Parallel Computing Toolbox software to speed up the time-domain response optimization of a Simulink model. Using parallel computing may reduce your model’s optimization time in the following cases:

- The model contains a large number of tuned parameters, and the Gradient descent method is selected for optimization.
- The Pattern search method is selected for optimization.
- The model contains a large number of uncertain parameters and uncertain parameter values.
- The model is complex and takes a long time to simulate.

When you use parallel computing, Simulink Design Optimization software distributes independent simulations to run them in parallel on multiple MATLAB sessions, also known as *workers*. Distributing the simulations significantly reduces the optimization time because the time required to simulate the model dominates the total optimization time. For more information on how the software distributes the simulations and the expected speedup, see “How Parallel Computing Speeds Up Optimization” on page 3-61.

The following sections describe how to configure your system, and use parallel computing:

- “Configuring Your System for Parallel Computing” on page 3-64
- “How to Use Parallel Computing in the GUI” on page 3-66
- “How to Use Parallel Computing at the Command Line” on page 3-70

How Parallel Computing Speeds Up Optimization

You can enable parallel computing with the Gradient descent and Pattern search optimization methods in the Simulink Design Optimization software. When you enable parallel computing, Simulink Design Optimization software distributes independent simulations during optimization on multiple MATLAB sessions. The following sections describe which simulations are distributed and the expected speedup using parallel computing:

- “Parallel Computing with the Gradient descent Method” on page 3-61
- “Parallel Computing with the Pattern search Method” on page 3-62

Parallel Computing with the Gradient descent Method

When you select Gradient descent as the optimization method, the model is simulated during the following computations:

- Constraint and objective value computation — One simulation per iteration
- Constraint and objective gradient computations — Two simulations for every tuned parameter per iteration
- Line search computations — Multiple simulations per iteration

The total time, T_{total} , taken per iteration to perform these simulations is given by the following equation:

$$T_{total} = T + (N_p \times (2 \times T)) + (N_{ls} \times T) = T \times (1 + (2 \times N_p) + N_{ls})$$

where T is the time taken to simulate the model and is assumed to be equal for all simulations, N_p is the number of tuned parameters, and N_{ls} is the number of line searches.

When you use parallel computing, Simulink Design Optimization software distributes the simulations required for constraint and objective gradient computations. The simulation time taken per iteration when the gradient computations are performed in parallel, T_{totalP} , is approximately given by the following equation:

$$T_{totalP} = T + \left(\text{ceil} \left(\frac{N_p}{N_w} \right) \times 2 \times T \right) + (N_{ls} \times T) = T \times \left(1 + 2 \times \text{ceil} \left(\frac{N_p}{N_w} \right) + N_{ls} \right)$$

where N_w is the number of MATLAB workers.

Note The equation does not include the time overheads associated with configuring the system for parallel computing and loading Simulink software on the remote MATLAB workers.

The expected speedup for the total optimization time is given by the following equation:

$$\frac{T_{totalP}}{T_{total}} = \frac{1 + 2 \times \text{ceil} \left(\frac{N_p}{N_w} \right) + N_{ls}}{1 + (2 \times N_p) + N_{ls}}$$

For example, for a model with $N_p=3$, $N_w=4$, and $N_{ls}=3$, the expected speedup

$$\text{equals } \frac{1 + 2 \times \text{ceil} \left(\frac{3}{4} \right) + 3}{1 + (2 \times 3) + 3} = 0.6 .$$

For a demo on the performance improvement achieved with the Gradient descent method, see the Improving Optimization Performance Using Parallel Computing demo.

Parallel Computing with the Pattern search Method

The Pattern search optimization method uses search and poll sets to create and compute a set of candidate solutions at each optimization iteration.

The total time, T_{total} , taken per iteration to perform these simulations, is given by the following equation:

$$T_{total} = (T \times N_p \times N_{ss}) + (T \times N_p \times N_{ps}) = T \times N_p \times (N_{ss} + N_{ps})$$

where T is the time taken to simulate the model and is assumed to be equal for all simulations, N_p is the number of tuned parameters, N_{ss} is a factor for the search set size, and N_{ps} is a factor for the poll set size.

When you use parallel computing, Simulink Design Optimization software distributes the simulations required for the search and poll set computations, which are evaluated in separate parfor loops. The simulation time taken per iteration when the search and poll sets are computed in parallel, T_{totalP} , is given by the following equation:

$$\begin{aligned} T_{totalP} &= (T \times \text{ceil}(N_p \times \frac{N_{ss}}{N_w})) + (T \times \text{ceil}(N_p \times \frac{N_{ps}}{N_w})) \\ &= T \times (\text{ceil}(N_p \times \frac{N_{ss}}{N_w}) + \text{ceil}(N_p \times \frac{N_{ps}}{N_w})) \end{aligned}$$

where N_w is the number of MATLAB workers.

Note The equation does not include the time overheads associated with configuring the system for parallel computing and loading Simulink software on the remote MATLAB workers.

The expected speed up for the total optimization time is given by the following equation:

$$\frac{T_{totalP}}{T_{total}} = \frac{\text{ceil}(N_p \times \frac{N_{ss}}{N_w}) + \text{ceil}(N_p \times \frac{N_{ps}}{N_w})}{N_p \times (N_{ss} + N_{ps})}$$

For example, for a model with $N_p=3$, $N_w=4$, $N_{ss}=15$, and $N_{ps}=2$, the expected

$$\text{speedup equals } \frac{\text{ceil}(3 \times \frac{15}{4}) + \text{ceil}(3 \times \frac{2}{4})}{3 \times (15 + 2)} = 0.27.$$

Note Using the `Pattern` search method with parallel computing may not speed up the optimization time. To learn more, see “Why do I not see the optimization speedup I expected using parallel computing?” on page 3-78 in the “Troubleshooting Optimization Results” on page 3-72 section.

For a demo on the performance improvement achieved with the `Pattern` search method, see the Improving Optimization Performance Using Parallel Computing demo.

Configuring Your System for Parallel Computing

To use parallel computing, you must first configure your system as described in the following sections:

- “Configuring Parallel Computing on Multicore Processors” on page 3-64
- “Configuring Parallel Computing on Multiprocessor Networks” on page 3-64

After you configure your system for parallel computing, you can use the GUI or the command-line functions to optimize the model’s response using parallel computing.

Configuring Parallel Computing on Multicore Processors

With a basic Parallel Computing Toolbox license, you can establish a pool of up to four parallel MATLAB sessions in addition to the MATLAB client.

To start a pool of four MATLAB sessions in local configuration, type the following at the MATLAB prompt:

```
matlabpool open local
```

To learn more, see the `matlabpool` reference page in the Parallel Computing Toolbox documentation.

Configuring Parallel Computing on Multiprocessor Networks

To use parallel computing on a multiprocessor network, you must have the Parallel Computing Toolbox software and the MATLAB Distributed

Computing Server software. To learn more, see the Parallel Computing Toolbox and MATLAB Distributed Computing Server documentation.

To configure a multiprocessor network for parallel computing:

- 1 Create a user configuration file to include any model file dependencies, as described in “Defining Configurations” and FileDependencies reference page in the Parallel Computing Toolbox documentation.
- 2 Open the pool of MATLAB workers using the user configuration file, as described in “Applying Configurations in Client Code” in the Parallel Computing Toolbox documentation.

Opening the pool allows the remote workers to access the file dependencies included in the user configuration file.

Specifying Model Dependencies

Model dependencies are files, such as referenced models, data files and S-functions, without which a model cannot run. When you use parallel computing, Simulink Design Optimization software helps you identify model path dependencies. To do so, the software uses the Simulink Manifest Tools. The dependency analysis may not find all the files required by your model. For example, folder paths that contain code for your model or block callback. To learn more, see the “Scope of Dependency Analysis” in the Simulink documentation.

Before you start the optimization using parallel computing, verify that the response optimization project includes all model dependencies and the remote workers can access them.

Note The optimization errors out if the response optimization project does not contain all the model dependencies or the remote workers cannot access the dependencies.

You can add additional dependencies and make the dependencies accessible to remote workers, as described in the following sections:

- “Specifying Additional Path and File Dependencies” on page 3-66

- “Making Model Dependencies Accessible to Remote Workers” on page 3-66

Specifying Additional Path and File Dependencies

If your model has path and file dependencies that the software cannot find automatically, add the dependencies before you start the optimization using parallel computing:

- 1 Add the path dependencies using the GUI or at the command line, as described “How to Use Parallel Computing in the GUI” on page 3-66, and “How to Use Parallel Computing at the Command Line” on page 3-70.
- 2 Add the file dependencies, as described in “Configuring Parallel Computing on Multiprocessor Networks” on page 3-64.

Making Model Dependencies Accessible to Remote Workers

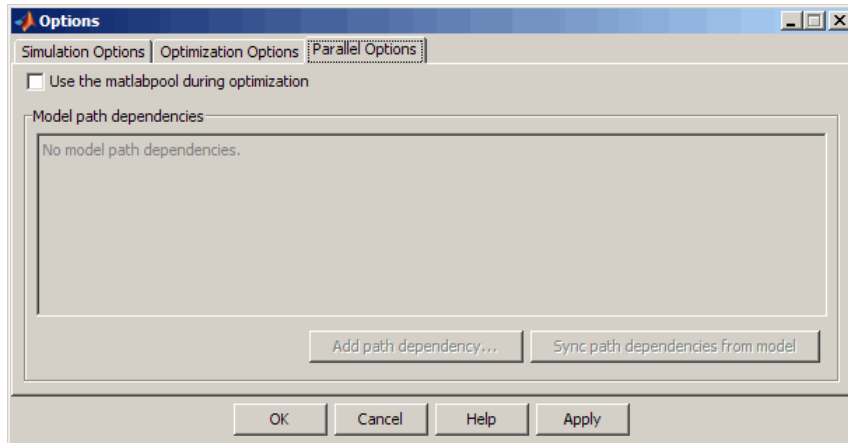
If your model has dependencies that the remote MATLAB workers cannot access directly, modify the list of dependencies to make them accessible. For example, remote workers cannot access model dependencies on your local drive. Update the list of paths so that the workers can access them, as described in “How to Use Parallel Computing in the GUI” on page 3-66, and “How to Use Parallel Computing at the Command Line” on page 3-70.

How to Use Parallel Computing in the GUI

After you configure your system for parallel computing, as described in “Configuring Your System for Parallel Computing” on page 3-64, you can use the GUI to optimize your model’s response:

- 1 Open the model that you want to optimize.
- 2 Configure the response optimization project for your model.

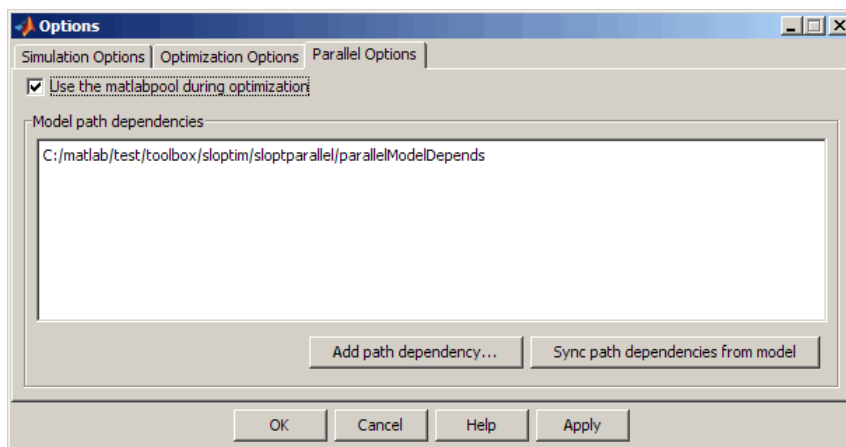
- 3 In the Signal Constraint block, select **Optimization > Parallel Options** to open the **Parallel Options** tab.



4 Select the **Use the matlabpool during optimization** option.

This action checks for model path dependencies in your Simulink model and displays the path dependencies in the **Model path dependencies** list box.

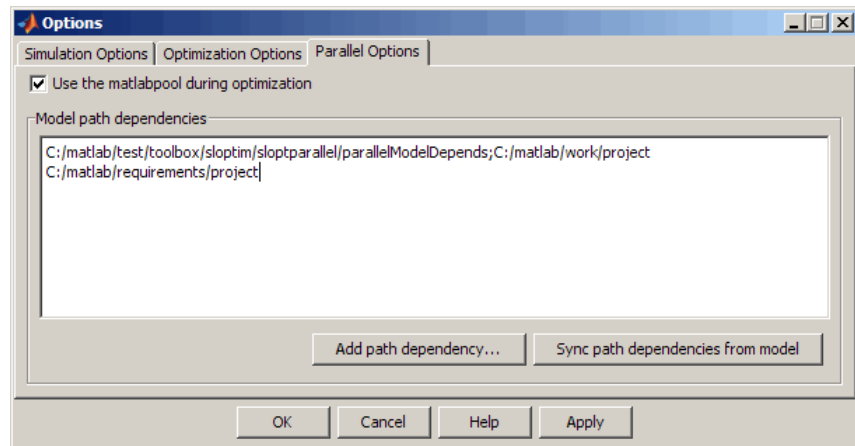
Note As described in “Specifying Model Dependencies” on page 3-65, the automatic path dependencies check may not detect all the path dependencies in your model.



5 (Optional) Add the path dependencies that the automatic check does not detect to the response optimization project.

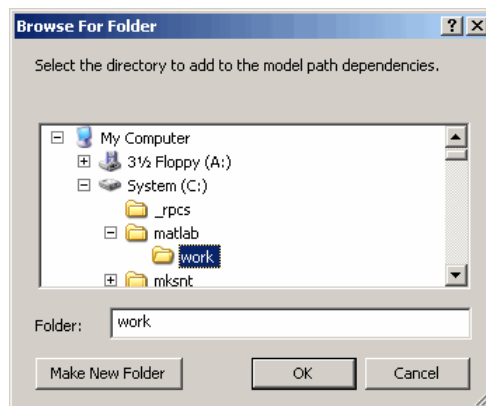
- a Specify the paths in the **Model path dependencies** list box.

You can specify the paths separated with a semicolon, or on a new line.



- b Click **Apply** to include the new paths in the response optimization project.

Alternatively, you can click **Add path dependency** to open a Browse For Folder dialog box where you can select the folder to add.



- 6 (Optional) In the **Model path dependencies** list box, update the paths on local drives to make them accessible to remote workers. For example, change `C:\` to `\\\\hostname\C$\`.
- 7 (Optional) If you modify the Simulink model such that it introduces a new path dependency, then you must resync the path dependencies. Click **Sync path dependencies from model** in the **Parallel Options** tab to rerun the automatic dependency check for your model.

This action updates the **Model path dependencies** list box with any new path dependency found in the model.

- 8 Click **OK**.
- 9 In the Signal Constraint block window, select **Optimization > Start** to optimize the model response using parallel computing.

For more information on how to troubleshoot problems that occur during optimization using parallel computing, see “Optimization Speed and Parallel Computing” on page 3-76.

How to Use Parallel Computing at the Command Line

After you configure your system for parallel computing, as described in “Configuring Your System for Parallel Computing” on page 3-64, you can optimize your model’s response using the command-line functions:

- 1 Open the model that you want to optimize.
- 2 Configure a response optimization project, `proj`.
- 3 Enable the parallel computing option in the response optimization project by typing the following command:

```
optimset(proj, 'UseParallel', 'always');
```

- 4 Find the model path dependencies by typing the `finddepend` command.

```
dirs=finddepend(proj)
```

This command returns the model path dependencies in your Simulink model.

Note As described in “Specifying Model Dependencies” on page 3-65, the `finddepend` command may not detect all the path dependencies in your model.

- 5** (Optional) Modify `dirs` to include the model path dependencies that `finddepend` does not detect.

```
dirs=vertcat(dirs,'\\hostname\C$\matlab\work')
```

- 6** (Optional) Modify `dirs` to make paths on local drives accessible to remote workers.

```
dirs = regexprep(dirs,'C:/', '\\\\hostname\C$\')
```

- 7** Add the updated path dependencies to the response optimization project, `proj` by typing the following command:

```
optimset(proj, 'ParallelPathDependencies', dirs)
```

- 8** Run the optimization by typing the following command:

```
optimize(proj)
```

For more information on how to troubleshoot problems that occur during optimization using parallel computing, see “Optimization Speed and Parallel Computing” on page 3-76 in “Troubleshooting Optimization Results” on page 3-72.

Refining and Troubleshooting Optimization Results

Troubleshooting Optimization Results

When optimizing the model parameters, the optimization method may run into issues. Simulink Design Optimization software provides visual cues to inform you about issues during the progress of an optimization. The following list represents the commonly encountered problems, and recommends solutions, advice, and tips to help you troubleshoot the issue.

- “Optimization Does Not Make Progress” on page 3-72
- “Optimization Convergence” on page 3-73
- “Optimization Speed and Parallel Computing” on page 3-76
- “Undesirable Parameter Values” on page 3-79
- “Reverting to Initial Parameter Values” on page 3-80

Optimization Does Not Make Progress

- “Should I worry about the scale of my responses and how constraints and design requirements are discretized?” on page 3-72
- “Why don’t the responses and parameter values change at all?” on page 3-72
- “Why does the optimization stall?” on page 3-73

Should I worry about the scale of my responses and how constraints and design requirements are discretized?.

No. Simulink Design Optimization software automatically normalizes constraints, design requirement and response data. Unlike its predecessor, the Nonlinear Control Design Blockset software, it does not discretize the constraints or design requirements.

Why don’t the responses and parameter values change at all?.

- The optimization problem you formulated might be nonsmooth. This means that small parameter changes have no effect on the amount by which response signals satisfy or violate the constraints and only large changes will make a difference. Try switching to a search-based method such as

simplex search or pattern search. Alternatively, look for initial guesses outside of the dead zone where parameter changes have no effect. If you are directly optimizing the response of a Simulink model using a Signal Constraint block, you could also try removing nonlinear blocks such as the Quantizer or Dead Zone block.

- If you are using the **Refined** option for **Gradient type** with the gradient descent method, try the **Basic** option for **Gradient type** instead. The gradient model that the **Refined** option uses might be invalid for your problem.

Why does the optimization stall? When using a Signal Constraint block to directly optimize a Simulink model, certain parameter combinations can make the simulation stall for models with strong nonlinearities or frequent mode switching. In these cases, the ODE solvers take smaller and smaller step sizes. Stalling can also occur when the model's ODEs become too stiff for some parameter combinations. A symptom of this behavior is when the Simulink model status is **Running** and clicking the **Stop** button fails to interrupt the optimization. When this happens, you can try one of the following solutions:

- Switch to a different ODE solver, especially one of the stiff solvers.
- Specify a minimum step size.
- Disable zero crossing detection if chattering is occurring.
- Tighten the lower and upper bounds on parameters that cause simulation difficulties. In particular, eliminate regions of the parameter space where some model assumptions are invalid and the model behavior can become erratic.

Optimization Convergence

- “What to do if the optimization does not get close to an acceptable solution?” on page 3-74
- “Why does the optimization terminate before exceeding the maximum number of iterations, with a solution that does not satisfy all the constraints or design requirements?” on page 3-74
- “What to do if the optimization takes a long time to converge even though it is close to a solution?” on page 3-75

- “What to do if the response becomes unstable and does not recover?” on page 3-75

What to do if the optimization does not get close to an acceptable solution?.

- If you’re using gradient descent, the default method, try the **Refined** option for **Gradient type**. This option yields more accurate gradient estimates when using variable-step solvers and can facilitate convergence.
- If you are using pattern search, check that you have specified appropriate maximum and minimum values for all your tuned parameters or compensator elements. The pattern search method looks inside these bounds for a solution. When they are set to their default values of Inf and -Inf, the method searches within $\pm 100\%$ of the initial values of the parameters. In some cases this region is not large enough and changing the maximum and minimum values can expand the search region.
- Your optimization problem might have local minima. Consider running one of the search-based methods first to get closer to an acceptable solution.
- Reduce the number of tuned parameters and compensator elements by removing from the **Tuned parameters** list (when using a Signal Constraint block) or from the **Compensators** pane (when using a SISO Design Task) those parameters that you know only mildly influence the optimized responses. After you identify reasonable values for the key parameters, add the fixed parameters back to the tunable list and restart the optimization using these reasonable values as initial guesses.

Why does the optimization terminate before exceeding the maximum number of iterations, with a solution that does not satisfy all the constraints or design requirements?.

- It might not be possible to achieve your specifications. Try relaxing the constraints or design requirements that the response signals violate the most. After you find an acceptable solution to the relaxed problem, tighten some constraints again and restart the optimization.
- The optimization might have converged to a local minimum that is not a feasible solution. Restart the optimization from a different initial guess and/or use one of the search-based methods to identify another local minimum that satisfies the constraints.

What to do if the optimization takes a long time to converge even though it is close to a solution?

- In a Signal Constraint window, use the Stop button, or select **Optimization > Stop**, to interrupt the optimization when you think the current optimized response signals are acceptable.

When you use a SISO Design Task, click **Stop Optimization** in the **Optimization** panel of the **Response Optimization** node in the Control and Estimation Tools Manager, when you think the current optimized response signals are acceptable.

- If you use the gradient descent method, try restarting the optimization. Restarting resets the Hessian estimate and might speed up convergence.
- Increase the convergence tolerances in the Optimization Options dialog to force earlier termination.
- Relax some of the constraints or design requirements to increase the size of the feasibility region.

What to do if the response becomes unstable and does not recover?.

While the optimization formulation has explicit safeguards against unstable or divergent response signals, the optimization can sometimes venture into an unstable region where simulation results become erratic and gradient methods fail to find a way back to the stable region. In these cases, you can try one of the following solutions:

- Add or tighten the lower and upper bounds on compensator element and parameter values. Instability often occurs when you allow some parameter values to become too large.
- Use a search-based method to find parameter values that stabilize the response signals and then start the gradient-based method using these initial values.
- When optimizing responses in a SISO Design Task, you can try adding additional design requirements that achieve the same or similar goal. For example, in addition to a settling time design requirement on a step response plot, you could add a settling time design requirement on a root-locus plot that restricts the location of the real parts of the poles. By adding overlapping design requirements in this way, you can force the optimization to meet the requirements.

Optimization Speed and Parallel Computing

- “How can I speed up the optimization?” on page 3-76
- “Why are the optimization results with and without using parallel computing different?” on page 3-77
- “Why do I not see the optimization speedup I expected using parallel computing?” on page 3-78
- “Why does the optimization using parallel computing not make any progress?” on page 3-78
- “Why do I receive an error “Cannot save model tpe5468c55_910c_4275_94ef_305e2eeeeeef4”?” on page 3-78
- “Why does the optimization using parallel computing not stop when I click the **Stop optimization** button?” on page 3-79

How can I speed up the optimization?.

- The optimization time is dominated by the time it takes to simulate the model. When using a Signal Constraint block to directly optimize a Simulink model, you can enable the Accelerator mode using **Simulation > Accelerator** in the Simulink model window, to dramatically reduce the optimization time.

Note The Rapid Accelerator mode in Simulink software is not supported for speeding up the optimization. For more information, see “Accelerating Model Simulations During Optimization” on page 3-58.

- The choice of ODE solver can also significantly affect the overall optimization time. Use a stiff solver when the simulation takes many small steps, and use a fixed-step solver when such solvers yield accurate enough simulations for your model. (These solvers must be accurate in the entire parameter search space.)
- Reduce the number of tuned compensator elements or parameters and constrain their range to narrow the search space.

- When specifying parameter uncertainty (not available when optimizing responses in a SISO Design Task), keep the number of sample values small since the number of simulations grows exponentially with the number of samples. For example, a grid of 3 parameters with 10 sample values for each parameter requires $10^3=1000$ simulations per iteration.

Why are the optimization results with and without using parallel computing different?

- When you use parallel computing, different numerical precision on the client and worker machines can produce marginally different simulation results. Thus, the optimization method takes a completely different solution path and produces a different result.

Note Numerical precision can differ because of different operating systems or hardware on the client and worker machines.

- When you use parallel computing, the state of the model on the client and the worker machines can differ, and thus lead to a different result. For example, the state can become different if you change a parameter value initialized by a callback function on the client machine *after* the workers have loaded the model. The model parameter values on the workers and the client are now out of sync, which can lead to a different result.

After you change the model parameter values initialized by a callback function, verify that the parameters exist in the model workspace or update the callback function so that the remote workers have access to the changed parameter values.

- When you use parallel computing with the `Pattern` search method, the `Pattern` search method searches for a candidate solution more comprehensively than when you do not use parallel computing. This more comprehensive search can result in a different solution. To learn more, see “Parallel Computing with the `Pattern` search Method” on page 3-62.

Why do I not see the optimization speedup I expected using parallel computing?.


- When you optimize a model that does not have a large number of parameters or does not take long to simulate, the resulting optimization time may not be any faster. In such cases, the overheads associated with creating and distributing the parallel tasks outweighs the benefits of running the simulations during optimization in parallel.
- Using `Pattern` search method with parallel computing may not speed up the optimization time. When you do not use parallel computing, the method stops searching for a candidate solution at each iteration as soon as it finds a solution better than the current solution. The candidate solution search is more comprehensive when you use parallel computing. Although the number of iterations may be larger, the optimization without using parallel computing may be faster.

To learn more about the expected speedup, see “Parallel Computing with the `Pattern` search Method” on page 3-62.

Why does the optimization using parallel computing not make any progress?. In some cases, the gradient computations on the remote worker machines may silently error out when you use parallel computing. In such cases, the Optimization Progress window shows that the $f(x)$ and `max` constraint values do not change, and the optimization terminates after two iterations with the message `Unable to satisfy constraints`. To troubleshoot the problem:

- 1 Run the optimization for a few iterations without parallel computing to see if the optimization progresses.
- 2 Check if the remote workers have access to all model dependencies. To learn more, see “Making Model Dependencies Accessible to Remote Workers” on page 3-66.

Why do I receive an error “Cannot save model tpe5468c55_910c_4275_94ef_305e2eeeeef4”?. When you select `Refined` as the `Gradient type`, the software may error out when it saves a temporary model to a nonwriteable folder, and then displays this error message. Change the `Gradient type` to `Basic` to clear this error. To learn more, see “Gradient Type” on page 3-31.

Why does the optimization using parallel computing not stop when I click the Stop optimization button? When you use parallel computing, the software has to wait till the current iteration completes before it notifies the workers to stop the optimization. The optimization does not terminate immediately when you click the **Stop optimization** button , and appears to continue to run.

Undesirable Parameter Values

- “What to do if the optimization drives the tuned compensator elements and parameters to undesirable values?” on page 3-79
- “What to do if the optimization violates bounds on parameter values?” on page 3-79

What to do if the optimization drives the tuned compensator elements and parameters to undesirable values?.

- When a tuned compensator element or parameter is positive, or when its value is physically constrained to a given range, enter the lower and upper bounds (**Minimum** and **Maximum**) in one of the following:
 - Tuned Parameters dialog box (from a Signal Constraint block)
 - **Compensators** pane (in a SISO Design Task)

This information helps guide the optimization method towards a reasonable solution.

- In the Tuned Parameters dialog box (from a Signal Constraint block) or the **Compensators** pane (in a SISO Design Task), specify initial guesses that are within the range of desirable values.
- In the **Compensators** pane in a SISO Design Task, verify that no integrators/differentiators are selected for optimization. Optimizing the pole/zero location of integrators/differentiators can result in drastic changes in the system gain and lead to undesirable values.

What to do if the optimization violates bounds on parameter values?. The Gradient descent optimization method `fmincon` violates the parameter bounds when it cannot simultaneously satisfy the signal constraints and the bounds. When this happens, try one of the following:

- Specify a different value for the parameter bound and restart the optimization. A guideline is to adjust the bound by 1% of the typical value. For example, for a parameter with a typical value of 1 and lower bound of 0, change the lower bound to 0.01.
- Relax the signal constraints and restart the optimization. This approach results in a different solution path for the Gradient descent method.
- Restart the optimization immediately after it terminates by using the **Start optimization** button in the Signal Constraint window. This approach uses the previous optimization results as the starting point for the next optimization cycle to refine the results.
- Use the following two-step approach to perform the optimization:
 - 1** Run an initial optimization to satisfy the signal constraints.

For example, run the optimization using the Simplex search method. This method satisfies the signal constraints but does not support the bounds on parameter values. The results obtained using this method provide the starting point for the optimization performed in the next step. To learn more about this method, see the `fminsearch` function reference page in the Optimization Toolbox documentation.
 - 2** Reconfigure the optimization by selecting a different optimization method to satisfy both the signal constraints and the parameter bounds.

For example, change the optimization method to Gradient descent and run the optimization again.

Tip If Global Optimization Toolbox software is installed, you can select the Pattern search optimization method to optimize the model response.

Reverting to Initial Parameter Values

How do I quit an optimization and revert to my initial parameter values?.

- When using a Signal Constraint block in a Simulink model, click the **Stop** button or select **Optimization > Stop** in the Signal Constraint window

to stop the optimization. To revert to your initial parameter values, select **Edit > Undo Optimize Parameters**.

- When using a SISO Design Task, the **Start Optimization** button becomes a **Stop Optimization** button after the optimization has begun. To quit the optimization, click the **Stop Optimization** button. To revert to the initial parameter values, select **Edit > Undo Optimize compensators** from the menu in the SISO Design Tool window.

Saving and Loading Response Optimization Projects

In this section...

“Saving Response Optimization Projects” on page 3-82

“Saving Additional Settings” on page 3-85

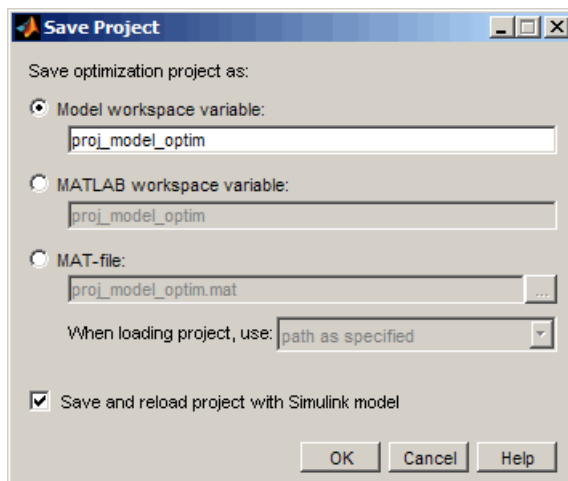
“Reloading Response Optimization Projects” on page 3-86

Saving Response Optimization Projects

Saving a response optimization project lets you reuse your settings during a later session. These settings include constraint bounds, tuned and uncertain parameters, and settings for optimization and simulation. The software saves the settings from *all* Signal Constraint blocks in the model in a single project.


You can save the response optimization project as a workspace variable or a MAT-file:

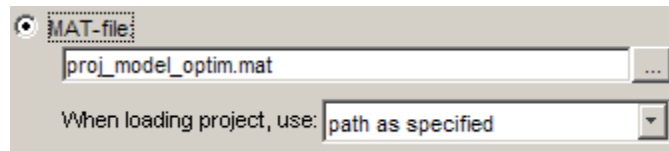
- 1 Select **File** > **Save** in a Signal Constraint window. This action opens the Save Project dialog box.



- 2 Save the project using one of the following options in the Save Project dialog box:

- **Model workspace variable:** Specify a variable name. This option saves the project in the model workspace.
- **MATLAB workspace variable:** Specify a variable name. The project no longer exists after you terminate the MATLAB session.
- **MAT-file:** Type a file name. For example, `myproject.mat`, which saves the project in the current MATLAB folder.

To specify a different location for the MAT-file, type the path or click  to select a folder. Optionally, you can specify how the software searches for the path when loading the project in the **When loading project, use** drop-down list.



By default, the software uses the path specified in the **MAT-file** edit field. For more path options, see “Path Options for Project MAT-files” on page 3-84.

- 3** Verify that the **Save and reload project with Simulink model** check box is selected.

This option automatically loads the project when you reopen the Simulink model. If the software cannot find the project, a warning message appears.

- 4** Click **OK** to save the project.

To save the response optimization project using a new file name, select **File > Save As** from a Signal Constraint window. Then, follow the preceding instructions.

Path Options for Project MAT-files

When loading a response optimization project, the software requires the path of the project MAT-file. You can specify this information in the **When loading project**, use drop-down list, using one of the following options:

Path Option	Path Searched When Loading a Project	Example
path as specified (Default)	Specified in the MAT-file edit field.	Current MATLAB folder: C:\work Model location: C:\work\models Project location: C:\work\projects\proj_model_optim.mat Path searched: C:\work\projects\proj_model_optim.mat
path relative to model folder	Relative to the location of the Simulink model.	Current MATLAB folder: C:\work Model location: C:\work\models Project location: C:\work\projects\proj_model_optim.mat Path searched: ..\projects\proj_model_optim.mat

Path Option	Path Searched When Loading a Project	Example
path relative to current folder	Relative to the current MATLAB folder.	Current MATLAB folder: C:\work Model location: C:\work\models Project location: C:\work\projects\proj_model_optim.mat Path searched: projects\proj_model_optim.mat
no path (file name only)	Current MATLAB folder.	Current MATLAB folder: C:\work Model location: C:\work\models Project location: C:\work\projects\proj_model_optim.mat Path searched: proj_model_optim.mat

Saving Additional Settings

In addition to settings that you save with the response optimization project, the software automatically saves several other settings with the model. These settings include the following:

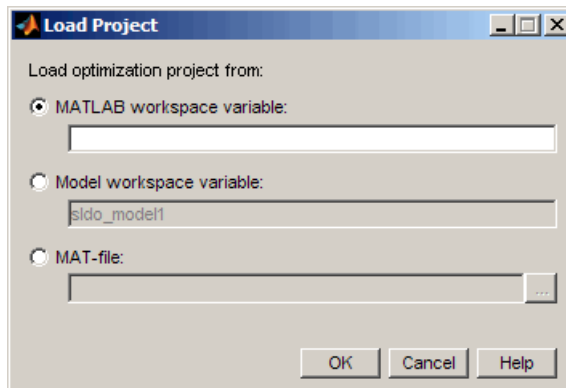
- The position of the Signal Constraint window on the screen
- The axis limit settings within the Signal Constraint window

- The location where the response optimization project, either a workspace variable or a MAT-file, is saved
- The name of the response optimization project

You must save the Simulink model to retain the settings when reloading the model in a subsequent session. To save the model, select **File > Save** in the model window.

Reloading Response Optimization Projects

To reload a response optimization project from the MATLAB workspace, model workspace, or a file, select **File > Load** in a Signal Constraint window in the model. In the Load Project dialog box (shown next), enter the name of the MATLAB workspace variable, model workspace variable, or MAT-file that contains the project, and then click **OK**. Alternatively, you can load the project from an existing file by clicking the button to the right of **MAT-file** and selecting a file from the folder.



Although the load command is issued from a single Signal Constraint window, the constraints are loaded into *all* Signal Constraint blocks in the model. Additionally, tuned parameters, uncertain parameters, and optimization and simulation setup options are loaded into the model.

Note Loading a project cannot be undone.

Optimizing Parameters at the Command Line

In this section...

“Workflow for Optimizing Parameters at the Command Line” on page 3-87

“Configuring a Simulink Model for Optimizing Parameters” on page 3-88

“Creating or Extracting a Response Optimization Project” on page 3-88

“Specifying Design Requirements” on page 3-89

“Specifying Parameter Settings” on page 3-93

“Configuring Optimization and Simulation Settings” on page 3-94

“Running the Optimization” on page 3-95

Workflow for Optimizing Parameters at the Command Line

The workflow for optimizing model parameters at the command line includes the following tasks:

- 1 “Configuring a Simulink Model for Optimizing Parameters” on page 3-88
- 2 “Creating or Extracting a Response Optimization Project” on page 3-88
- 3 “Specifying Design Requirements” on page 3-89
- 4 “Specifying Parameter Settings” on page 3-93
- 5 “Configuring Optimization and Simulation Settings” on page 3-94
- 6 “Running the Optimization” on page 3-95

For a tutorial on how to optimize parameters at the command line, see “Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the Command Line”.

Tip To learn how to optimize parameters using the GUI, see “Optimizing Parameters Using the GUI” on page 3-3.

Configuring a Simulink Model for Optimizing Parameters

To optimize parameters of a Simulink model, first configure the model:

- 1 Open the Simulink model by typing the model name at the MATLAB prompt.
- 2 Open the Simulink Design Optimization library by typing `sdolib` at the MATLAB prompt.
- 3 Drag and drop the Signal Constraint block into the Simulink model window.

To learn more about the block, see [Signal Constraint block reference page](#).

- 4 Connect the Signal Constraint block to the signal to which you want to add specific design requirements.

Creating or Extracting a Response Optimization Project

Simulink Design Optimization provides the following commands for creating or extracting a response optimization project from a Simulink model that contains at least one Signal Constraint block:

Command	Use for
<code>newsro</code>	Create a new response optimization project with default settings
<code>getsro</code>	Extract a response optimization project

When you add a Signal Constraint block to the Simulink model, as described in “Configuring a Simulink Model for Optimizing Parameters” on page 3-88, the software creates a response optimization project for this model automatically. This response optimization contains default settings for design requirements, parameter values, optimization options, and simulation options.

Use the `newsro` command to extract this default response optimization project from the model.

Note When you use `newsro`, you also specify the model parameters for optimization. See the reference page for more information.

After extracting the default response optimization project, you can modify the requirements, as described in “Specifying Design Requirements” on page 3-89.

When a Simulink model already contains specific design requirements and settings, use the `getsro` command to extract the response optimization project from this model.

Tip You can also use `newsro` to create a new response optimization project with default settings for the model that already contains specific design requirements and settings.

The response optimization project, `proj`, returned by `newsro` and `getsro` has the following structure:

```
Name: 'sldo_model1'  
Parameters: [3x1 ResponseOptimizer.Parameter]  
OptimOptions: [1x1 sroengine.OptimOptions]  
Tests: [1x1 ResponseOptimizer.SimTest]  
Model: 'sldo_model1'
```

Response Optimization Project.

Specifying Design Requirements

After you create a response optimization project, as described in “Creating or Extracting a Response Optimization Project” on page 3-88, use the `findconstr` function to extract the constraints specified in the Signal Constraint block of the Simulink model.

The constraint object `constr` returned by `findconstr` contains data defining the design requirements. Design requirements include positions of the constraint bound segments and reference signals specified in the Signal Constraint block. The constraints define the region in which the response signal must lie.

The constraint object `constr` has the following structure:

```
ConstrEnable: 'on'  
isFeasible: 1  
CostEnable: 'off'  
Enable: 'on'  
Name: 'Signal Constraint'  
SignalSize: [1 1]  
LowerBoundX: [3x2 double]  
LowerBoundY: [3x2 double]  
LowerBoundWeight: [3x1 double]  
UpperBoundX: [2x2 double]  
UpperBoundY: [2x2 double]  
UpperBoundWeight: [2x1 double]  
ReferenceX: []  
ReferenceY: []  
ReferenceWeight: []
```

Signal Constraint.

To learn how to modify design requirements specified in the constraint object, see the following sections:

- “Specifying Signal Bounds” on page 3-90
- “Specifying a Reference Signal” on page 3-92

Specifying Signal Bounds

The `LowerBoundX`, `LowerBoundY`, `UpperBoundX`, and `UpperBoundY` properties of the constraint object represent numeric values of design requirements on a signal, specified in the Signal Constraint block of the model. These properties define the amplitude and time for the beginning and end points of each segment in the Signal Constraint block:

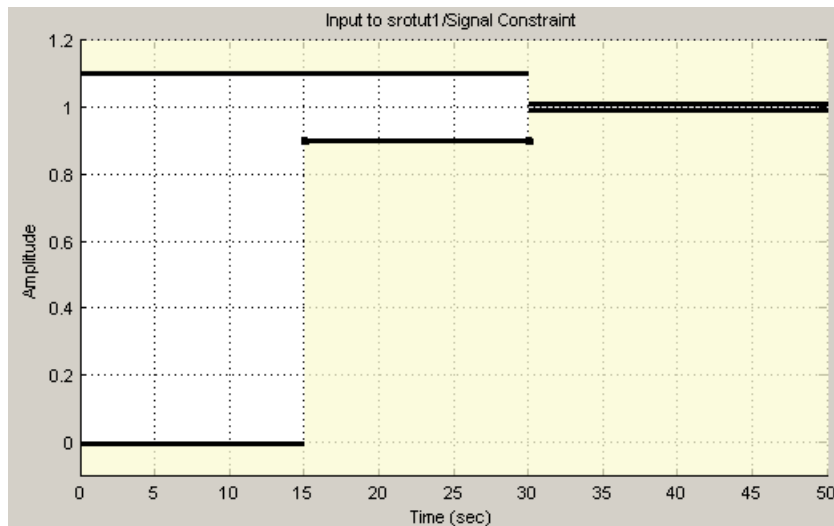
- `LowerBoundY` and `UpperBoundY` properties specify the start and end amplitude of the lower- and the upper-constraint bound segments.
- `LowerBoundX`, and `UpperBoundX` properties specify the time segments corresponding to the `LowerBoundY` and `UpperBoundY` amplitude properties, respectively.

Modify the properties `UpperBoundX`, `UpperBoundY`, `LowerBoundX`, and `LowerBoundY` to specify new signal bounds. For example:

```
constr.UpperBoundY=[1.1 1.1;1.01 1.01];  
constr.UpperBoundX=[0 30;30 50];  
constr.LowerBoundY=[0 0;0.9 0.9;0.99 0.99];  
constr.LowerBoundX=[0 15;15 30;30 50];
```

Note When you specify time values for the constraint bound segments, make sure that two consecutive time values do not overlap or have gaps between them.

If you add these constraints graphically in the Signal Constraint block, the constraints appear as shown in the following figure.



Note The Signal Constraint block in the Simulink model does not update to display the modified constraints. However, the software uses the updated constraint bounds specified in the constraint object when you optimize the parameters from the command line.

To view the updated value of the property, type `constr.PropertyName`. For example:

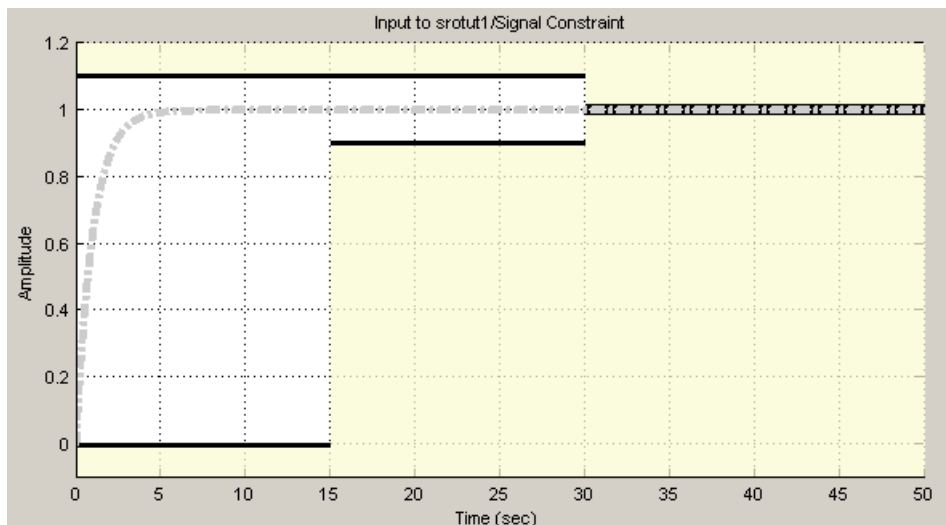
```
constr.LowerBoundY
```

Specifying a Reference Signal

`ReferenceX` and `ReferenceY` properties contain the time and amplitude vectors of the reference signal, respectively. Modify these properties to specify a new reference signal to track. For example:

```
constr.CostEnable='on';  
constr.ReferenceX=linspace(0,50,1000);  
constr.ReferenceY=1-exp(-linspace(0,50,1000));
```

If you include the reference signal graphically in the Signal Constraint block, the reference signal appears as shown in the following figure.



Note The Signal Constraint block in the Simulink model does not update to display the reference signal. However, the software uses the reference signal specified in the constraint object when you optimize the parameters from the command line.

To view the updated value of the property, type `constr.PropertyName`. For example:

```
constr.ReferenceX
```

Specifying Parameter Settings

After you create a response optimization project, as described in “Creating or Extracting a Response Optimization Project” on page 3-88, you can specify the parameters settings to use for optimization:

- 1 Use the `findpar` command to retrieve the parameter object `param` from the response optimization project `proj`.

The parameter object `param` has the following structure:

```
Name: 'Kd'
Value: 0
InitialGuess: 0
Minimum: -Inf
Maximum: Inf
TypicalValue: 0
ReferencedBy: {0x1 cell}
Description: ''
Tuned: 1
```

Tuned parameter.

The `Value` property specifies a value for the parameter in the Simulink model. Use the following properties to specify the parameter settings:

- `InitialGuess` — Initial value
- `Minimum` and `Maximum` — Parameter bounds
- `TypicalValue` — Scaling factor

To learn more about these properties, see the `findpar` reference page.

- 2 To specify parameter settings, edit the properties using dot notation.

For example:

```
param.Minimum=0;
```

Configuring Optimization and Simulation Settings

After you specify the design requirements and parameters for optimization, as described in “Specifying Design Requirements” on page 3-89 and “Specifying Parameter Settings” on page 3-93, you can optionally modify the optimization and simulation settings.

To modify the current optimization settings:

- 1 Extract the current optimization settings `opt_options` from the response optimization project using the `optimget` command.

The options object `opt_options` has the following structure:

```
Method: 'fmincon'  
Algorithm: 'active-set'  
Display: 'iter'  
GradientType: 'basic'  
MaximallyFeasible: 0  
MaxIter: 100  
TolCon: 1.0000e-003  
TolFun: 1.0000e-003  
TolX: 1.0000e-003  
Restarts: 0  
UseParallel: 'never'  
ParallelPathDependencies: {0x1 cell}  
SearchMethod: []
```

- 2 Modify the object properties using the `optimset` command.

For example:

```
optimset(proj, 'MaxIter', 150)
```

To learn more, see the corresponding reference pages.

Similarly, use the `simget` and `simset` commands to extract and modify the simulation settings, respectively. To learn more, see the corresponding reference pages.

Running the Optimization

Use the `optimize` command to optimize the response optimization project. After the optimization completes, you see the optimized parameter values displayed at the MATLAB prompt and also update in the response optimization project.

Tip To run another optimization using the updated parameter values as their initial values, use `initpar`. See the reference page for more information.

Optimization-Based Linear Control Design

- “Overview of Optimization-Based Compensator Design” on page 4-2
- “Supported Time- and Frequency-Domain Requirements” on page 4-4
- “Designing Optimization-Based Controllers for LTI Systems” on page 4-8
- “Designing Linear Controllers for Simulink Models” on page 4-30

Overview of Optimization-Based Compensator Design

When you have Control System Toolbox software installed, you can design and optimize control systems by tuning controller elements or parameters within a SISO Design Task in the Control and Estimation Tools Manager. You can tune elements or parameters such as poles, zeros, and gains within any controller in the system and optimize the open and closed loop responses.

Optimize the responses of systems in the SISO Design Task to meet both time- and frequency-domain performance requirements by graphically constraining signals:

- Add frequency-domain design requirements to plots such as root-locus, Nichols, and Bode in the SISO Design Task graphical tuning editor called SISO Design Tool.
- Add time-domain design requirements to plots such as step or impulse response (when displayed within the LTI Viewer as part of a SISO Design Task).

You can use optimization methods in a SISO Design Task in the Control and Estimation Tools Manager to tune both command-line LTI models as well as Simulink models:

- Create an LTI model using the Control System Toolbox command-line functions and use the `sisotool` function to create a SISO Design Task for the model.
- Use a Simulink Compensator Design task (from Simulink Control Design software) to automatically analyze the model and then create a SISO Design Task for a linearized version of the model. You can then use the optimization techniques in the SISO Design Task to tune the response of the linearized Simulink model.

Note When using response optimization within a SISO Design Task you cannot add uncertainty to system parameters.

When using a SISO Design Task, Simulink Design Optimization software automatically sets the model's simulation start and stop time and you cannot

directly change them. By default, the simulation starts at 0 and continues until the SISO Design Task determines that the dynamics of the model have settled out. In addition, when the design requirements extend beyond this point, the simulation continues to the extent of the design requirements. Although you cannot directly adjust the start or stop time of the simulation, you can adjust the design requirements to extend further in time and thus force the simulation to continue to a certain point.

Supported Time- and Frequency-Domain Requirements

In this section...

“Root Locus Diagrams” on page 4-4

“Open-Loop and Prefilter Bode Diagrams” on page 4-6

“Open-Loop Nichols Plots” on page 4-6

“Step/Impulse Response Plots” on page 4-7

This section lists the time- and frequency-domain requirements that you can specify for optimization-based control design in Simulink Design Optimization software:

Root Locus Diagrams

Settling Time

If you specify a settling time in the continuous-time root locus, a vertical line appears on the root locus plot at the pole locations associated with the value provided (using a first-order approximation). In the discrete-time case, the constraint is a curved line.

It is required that $\text{Re}\{pole\} < -4.6/T_{\text{settling}}$ for continuous systems and

$\log(\text{abs}(pole))/T_{\text{discrete}} < -4.6/T_{\text{settling}}$ for discrete systems. This is an approximation of the settling time based on second order dominant systems.

Percent Overshoot

Specifying percent overshoot in the continuous-time root locus causes two rays, starting at the root locus origin, to appear. These rays are the locus of poles associated with the percent value (using a second-order approximation). In the discrete-time case, the constraint appears as two curves originating at (1,0) and meeting on the real axis in the left-hand plane.

The percent overshoot $p.o$ constraint can be expressed in terms of the damping ratio, as in this equation:

$$p.o. = 100e^{-\pi\zeta/\sqrt{1-\zeta^2}}$$

where ζ is the damping ratio.

Damping Ratio

Specifying a damping ratio in the continuous-time root locus causes two rays, starting at the root locus origin, to appear. These rays are the locus of poles associated with the damping ratio. In the discrete-time case, the constraint appears as curved lines originating at (1,0) and meeting on the real axis in the left-hand plane.

The damping ratio defines a requirement on $-\text{Re}\{pole\}/\text{abs}\{pole\}$ for continuous systems and on

$$\begin{aligned} r &= \text{abs}(pSys) \\ t &= \text{angle}(pSys) \\ c &= -\log(r) / \sqrt{(\log(r))^2 + t^2} \end{aligned}$$

for discrete systems.

Natural Frequency

If you specify a natural frequency, a semicircle centered around the root locus origin appears. The radius equals the natural frequency.

The natural frequency defines a requirement on $\text{abs}\{pole\}$ for continuous systems and on

$$\begin{aligned} r &= \text{abs}(pSys) \\ t &= \text{angle}(pSys) \\ c &= \sqrt{(\log(r))^2 + t^2} / T_{s_{model}} \end{aligned}$$

for discrete systems.

Region Constraint

Specifies an exclusion region in the complex plane, causing a line to appear between the two specified points with a shaded region below the line. The poles must not lie in the shaded region.

Open-Loop and Prefilter Bode Diagrams

Gain and Phase Margins

Specify a minimum phase and or a minimum gain margin.

Upper Gain Limit

You can specify an upper gain limit, which appears as a straight line on the Bode magnitude curve. You must select frequency limits, the upper gain limit in decibels, and the slope in dB/decade.

Lower Gain Limit

Specify the lower gain limit in the same fashion as the upper gain limit.

Open-Loop Nichols Plots

Phase Margin

Specify a minimum phase amount.

While displayed graphically at only one location around a multiple of -180 degrees, this requirement applies to phase margin regardless of actual phase (i.e., it is interpreted for all multiples of -180).

Gain Margin

Specify a minimum gain margin.

While displayed graphically at only one location around a multiple of -180 degrees, this requirement applies to gain margin regardless of actual phase (i.e., it is interpreted for all multiples of -180).

Closed-Loop Peak Gain

Specify a peak closed-loop gain at a given location. The specified value can be positive or negative in dB. The constraint follows the curves of the Nichols plot grid, so it is recommended that you have the grid on when using this feature.

While displayed graphically at only one location around a multiple of -180 degrees, this requirement applies to gain margin regardless of actual phase (i.e., it is interpreted for all multiples of -180).

Gain-Phase Requirement

Specifies an exclusion region for the response on the Nichols plot. The response must not pass through the shaded region.

This only applies to the region (phase and gain) drawn.

Step/Impulse Response Plots

Upper Time Response Bound

You can specify an upper time response bound.

Lower Time Response Bound

You can specify a lower time response bound.

Designing Optimization-Based Controllers for LTI Systems

In this section...

“How to Design Optimization-Based Controllers for LTI Systems” on page 4-8

“Example — Frequency-Domain Optimization for LTI System” on page 4-9

How to Design Optimization-Based Controllers for LTI Systems

To design optimization-based linear controller for an LTI model:

- 1** Create and import a linear model into a SISO Design Task. You can create an LTI model at the MATLAB command line, as described in “Creating an LTI Plant Model” on page 4-10.
- 2** Create a SISO Design Task with design and analysis plots, as described in “Creating Design and Analysis Plots” on page 4-11.

To learn more about SISO Design Tool, see “Using the SISO Design Task in the Controls & Estimation Tools Manager” in the Control System Toolbox documentation.

- 3** Under **Automated Tuning** select **Optimization based tuning** as the **Design Method** and then click the **Optimize Compensators** button to create a **Response Optimization** task within the Control and Estimation Tools Manager. See “Creating a Response Optimization Task” on page 4-14 for more information.
- 4** Within the **Response Optimization** node, select the **Compensators** pane to select and configure the compensator elements you want to tune during the response optimization. See “Selecting Tunable Compensator Elements” on page 4-16 for more information.
- 5** Under **Design requirements** in the **Response Optimization** node, select the design requirements you want the system to satisfy. See “Adding Design Requirements” on page 4-17 for more information.

- 6 Click the **Start Optimization** button within the **Response Optimization** node. The optimization progress results appear under **Optimization**. The **Compensators** pane contains the new, optimized compensator element values. See “Optimizing the System’s Response” on page 4-25 for more information.

Example — Frequency-Domain Optimization for LTI System

- “Introduction” on page 4-9
- “Design Requirements” on page 4-10
- “Creating an LTI Plant Model” on page 4-10
- “Creating Design and Analysis Plots” on page 4-11
- “Creating a Response Optimization Task” on page 4-14
- “Selecting Tunable Compensator Elements” on page 4-16
- “Adding Design Requirements” on page 4-17
- “Optimizing the System’s Response” on page 4-25
- “Creating and Displaying the Closed-Loop System” on page 4-28

Introduction

When you have Control System Toolbox software, you can place Simulink Design Optimization design requirements or constraints on plots in the SISO Design Tool graphical tuning editor and analysis plots that are part of a SISO Design Task. This allows you to include design requirements for response optimization in the frequency-domain in addition to the time-domain. This section guides you through an example using frequency-domain design requirements to optimize the response of a system in the SISO Design Task.

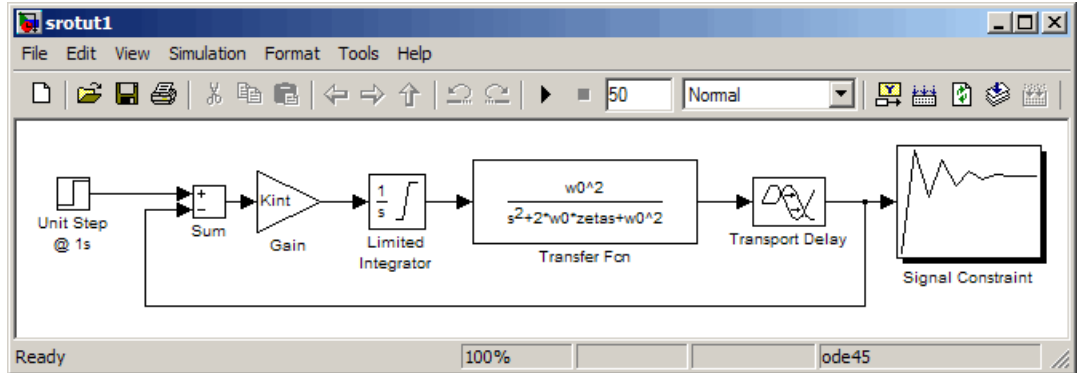
You can specify frequency-domain design requirements to optimize response signals for any model that you can design within a SISO Design Task:

- Command-line LTI models created with the Control System Toolbox commands

- Simulink models that have been linearized using Simulink Control Design software

Design Requirements

In this example, you use a linearized version of the following Simulink model.



You use optimization methods to design a compensator so that the closed loop system meets the following design specifications when you excite the system with a unit step input:

- A maximum 30-second settling time
- A maximum 10% overshoot
- A maximum 10-second rise time
- A limit of ± 0.7 on the actuator signal

Creating an LTI Plant Model

In the srotut1 model, the plant model is composed of a gain, a limited integrator, a transfer function, and a transport delay.

You want to design the compensator for the open loop transfer function of the linearized srotut1 model. The linearized srotut1 plant model is composed of the gain, an unlimited integrator, the transfer function, and a Padé approximation to the transport delay.

To create an open loop transfer function based on the linearized `srotut1` model, enter the following commands:

```
w0      = 1;
zeta    = 1;
Kint    = 0.5;
Tdelay  = 1;
[delayNum,delayDen] = pade(Tdelay,1);
integrator = tf(Kint,[1 0]);
transfer_fcn = tf(w0^2,[1 2*w0*zeta w0^2]);
delay_block = tf(delayNum,delayDen);
open_loopTF = integrator*transfer_fcn*delay_block;
```

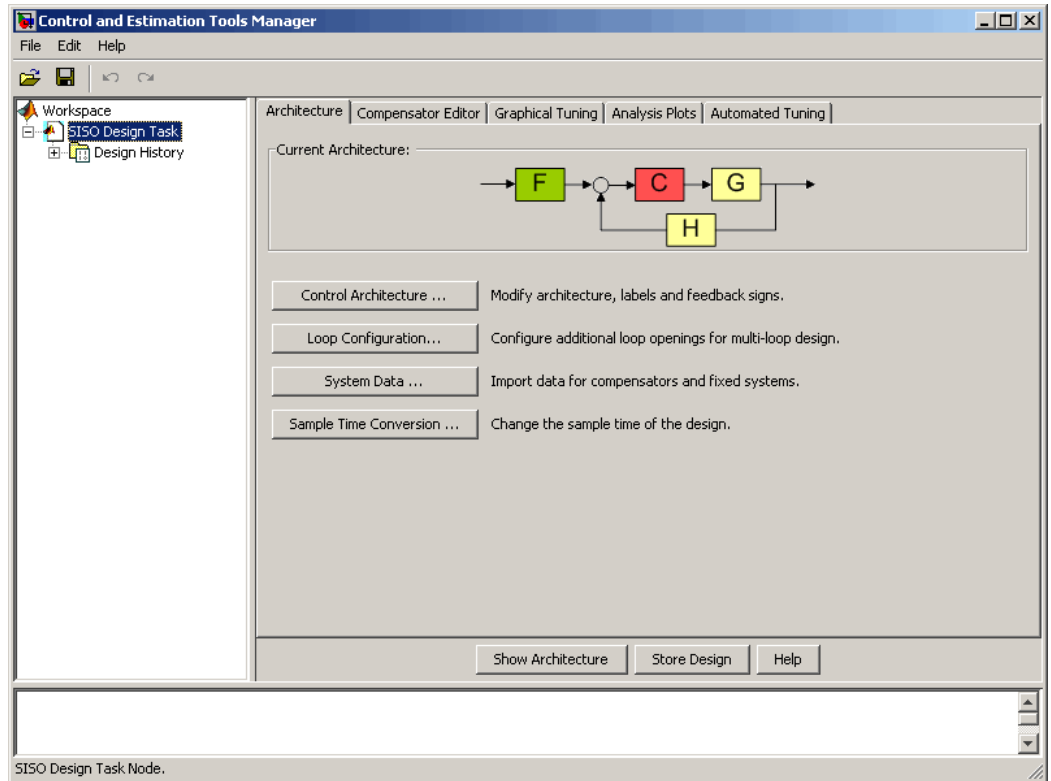
Note It is also possible to directly linearize the Simulink model `srotut1` using Simulink Control Design software.

Creating Design and Analysis Plots

This example uses a root locus diagram to design the response of the open loop transfer function, `open_loopTF`. To create a SISO Design Task, containing a root-locus plot for the open loop transfer function, use the following command:

```
sisotool('rlocus',open_loopTF)
```

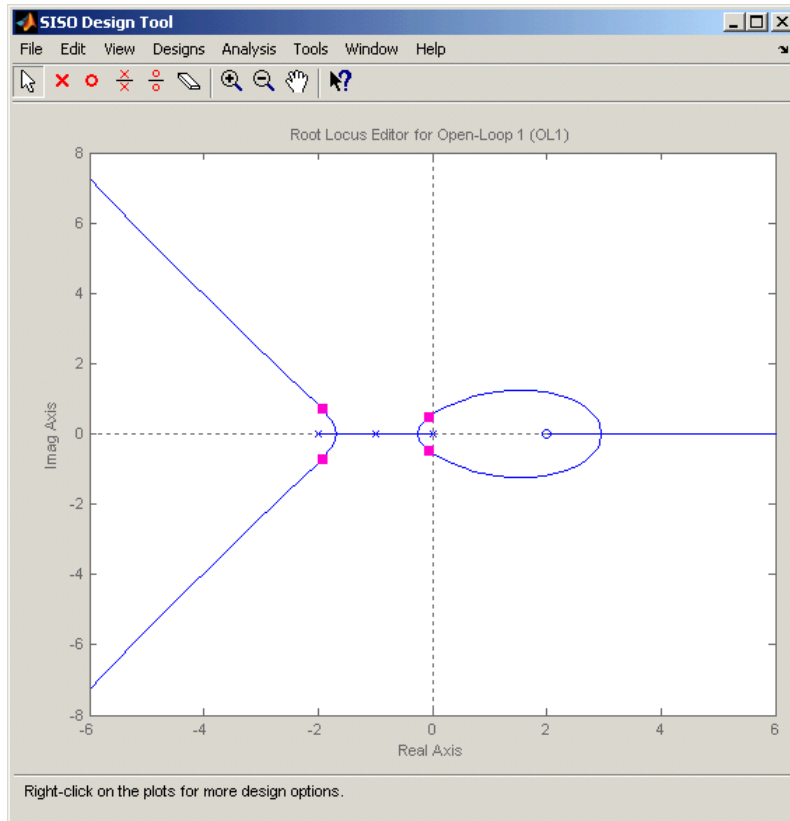
A **SISO Design Task** is created within the Control and Estimation Tools Manager, as shown in the following figure.



The Control and Estimation Tools Manager is a graphical environment for managing and performing tasks such as designing SISO systems. The SISO Design Task node contains five panels that perform actions related to designing SISO control systems. For more information, see “Using the SISO Design Task in the Controls & Estimation Tools Manager” in “Control System Toolbox” documentation.

The **Architecture** pane, within the **SISO Design Task** node, lets you choose the architecture for the control system you are designing. This example uses the default architecture. In this system, the plant model, G , is the open loop transfer function `open_loopTF`, the prefilter, F , and the sensor, H , are set to 1, and the compensator, C , is the compensator that will be designed using response optimization methods.

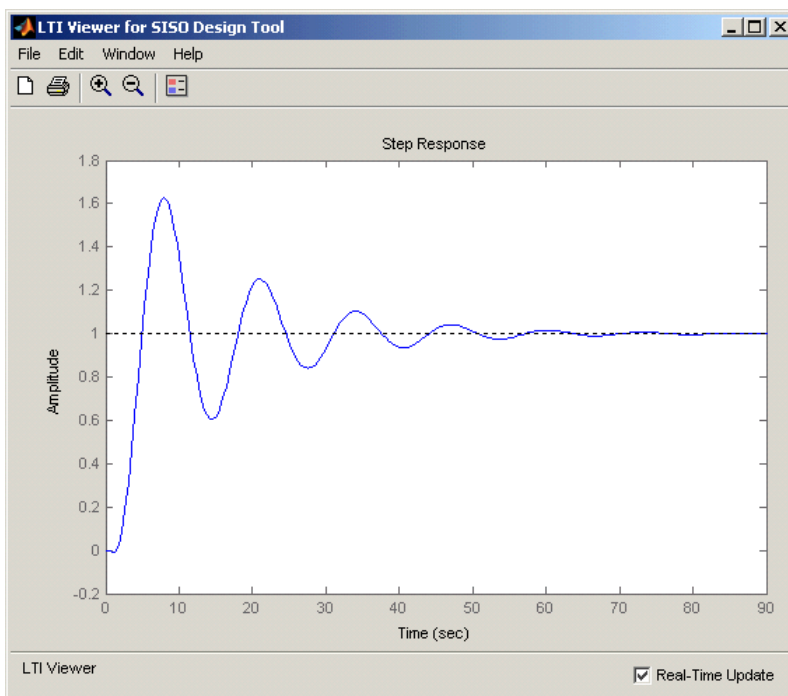
The SISO Design Task also contains a root locus diagram in the SISO Design Tool graphical tuning editor.



In addition to the root-locus diagram, it is helpful to visualize the response of the system with a step response plot. To add a step response:

- 1** Select the **Analysis Plots** pane with the **SISO Design Task** node of the Control and Estimation Tool Manager.
- 2** Select Step for the **Plot Type** of **Plot 1**.
- 3** Under **Contents of Plots**, select the check box in column **1** for the response **Closed Loop r to y**.

A step response plot appears in an LTI Viewer. The plot shows the response of the closed loop system from r (input to the prefilter, F) to y (output of the plant model, G):



Creating a Response Optimization Task

There are several possible methods for designing a SISO system; this example uses an automated approach involving response optimization methods. After creating the design and analysis plots as discussed in “Creating Design and Analysis Plots” on page 4-11, you are ready to start a response optimization task to design the compensator.

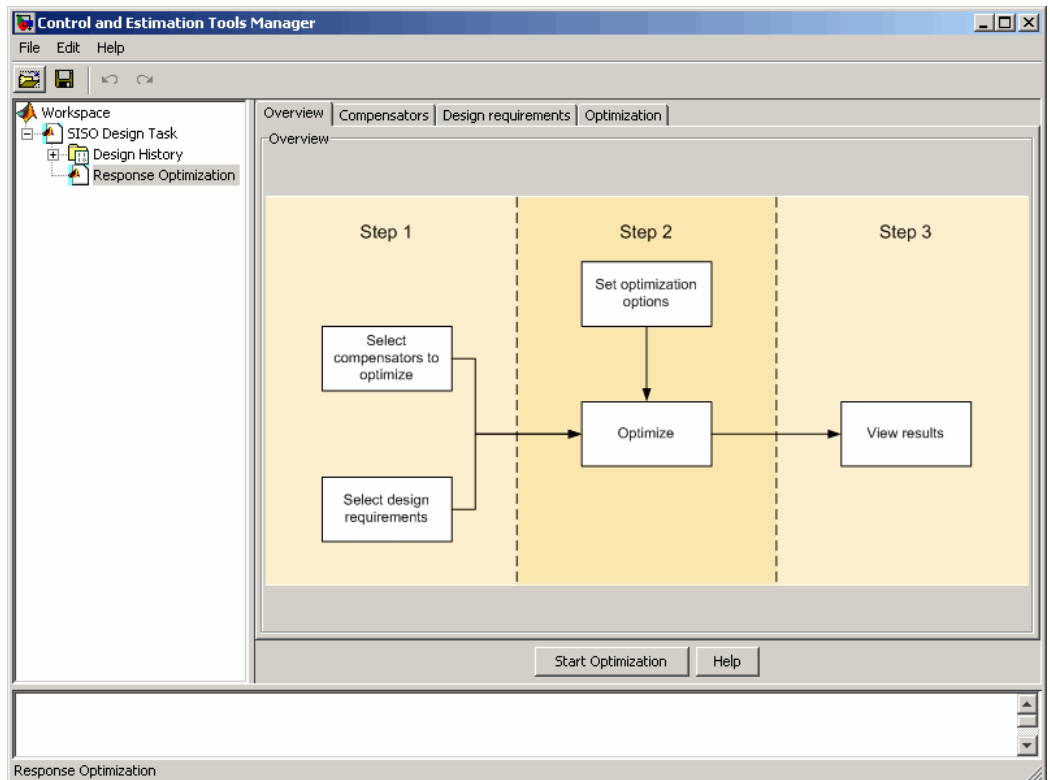
To create a response optimization task:

- 1 Select the **Automated Tuning** pane within the **SISO Design Task** node in the Control and Estimation Tools Manager.

2 In the **Automated Tuning** pane, select Optimization based tuning as the **Design Method**.

3 Click the **Optimize Compensators** button to create the **Response Optimization** node under the **SISO Design Task** node in the tree browser in the left pane of the Control and Estimation Tools Manager.

The **Response Optimization** node contains four panes as shown in the next figure.



With the exception of the first pane, each corresponds to a step in the response optimization process:

- **Overview:** A schematic diagram of the response optimization process.

- **Compensators:** Select and configure the compensator elements that you want to tune. See “Selecting Tunable Compensator Elements” on page 4-16.
- **Design requirements:** Select the design requirements that you want the system to meet after tuning the compensator elements. See “Adding Design Requirements” on page 4-17.
- **Optimization:** Configure optimization options and view the progress of the response optimization. See “Optimizing the System’s Response” on page 4-25.

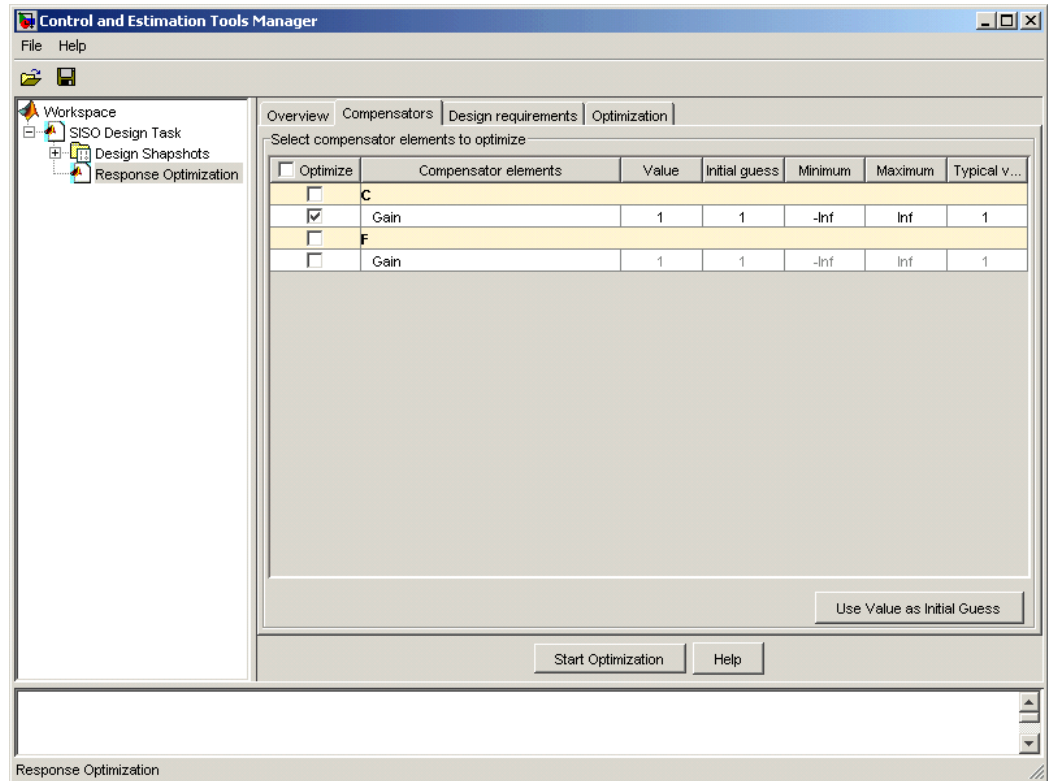
Note When optimizing responses in a SISO Design Task, you cannot add uncertainty to parameters or compensator elements.

Selecting Tunable Compensator Elements

You can tune elements or parameters within compensators in your system so that the response of the system meets the design requirements you specify. To specify the compensator elements to tune:

- 1** Select the **Compensators** pane within the **Response Optimization** node.
- 2** Within the **Compensators** pane, select the check boxes in the **Optimize** column that correspond to the compensator elements you want to tune.

In this example, to tune the **Gain** in the compensator **C**, select the check box next to this element, as shown in the following figure.



Note Compensator elements or parameters cannot have uncertainty when used with frequency-domain based response optimization.

Adding Design Requirements

You can use both frequency-domain and time-domain design requirements to tune parameters in a control system. The **Design requirements** pane within the **Response Optimization** node of the Control and Estimation Tools Manager provides an interface to create new design requirements and select those you want to use for a response optimization.

This example uses the design specifications described in “Design Requirements” on page 4-10. The following sections each create a new design requirement to meet these specifications:

- “Settling Time Design Requirement” on page 4-18
- “Overshoot Design Requirement” on page 4-19
- “Rise Time Design Requirement” on page 4-21
- “Actuator Limit Design Requirement” on page 4-22

After you add the design requirements, you can select a subset of requirements for controller design, as described in “Selecting the Design Requirements to Use During Response Optimization” on page 4-25.

Settling Time Design Requirement. The first design specification for this example is to have a settling time of 30 seconds or less. This specification can be represented on a root-locus diagram as a constraint on the real parts of the poles of the open loop system.

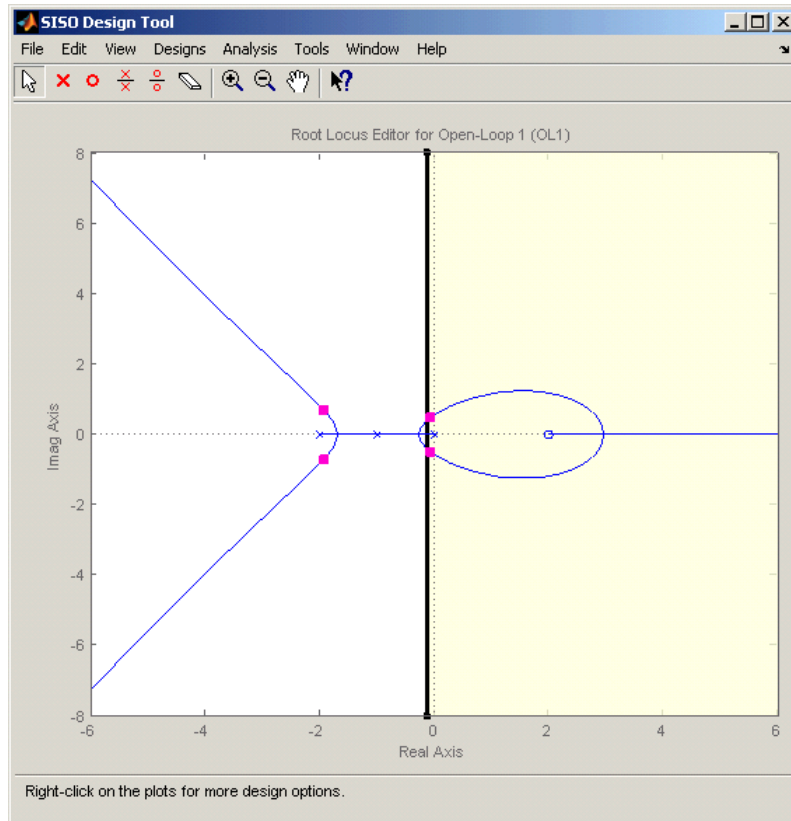
To add this design requirement:

- 1** Select the **Design requirements** pane within the **Response Optimization** node of the Control and Estimation Tools Manager.
- 2** Click the **Add new design requirement** button. This opens the New Design Requirement dialog box.

Within this dialog box you can specify new design requirements and add them to a new or existing design or analysis plot.

- 3** Add a design requirement to the existing root-locus diagram:
 - a** Select Pole/zero settling time from the **Design requirement type** menu.
 - b** Select Open-Loop L from the **Requirement for response** menu.
 - c** Enter 30 seconds for the **Settling time**.
 - d** Click **OK**.

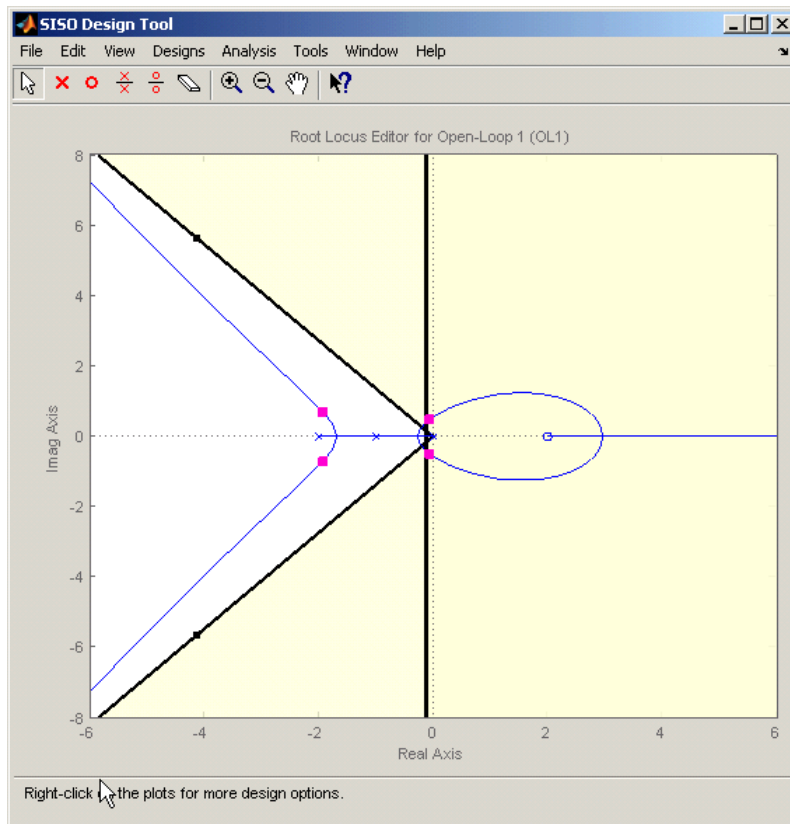
A vertical line should appear on the root-locus diagram, as shown in the following figure.



Overshoot Design Requirement. The second design specification for this example is to have a percentage overshoot of 10% or less. This specification is related to the damping ratio on a root-locus diagram. In addition to adding a design requirement with the **Add new design requirement** button, you can also right-click directly on the design or analysis plots to add the requirement, as shown next.

To add this design requirement:

- 1 Right-click anywhere within the white space of the root-locus diagram in the SISO Design Tool window. Select **Design Requirements** > **New** to open the New Design Requirement dialog box.
- 2 Select **Percent overshoot** as the **Design requirement type** and enter 10 as the **Percent overshoot**.
- 3 Click **OK** to add the design requirement to the root-locus diagram. The design requirement appears as two lines radiating at an angle from the origin, as shown in the following figure.

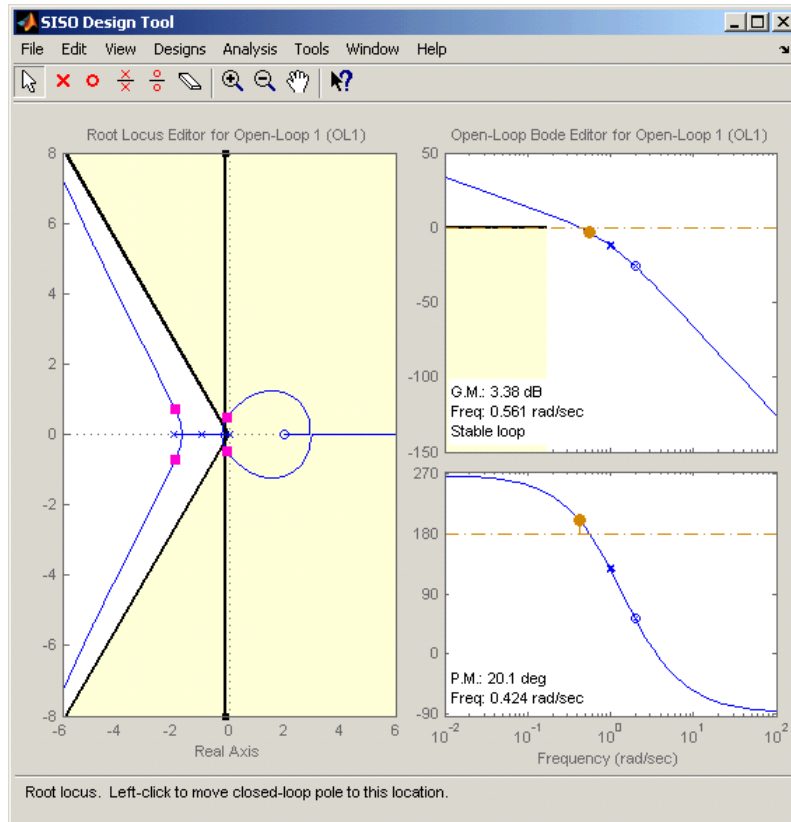


Rise Time Design Requirement. The third design specification for this example is to have a rise time of 10 seconds or less. This specification is related to a lower limit on a Bode Magnitude diagram.

To add this design requirement:

- 1** Select the **Graphical Tuning** pane in the **SISO Design Task** node of the Control and Estimation Tools Manager.
- 2** For Plot 2, set **Plot Type** to Open-Loop Bode.
- 3** Right-click anywhere within the white space of the open-loop bode diagram in the SISO Design Tool window. Select **Design Requirements > New** to open the New Design Requirement dialog box.
- 4** Create a design requirement to represent the rise time and add it to the new Bode plot:
 - a** Select **Lower gain limit** from the **Design requirement type** menu.
 - b** Enter $1e-2$ to 0.17 for the **Frequency** range.
 - c** Enter 0 to 0 for the **Magnitude** range.
 - d** Click **OK**.

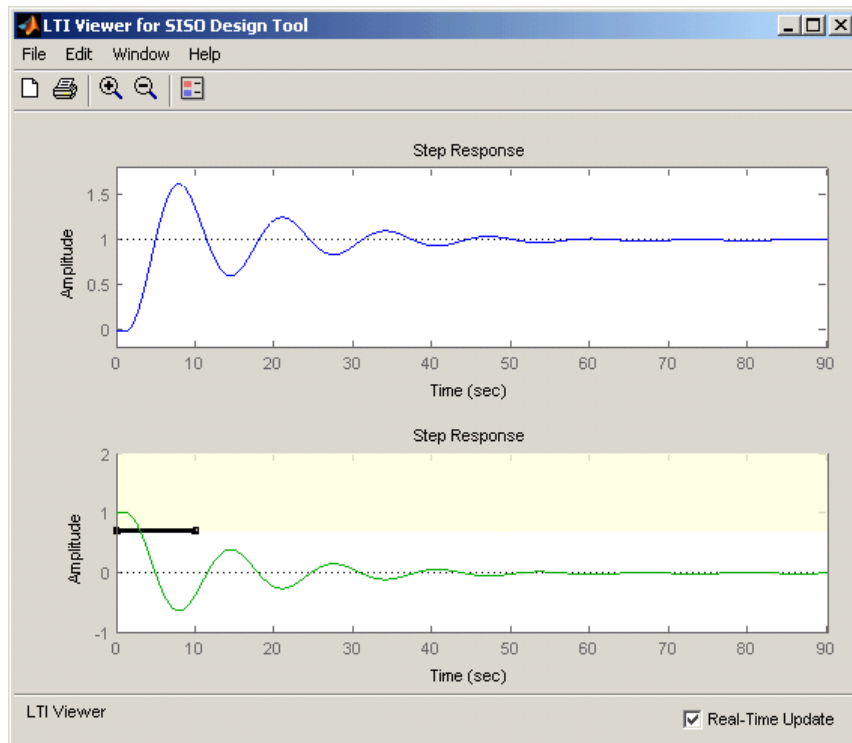
A Bode diagram appears within the SISO Design Tool window. The magnitude plot of the Bode diagram includes a horizontal line representing the design requirement, as shown in the following figure.



Actuator Limit Design Requirement. The fourth design specification for this example is to limit the actuator signal to within ± 0.7 . To add this design requirement:

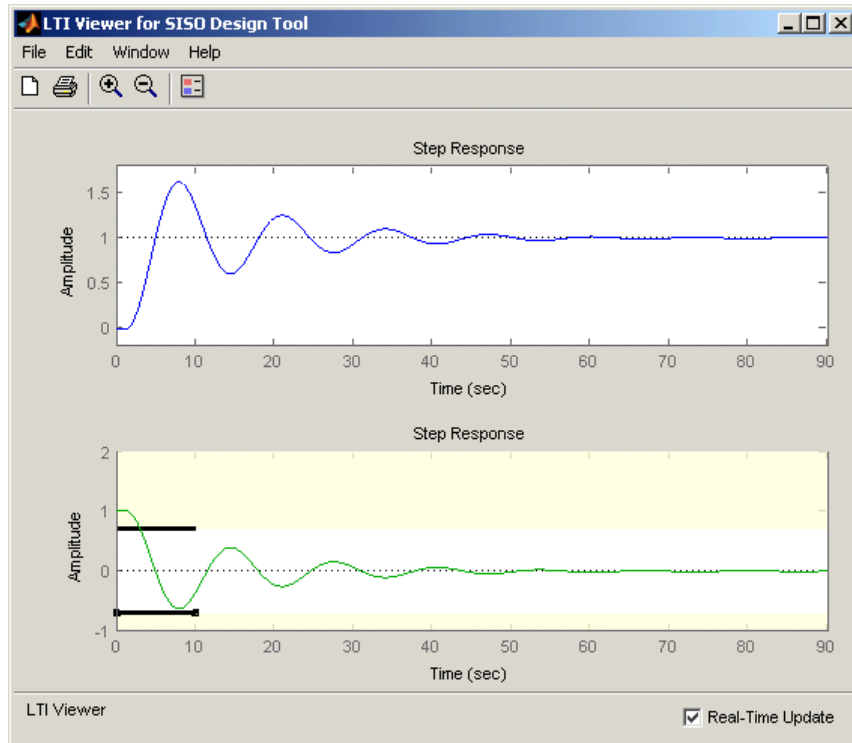
- 1 Select the **Design requirements** pane in the **Response Optimization** node of the Control and Estimation Tools Manager.
- 2 Click the **Add new design requirement** button to open the New Design Requirement dialog box.
- 3 Create a time-domain design requirement to represent the upper limit on the actuator signal, and add it to a new step response plot in the LTI Viewer:

- a Select Step response upper amplitude limit from the **Design requirement type** menu.
- b Select Closed Loop r to u from the **Requirement for response** menu.
- c Enter 0 to 10 for the **Time** range.
- d Enter 0.7 to 0.7 for the **Amplitude** range.
- e Click **OK**. A second step response plot for the closed loop response from r to u appears in the LTI Viewer. The plot contains a horizontal line representing the upper limit on the actuator signal.
- f To extend this limit for all times (to $t=\infty$), right click on the black edge of the design requirement, somewhere toward the right edge, and select **Extend to inf**. The diagram should now appear as shown next.



To add the corresponding design requirement for the lower limit on the actuator signal:

- 1** Select the **Design requirements** pane in the **Response Optimization** node of the Control and Estimation Tools Manager.
- 2** Click the **Add new design requirement** button to open the New Design Requirement dialog box.
- 3** Create a time-domain design requirement to represent the lower limit on the actuator signal, and add it to the step response plot in the LTI Viewer:
 - a** Select **Step response lower amplitude limit** from the **Design requirement type** menu.
 - b** Select **Closed Loop r to u** from the **Requirement for response** menu.
 - c** Enter 0 to 10 for the **Time** range.
 - d** Enter -0.7 to -0.7 for the **Amplitude** range.
 - e** Click **OK**. The step response plot now contains a second horizontal line representing the lower limit on the actuator signal.
 - f** To extend this limit for all times (to $t=\infty$), right-click in the yellow shaded area and select **Extend to inf**. The diagram should now appear as shown in the following figure.



Selecting the Design Requirements to Use During Response

Optimization. The design requirements give constraints on the dynamics of the system and the values of response signals. The table in the **Design requirements** tab lists all design requirements in the design and analysis plots. Select the check boxes next to the design requirements you want to use in the response optimization. This example uses all the current design requirements.

Optimizing the System's Response

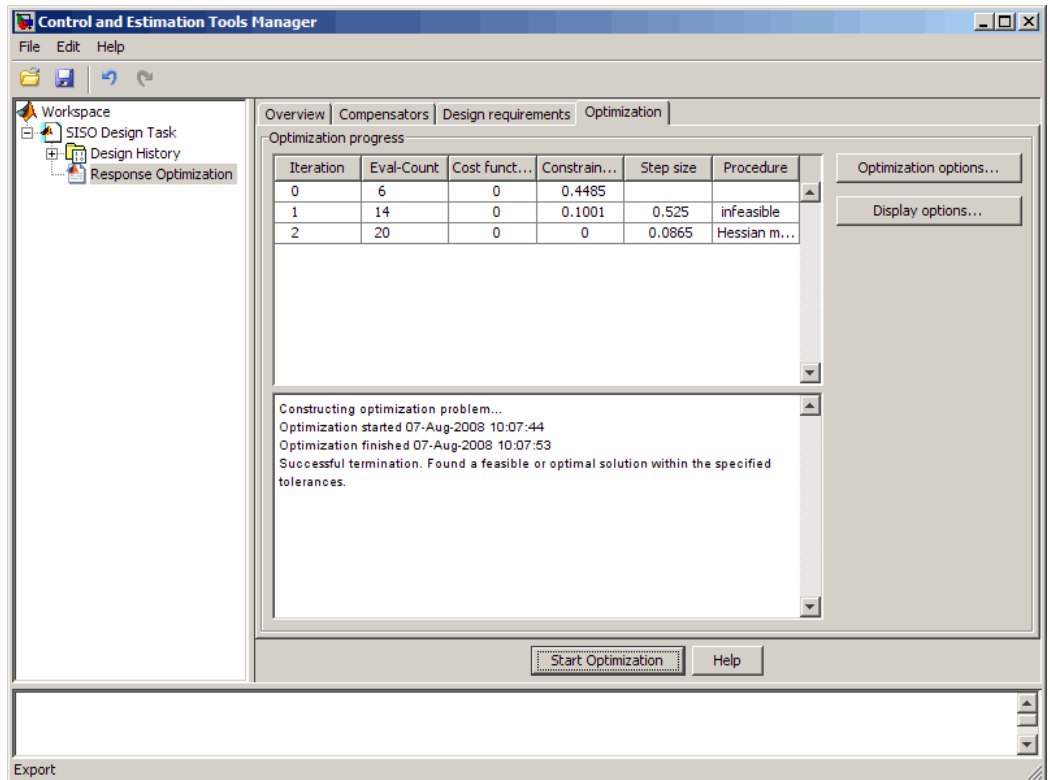
After selecting the compensator elements to tune and adding design requirements for the response signals to satisfy, you are ready to begin the response optimization.

The **Optimization** pane within the **Response Optimization** node of the Control and Estimation Tools Manager displays the progress of the response

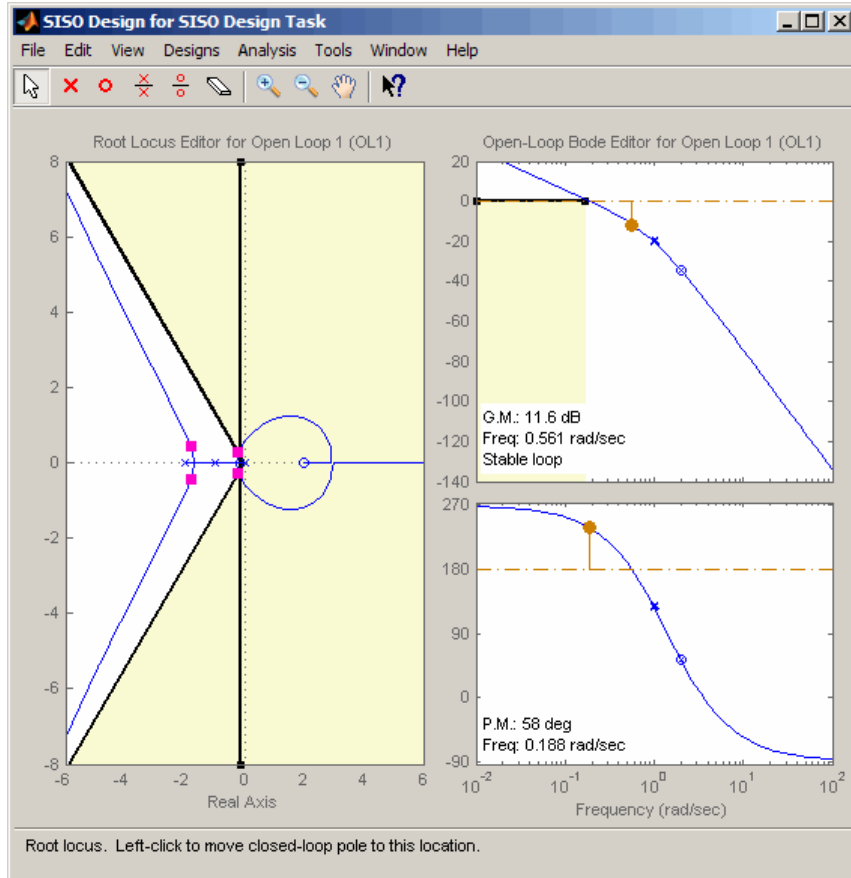
optimization. The pane also contains options to configure the types of progress information displayed during the optimization and options to configure the optimization methods and algorithms.

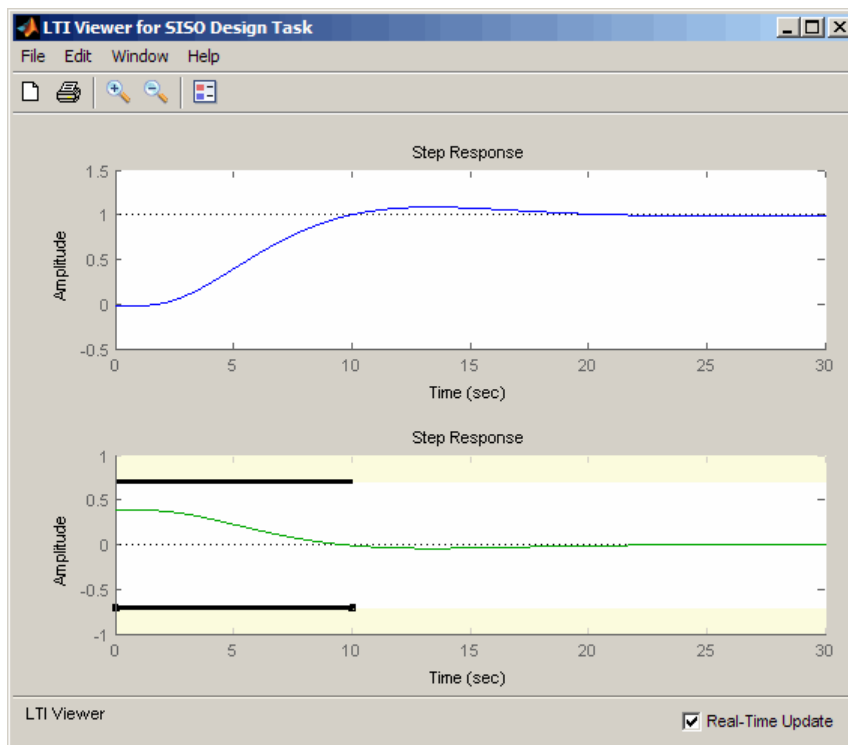
To optimize the response of the system in this example, click the **Start Optimization** button.

The **Optimization** pane displays the progress of the optimization, iteration by iteration, as shown next. Termination messages from the optimization method and suggestions for improving convergence also appear here.



The optimized signals in the design and analysis plots appear as follows:





Creating and Displaying the Closed-Loop System

After designing a compensator by optimizing the response of the system, you can export the compensator to the MATLAB workspace, and create a model of the full closed-loop system.

- 1 Within the SISO Design Tool window, select **File > Export** to open the SISO Tool Export dialog box.
- 2 Select the compensator you designed, **Compensator C**, and then click the **Export to Workspace** button.

At the command line, enter the following command to create the closed-loop system, `CL`, from the open-loop transfer function, `open_loopTF`, and the compensator, `C`:

```
CL=feedback(C*open_loopTF,1)
```

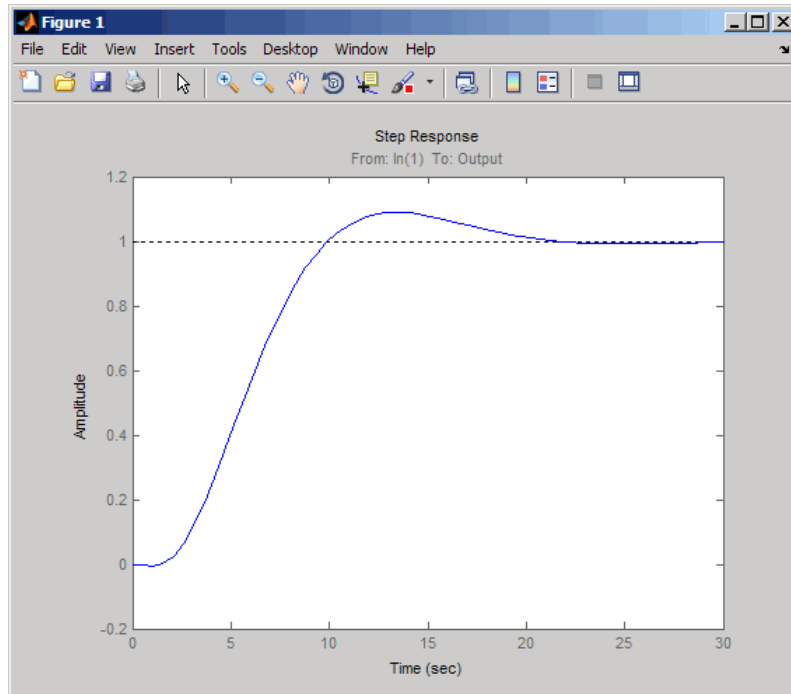
This returns the following model:

```
Zero/pole/gain from input to output "Output":  
-0.19414 (s-2)  
-----  
(s^2 + 0.409s + 0.1136) (s^2 + 3.591s + 3.418)
```

To create a step response plot of the closed loop system, enter the following command:

```
step(CL);
```

This produces the following figure:



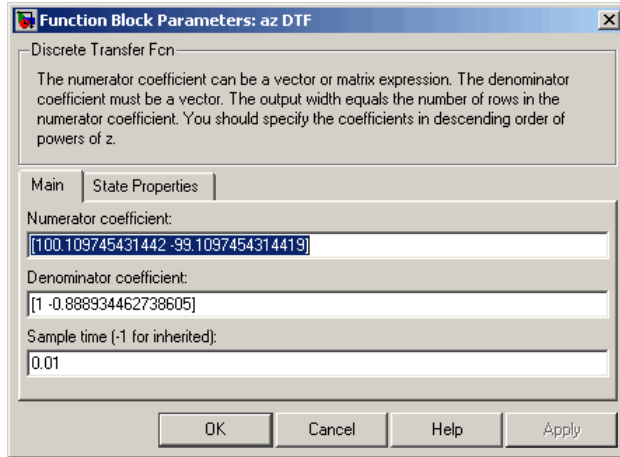
Designing Linear Controllers for Simulink Models

When you have Control System Toolbox and Simulink Control Design software, you can perform frequency-domain optimization of Simulink models.

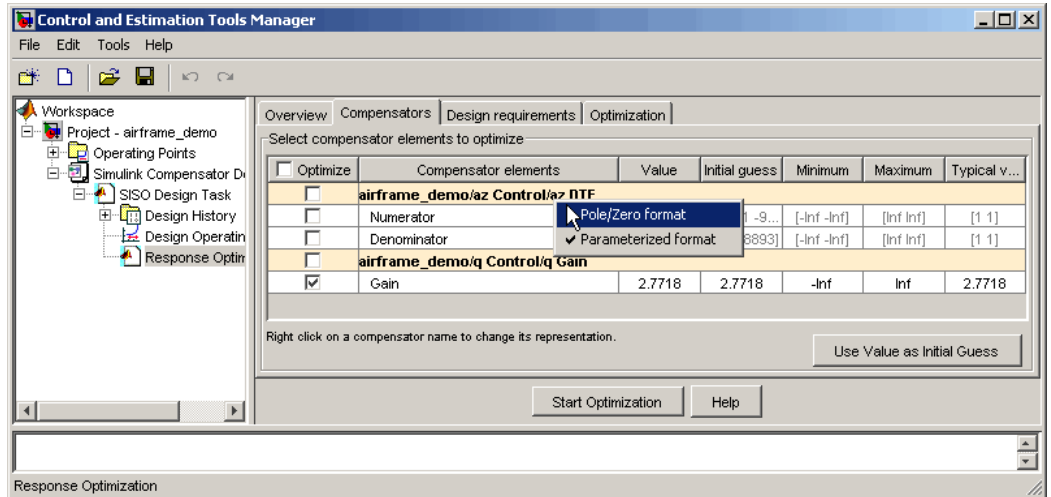
You can use Simulink Control Design software to configure SISO Design Tool with compensators, inputs, outputs, and loops computed from a Simulink model. For more information, see “Creating a SISO Design Task” in Simulink Control Design documentation.

After you configure the SISO Design Tool, use Simulink Design Optimization software to optimize the controller parameters of the linearized Simulink model. For an example of optimization-based control design for a model linearized using Simulink Control Design software, see “Tutorial — Designing a PID Controller Using Optimization-Based Tuning” in the *Simulink Design Optimization getting Started Guide*.

There is only one difference when tuning compensators derived from Simulink Control Design software: The tuning of compensators from a Simulink model is done through the masks of the Simulink blocks representing each compensator. When selecting parameters to optimize, users can tune the compensator in the pole, zero, or gain format, or in a format consistent with the Simulink block mask as shown in the following figure. Changing the compensator format is not possible when optimizing pure SISO Tool models (those not derived using Simulink Control Design software).



Mask of a Simulink® compensator block



Response optimization compensators pane

Lookup Tables

- “What Are Lookup Tables?” on page 5-2
- “Estimating Values of Lookup Tables” on page 5-5
- “Capturing Time-Varying System Behavior Using Adaptive Lookup Tables” on page 5-37

What Are Lookup Tables?

In this section...
“Static Lookup Tables” on page 5-2
“Adaptive Lookup Tables” on page 5-3

Static Lookup Tables

Lookup tables are tables that store numeric data in a multidimensional array format. In the simpler two-dimensional case, lookup tables can be represented by matrices. Each element of a matrix is a numerical quantity, which can be precisely located in terms of two indexing variables. At higher dimensions, lookup tables can be represented by multidimensional matrices, whose elements are described in terms of a corresponding number of *indexing variables*.

Lookup tables provide a means to capture the dynamic behavior of a physical (mechanical, electronic, software) system. The behavior of a system with M inputs and N outputs can be approximately described by using N lookup tables, each consisting of an array with M dimensions.

You usually generate lookup tables by experimentally collecting or artificially creating the input and output data of a system. In general, you need as many indexing parameters as the number of input variables. Each indexing parameter may take a value within a predetermined set of data points, which are called the *breakpoints*. The set of all breakpoints corresponding to an indexing variable is called a *grid*. Thus, a system with M inputs is gridded by M sets of breakpoints. The software uses the breakpoints to locate the array elements, where the output data of the system are stored. For a system with N outputs, the software locates the N array elements and then stores the corresponding data at these locations.

After you create a lookup table using the input and output measurements as described previously, you can use the corresponding multidimensional array of values in applications without having to remeasure the system outputs. In fact, you need only the input data to locate the appropriate array elements in the lookup table because the software reads the approximate system output from the data stored at these locations. Therefore, a lookup table provides a

suitable means of capturing the input-output mapping of a *static* system in the form of numeric data stored at predetermined array locations. For more information, see “About Lookup Table Blocks” in the Simulink documentation.

You can use Simulink Design Optimization software to estimate lookup table values, as described in “Estimating Values of Lookup Tables” on page 5-5.

Adaptive Lookup Tables

Statically defined lookup tables, as described in “Static Lookup Tables” on page 5-2, cannot accommodate the *time-varying* behavior (characteristics) of a physical plant. Static lookup tables establish a permanent and static mapping of input-output behavior of a physical system. Conversely, the behavior of actual physical systems often varies with time due to wear, environmental conditions, and manufacturing tolerances. With such variations, the static mapping of input-output behavior of a plant described by the lookup table may no longer provide a valid representation of the plant characteristics.

Adaptive lookup tables incorporate the time-varying behavior of physical plants into the lookup table generation and maintenance process while providing all of the functionality of a regular lookup table.

The adaptive lookup table receives the input and output measurements of a plant’s behavior, which are then used to dynamically create and update the content of the underlying lookup table. In addition to requiring the input data to create the lookup table, the adaptive lookup table also uses the output data of the plant to recalculate the table values. For example, you can collect the output data of the plant by placing sensors at appropriate locations in a physical system.

The software uses the input measurements to locate the array elements by comparing these input values with the breakpoints defined for each indexing variable. Next, it uses the output measurements to recalculate the numeric value stored at these array locations. However, unlike a regular table, which only stores the array data before the actual use of the lookup table, the adaptive table continuously improves the content of the lookup table. This continuous improvement of the table data is referred to as the *adaptation process* or *learning process*.

The adaptation process involves statistical and signal processing algorithms to recapture the input-output behavior of the plant. The adaptive lookup table always tries to provide a valid representation of the plant dynamics even though the plant behavior may be time varying. The underlying signal processing algorithms are also robust against reasonable measurement noise and they provide appropriate filtering of noisy output measurements. To learn more about how to model systems using adaptive lookup tables, see “Capturing Time-Varying System Behavior Using Adaptive Lookup Tables” on page 5-37.

Estimating Values of Lookup Tables

In this section...

“How to Estimate Values of a Lookup Table” on page 5-5

“Example — Estimating Lookup Table Values from Data” on page 5-6

“Example — Estimating Constrained Values of a Lookup Table” on page 5-20

How to Estimate Values of a Lookup Table

You can use lookup table Simulink blocks to approximate a system’s behavior, as described in “Working with Lookup Tables” in the Simulink documentation. After you build your system using lookup tables, you can use Simulink Design Optimization software to estimate the table values from measured I/O data.

Estimating lookup table values is an example of estimating parameters which are matrices or multi-dimensional arrays. The workflow for estimating parameters of a lookup table consist of the following tasks:

- 1** Creating a Simulink model using lookup table blocks.
- 2** Importing the measured input and output (I/O) data from which you want to estimate the table values.
- 3** Analyzing and preparing the I/O data for estimation.
- 4** Estimating the lookup table values.
- 5** Validating the estimated table values using a validation data set.

The following examples illustrate how to estimate the lookup table values:

- “Example — Estimating Lookup Table Values from Data” on page 5-6
- “Example — Estimating Constrained Values of a Lookup Table” on page 5-20

Example — Estimating Lookup Table Values from Data

- “Objectives” on page 5-6
- “About the Data” on page 5-6
- “Configuring a Project for Parameter Estimation” on page 5-6
- “Estimating the Table Values Using Default Settings” on page 5-8
- “Validating the Estimation Results” on page 5-15

Objectives

This example shows how to estimate lookup table values from time-domain input-output (I/O) data.

About the Data

In this example, you use the I/O data in `lookup_regular.mat` to estimate the values of a lookup table. The MAT-file includes the following variables:

- `xdata1` — Consists of 63 uniformly-sampled input data points in the range $[0,6.5]$.
- `ydata1` — Consists of output data corresponding to the input data samples.
- `time1` — Time vector.

You use the I/O data to estimate the lookup table values in the `lookup_regular` Simulink model. The lookup table in the model contains ten values, which are stored in the MATLAB variable `table`. The initial table values comprise a vector of 0s. To learn more about how to model a system using lookup tables, see “Working with Lookup Tables” in the Simulink documentation.

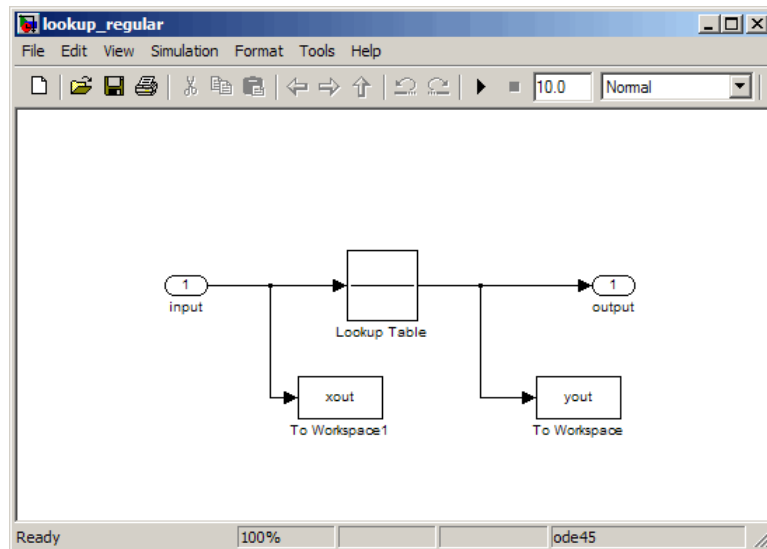
Configuring a Project for Parameter Estimation

To estimate the lookup table values, you must first configure a Control and Estimation Tools Manager project.

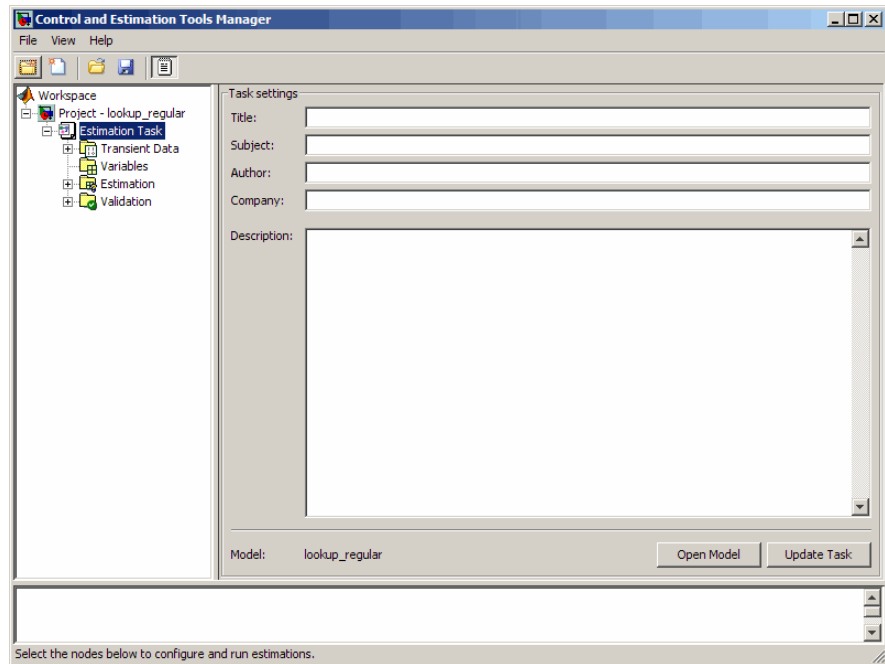
- 1 Open the lookup table model by typing the following command at the MATLAB prompt:

```
lookup_regular
```

This command opens the Simulink model, and loads the estimation data into the MATLAB workspace.



- In the Simulink model, select **Tools > Parameter Estimation** to open a new project named **lookup_regular** in the Control and Estimation Tools Manager GUI.



Estimating the Table Values Using Default Settings

After you configure a project for parameter estimation, as described in “Configuring a Project for Parameter Estimation” on page 5-6, use the following steps to estimate the lookup table values.

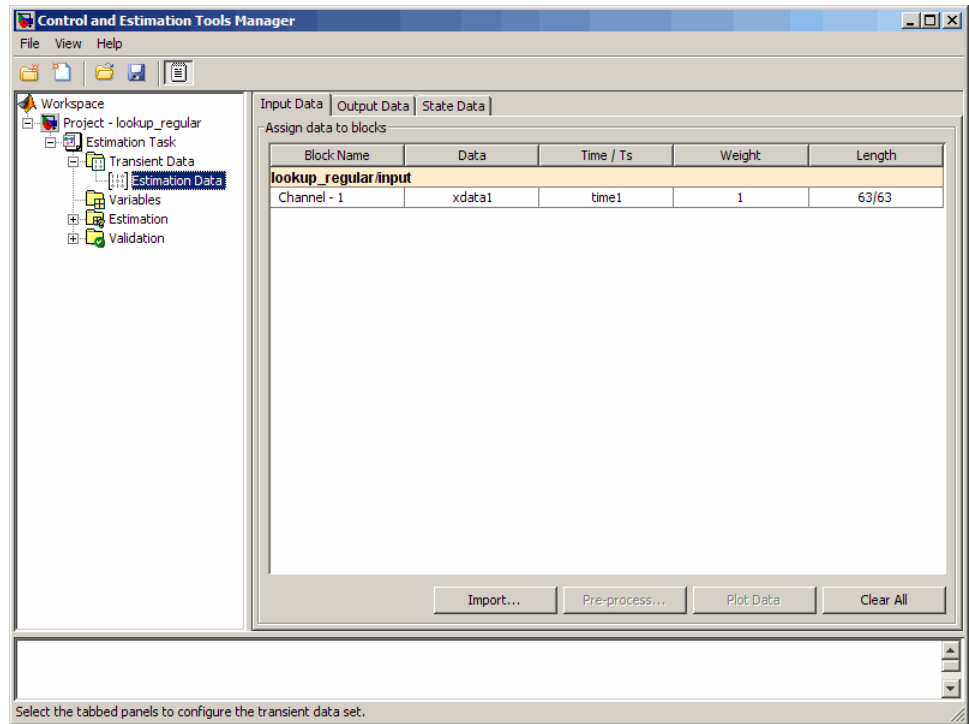
- Import the I/O data, `xdata1` and `ydata1`, and the time vector, `time1`, into the Control and Estimation Tools Manager GUI. For more information, see “Importing Data into the GUI” on page 1-5.

You can also load a preconfigured project that already contains the imported data. To do so, type the following command at the MATLAB prompt:

```
lookup_regular;
explorer.loadProject('lookup_regular_import', ...
```

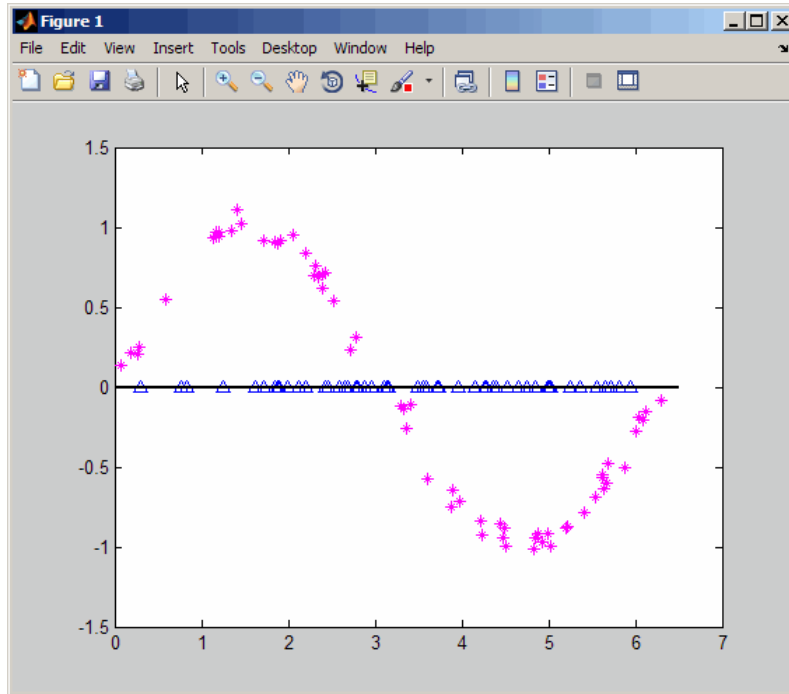


```
'Project - lookup_regular', 'Estimation Data');
```



- 2 Run an initial simulation to view the I/O data, simulated output, and the initial table values. To do so, type the following commands at the MATLAB prompt:

```
sim('lookup_regular')
figure(1); plot(xdata1,ydata1, 'm*', xout, yout, 'b^')
hold on; plot(linspace(0,6.5,10), table, 'k', 'LineWidth', 2)
```

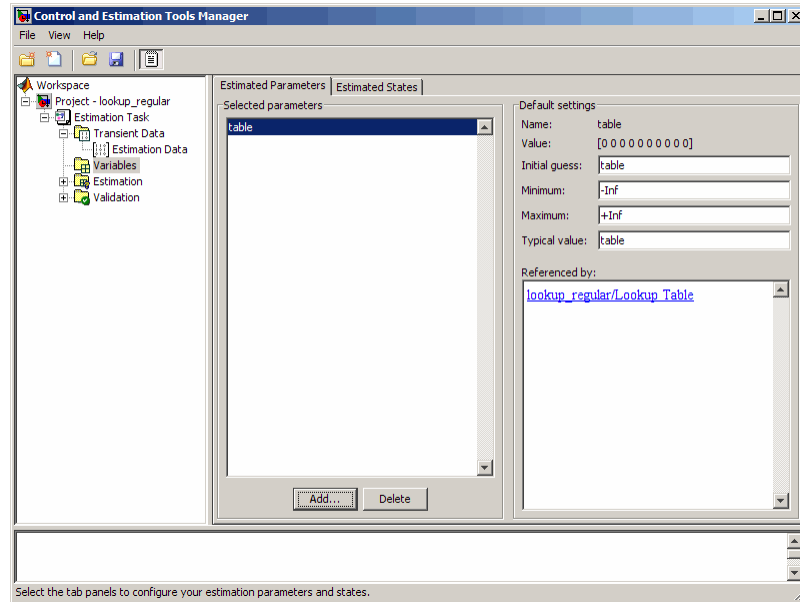


The x- and y-axes of the figure represent the input and output data, respectively. The figure shows the following plots:

- Measured data — Represented by the magenta stars (*).
- Initial table values — Represented by the black line.
- Initial simulation data — Represented by the blue deltas (Δ).

- 3 Select the table values to estimate.
 - a In the Control and Estimation Tools Manager GUI, select the **Variables** node under the **Estimation Task** node.

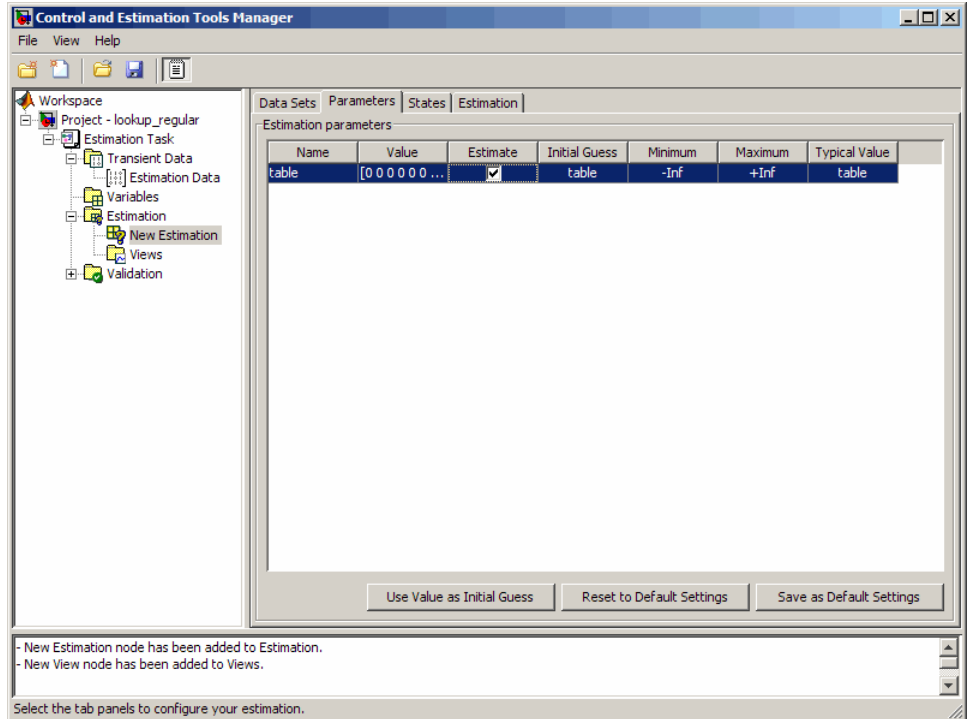
- b Click **Add** to open the Select Parameters dialog box, which shows the Simulink model parameters.
- c Select **table**, and click **OK** to add the table values to the **Estimated Parameters** tab.



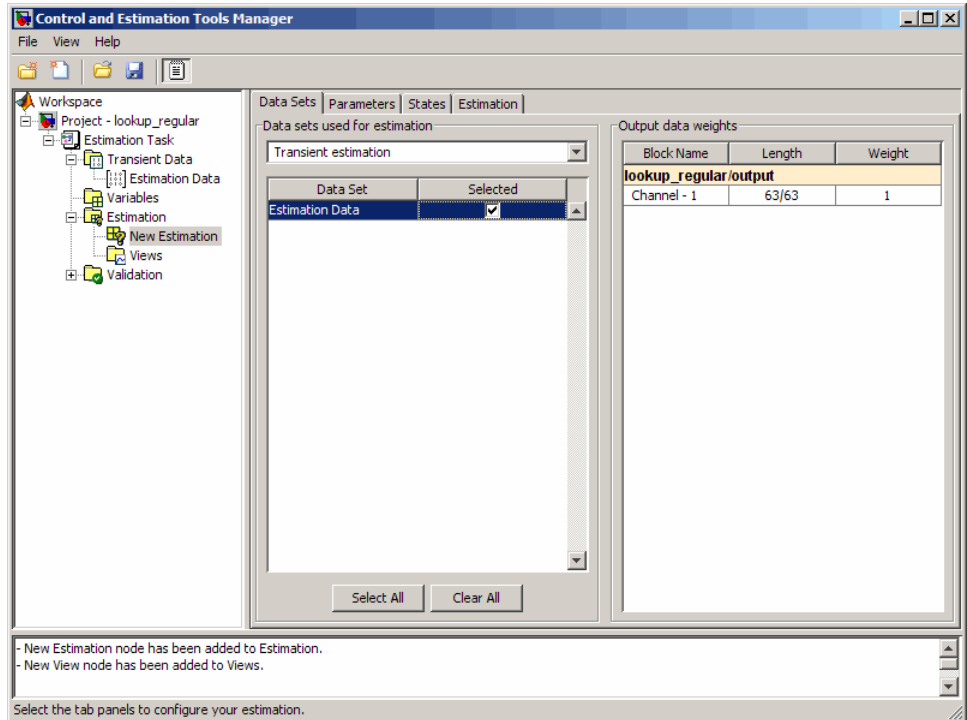
The **Default settings** area of the GUI displays the default settings for the table values. The **Value** field displays the initial table values, which comprise a vector of ten 0s.

- d Select the **Estimation** node, and click **New** to add a **New Estimation** node.

- e Select the **New Estimation** node. In the **Parameters** tab, select the **Estimate** check box to specify the lookup table values, table, for estimation.



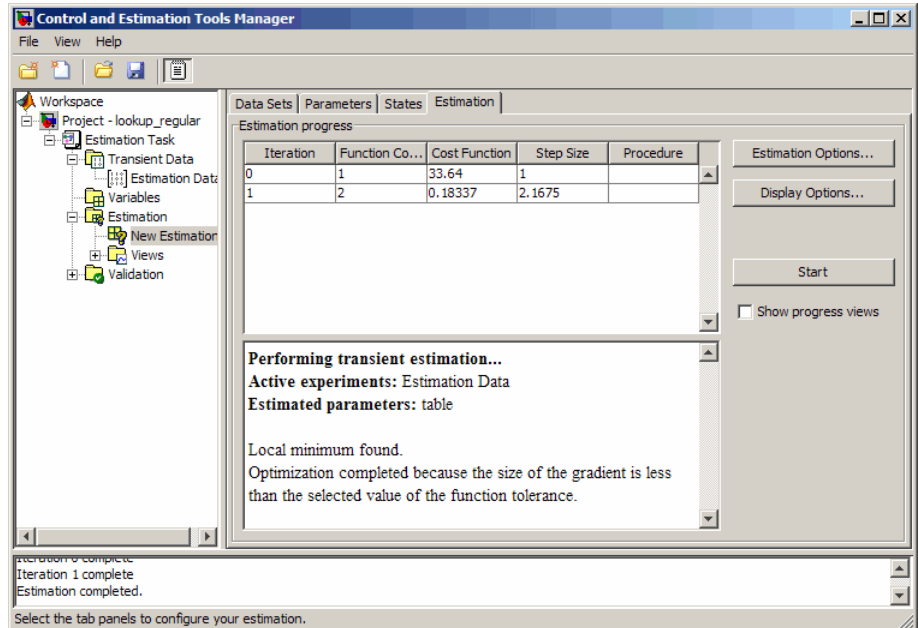
- 4** In the **Data Sets** tab of the **New Estimation** node, select the **Selected** check box to specify the estimation data set.



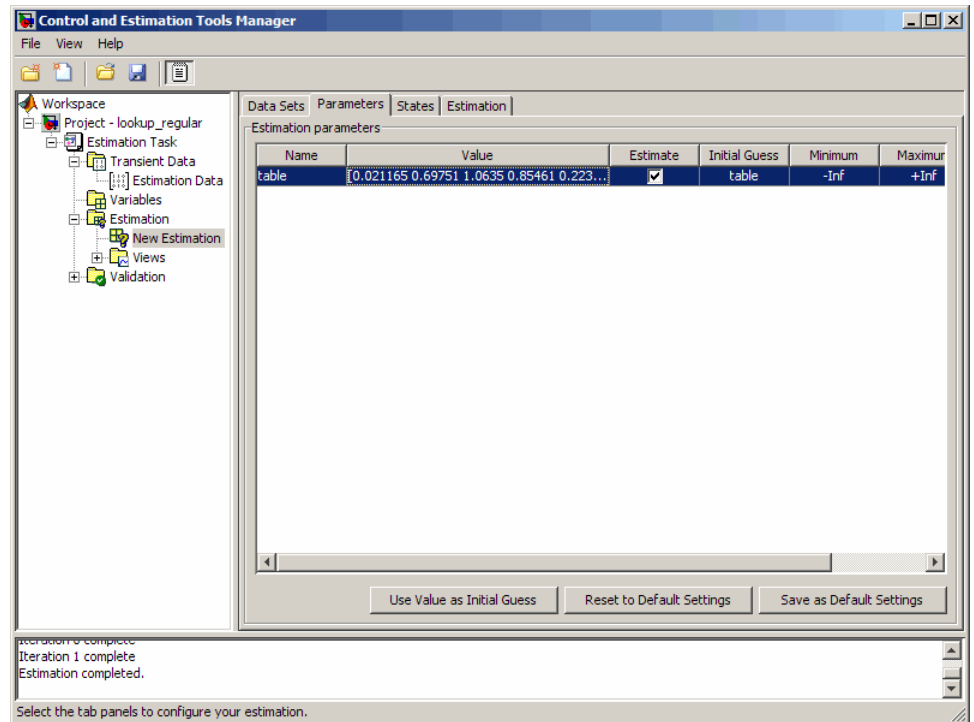
- 5** Estimate the table values using the default settings.

- a In the **Estimation** tab of the **New Estimation** node, click **Start** to start the estimation.

The Control and Tools Manager GUI updates at each iteration, and provides information about the estimation progress. After the estimation completes, the Control and Estimation Tools Manager GUI looks similar to the following figure.



- b** Select the **Parameters** tab in the **New Estimation** node to view the estimated table values, which appear in the **Value** field.



Validating the Estimation Results

After you estimate the table values, as described in “Estimating the Table Values Using Default Settings” on page 5-8, you must use another data set to validate that you have not overfitted the model. You plot and examine the following plots to validate the estimation results:

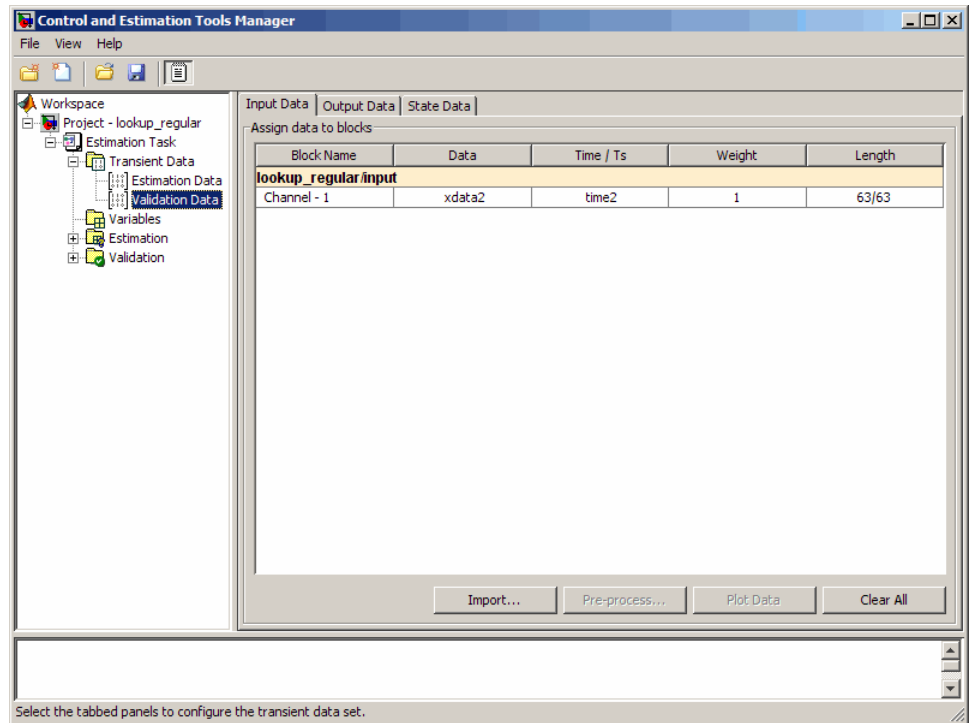
- Residuals plot
- Measured and simulated data plots

To validate the estimation results:

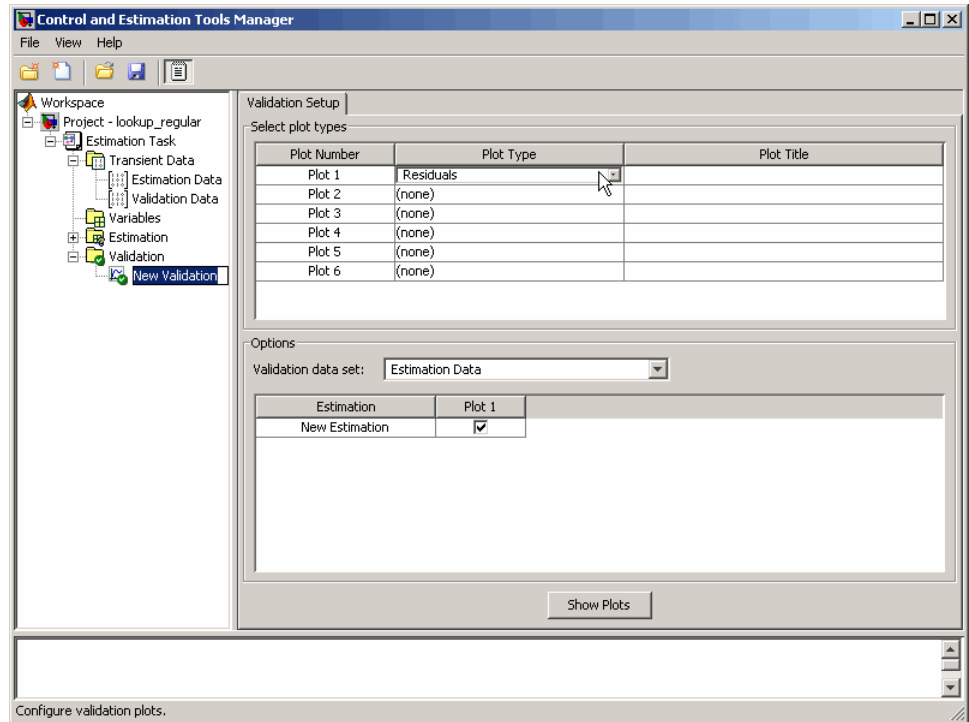
- 1 Import the validation I/O data, `xdata2` and `ydata2`, and time vector, `time2`, in the Control and Estimation Tools Manager GUI.

You can also load a project that already contains the estimated parameters, and the validation data set. To do so, type the following commands at the MATLAB prompt:

```
lookup_regular;
explorer.loadProject('lookup_regular_val',...
    'Project - lookup_regular', 'Validation Data');
```



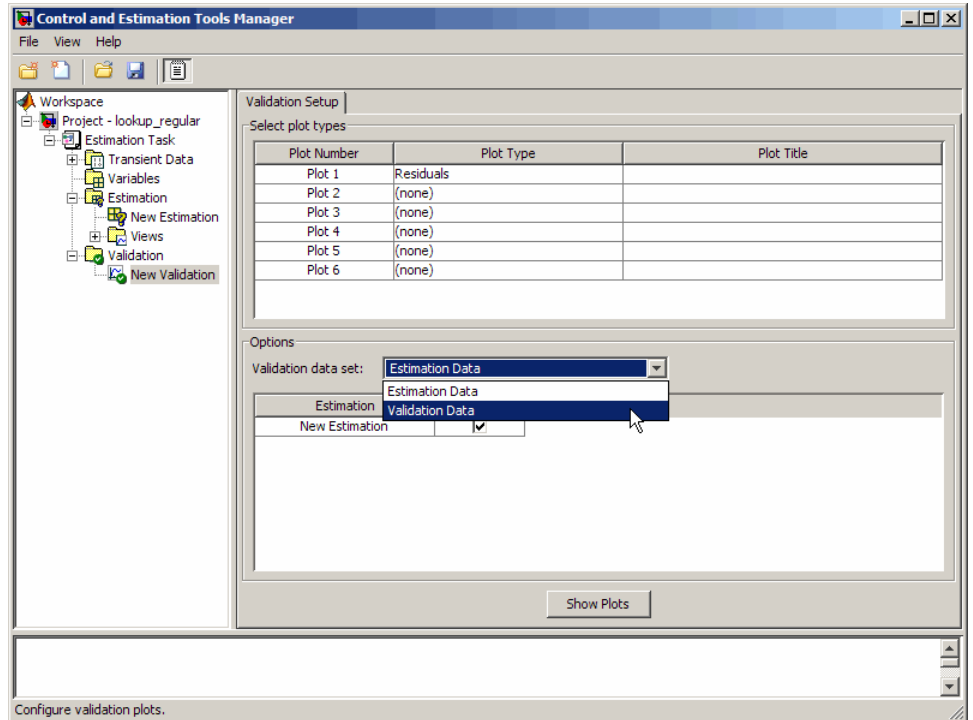
This project also contains the **Residuals** plot already configured in the **Select plot types** area of the GUI, as shown in the next figure. For more information on how to configure this plot, see “Performing Validation” on page 2-43.



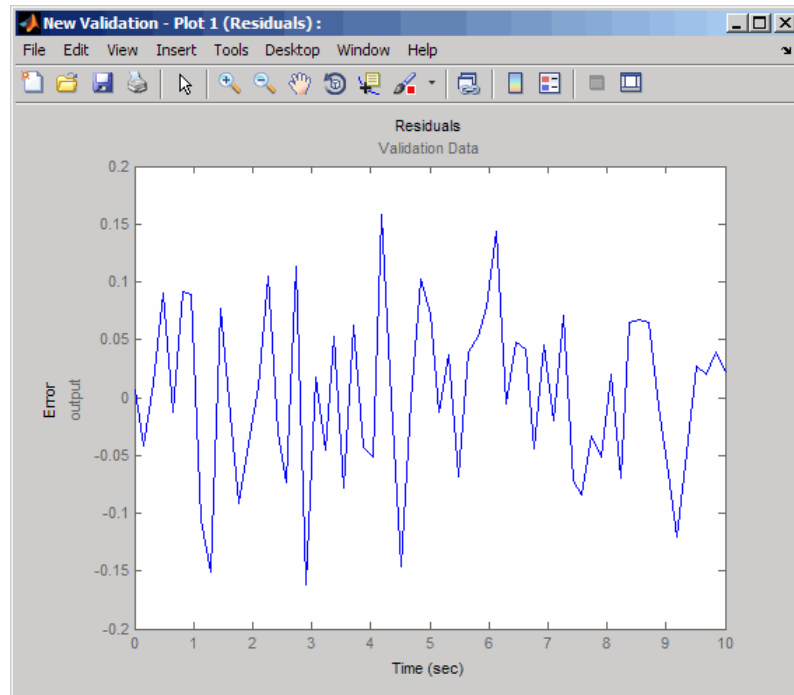
2 Plot and examine the residuals:

- Select the **New Validation** node under the **Validation** node.

- b In the **Options** area, select **Validation Data** from the **Validation data set** drop-down list.



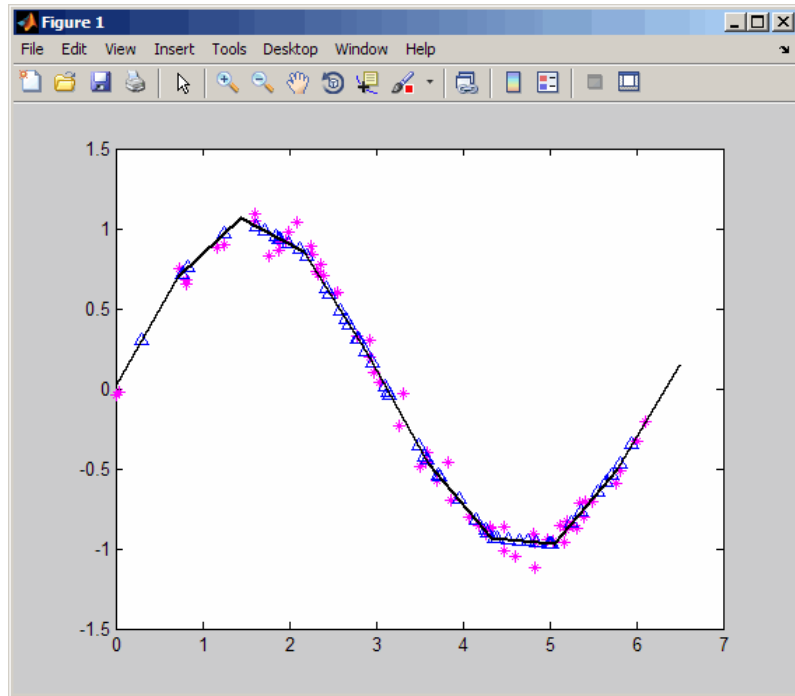
- c Click **Show Plots** to open the residuals plot.



The residuals, which show the difference between the simulated and measured data, lie in the range $[-0.15, 0.15]$ — within 15% of the maximum output variation. This indicates a good match between the measured and the simulated table data values.

- d Plot and examine the estimated table values against the validation data set and the simulated table values by typing the following commands at the MATLAB prompt.

```
sim('lookup_regular')
figure(2); plot(xdata2,ydata2, 'm*', xout, yout,'b^')
hold on; plot(linspace(0,6.5,10), table, 'k', 'LineWidth', 2)
```



The plot shows that the table values, displayed as the black line, match both the validation data and the simulated table values. The table data values cover the entire range of input values, which indicates that all the lookup table values have been estimated.

Example – Estimating Constrained Values of a Lookup Table

- “Objectives” on page 5-21

- “About the Data” on page 5-21
- “Configuring a Project for Parameter Estimation” on page 5-21
- “Estimating the Monotonically Increasing Table Values Using Default Settings” on page 5-24
- “Validating the Estimation Results” on page 5-31

Objectives

This example shows how to estimate constrained values of a lookup table. You apply monotonically increasing constraints to the lookup table values, and use the GUI to estimate the table values.

About the Data

In this example, you use `lookup_increasing.mat`, which contains the measured I/O data for estimating the lookup table values. The MAT-file includes the following variables:

- `xdata1` — Consists of 602 uniformly-sampled input data points in the range $[-5,5]$.
- `ydata1` — Output data corresponding to the input data samples.

Note The output data is a monotonically increasing function of the input data.

- `time1` — Time vector.

You use the I/O data to estimate the values of the lookup table in the `lookup_increasing` Simulink model. The table contains eleven values, which are stored in the MATLAB variable `table`. To learn more about how to specify the table’s values, see “Entering Breakpoints and Table Data” in the Simulink documentation.

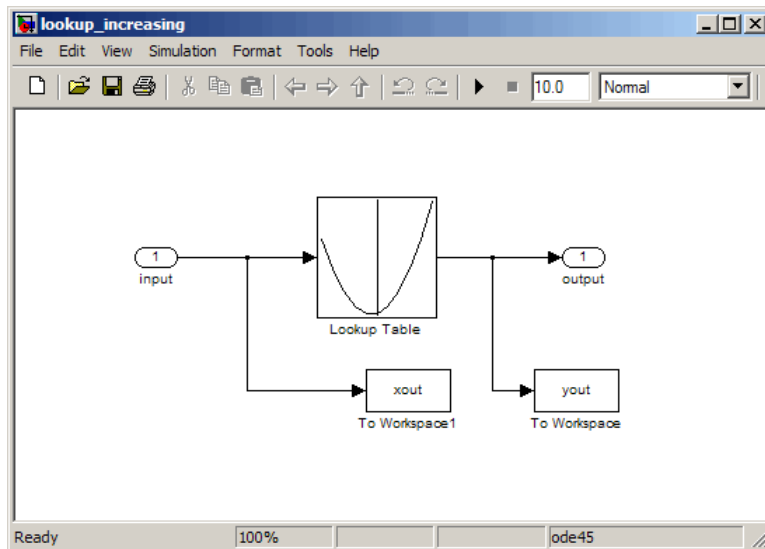
Configuring a Project for Parameter Estimation

To estimate the monotonically increasing lookup table values, you must first configure a Control and Estimation Tools Manager project.

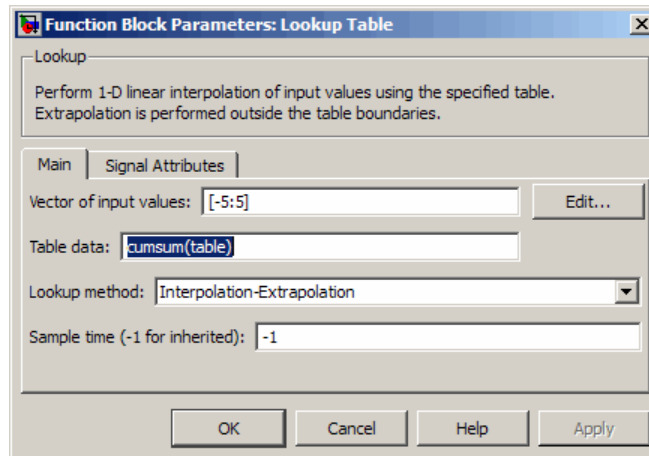
- 1 Open the lookup table model by typing the following command at the MATLAB prompt:

```
lookup_increasing
```

This command opens the Simulink model, and loads the estimation data in the MATLAB workspace.

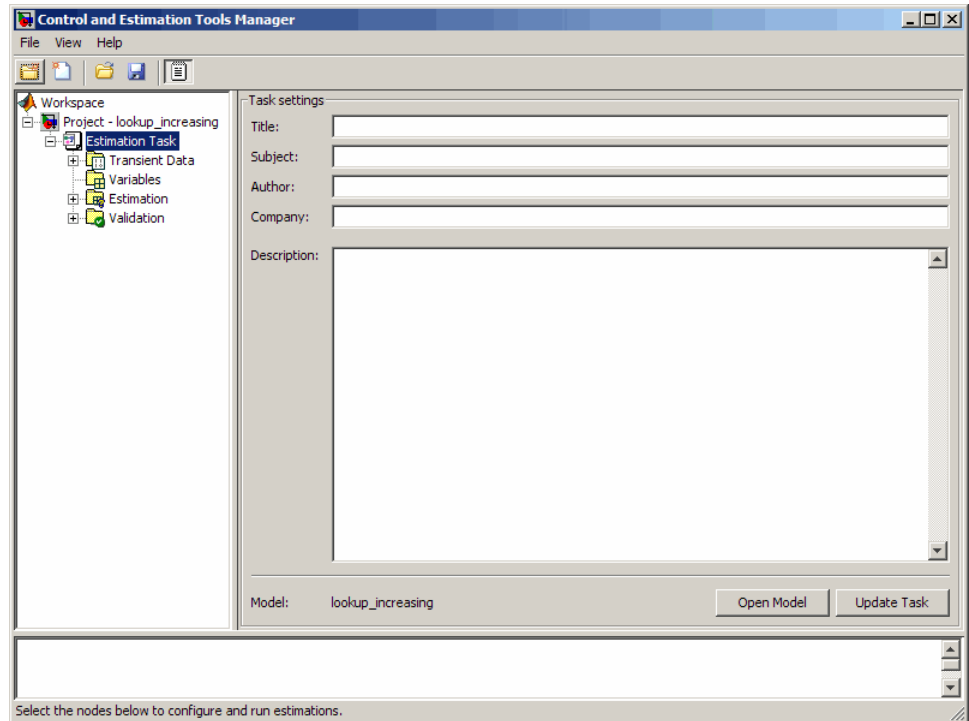


- 2 Double-click the Lookup Table block to view the monotonically increasing constraint applied to the table output values.



The **Table data** field of the Function Block Parameters dialog box shows the constraint. The cumulative sum function, `cumsum`, applies a monotonically increasing constraint on the table output values. This function computes the cumulative sum of the table values based on estimation of the individual table elements from the I/O data.

- 3 In the Simulink model, select **Tools > Parameter Estimation** to open a new project named **lookup_increasing** in the Control and Estimation Tools Manager GUI.



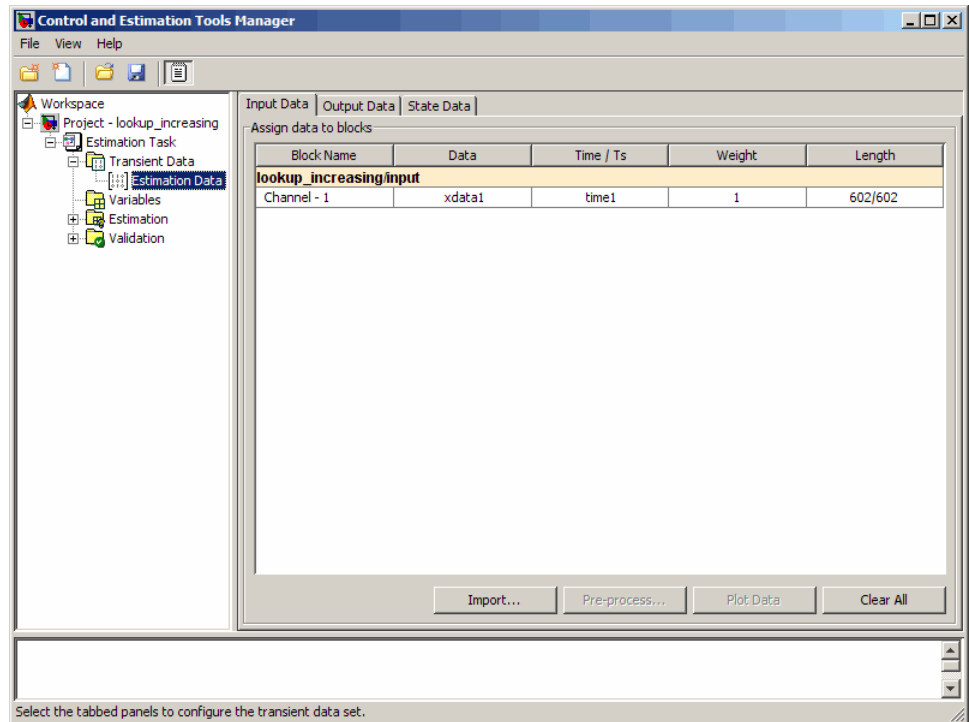
Estimating the Monotonically Increasing Table Values Using Default Settings

After you configure a project for parameter estimation, as described in “Configuring a Project for Parameter Estimation” on page 5-21, use the following steps to estimate the constrained lookup table values:

- 1 Import the estimation data, as described in the “Importing Data into the GUI” section of “Tutorial — Preparing Data for Parameter Estimation Using the GUI”.

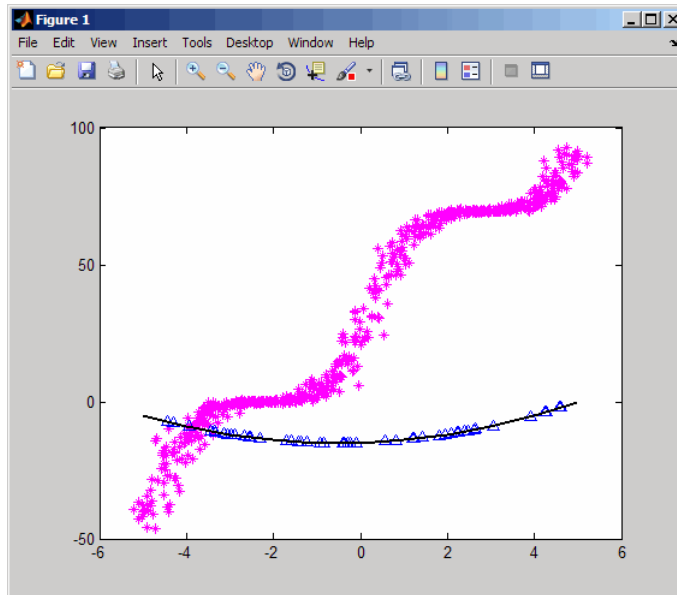
You can also load a preconfigured project that already contains the imported data. To do so, type the following commands at the MATLAB prompt:

```
lookup_increasing;  
explorer.loadProject('lookup_increasing_import',...  
    'Project - lookup_increasing','Estimation Data')
```



- 2 Run an initial simulation to view the measured data, simulated table values and the initial table values by typing the following commands at the MATLAB prompt:

```
sim('lookup_increasing')  
figure(1); plot(xdata1,ydata1, 'm*', xout, yout, 'b^')  
hold on; plot(-5:5, cumsum(table), 'k', 'LineWidth', 2)
```



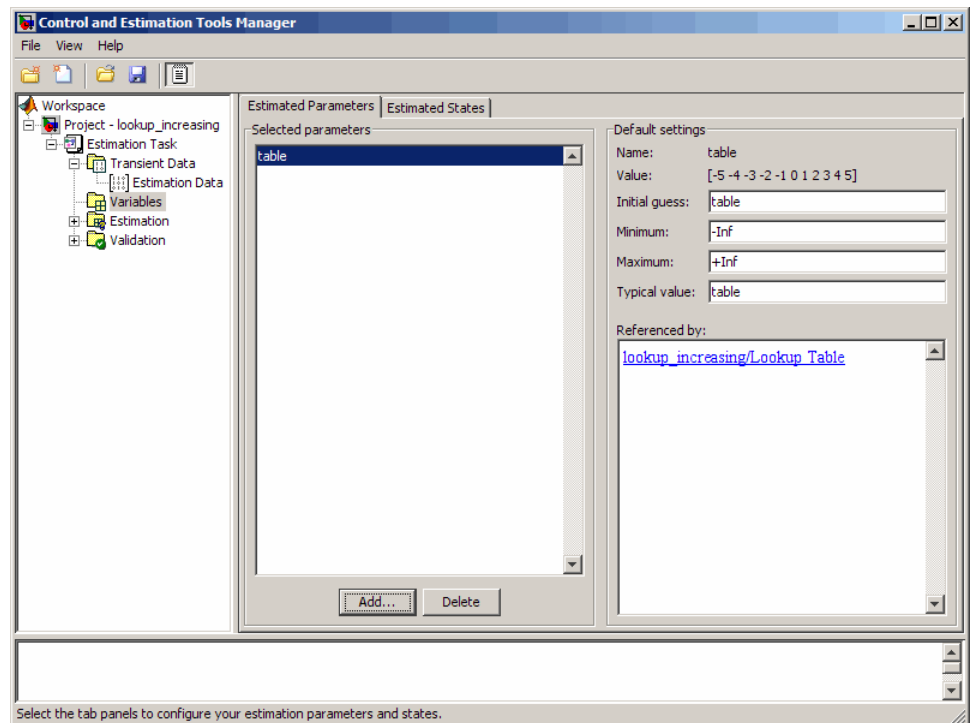
The x- and y-axes represents the input and output data, respectively. The figure shows the following plots:

- Measured data — Represented by the magenta stars (*).

Note As described in “About the Data” on page 5-21, the output data is a monotonically increasing function of the input data.

- Initial table values — Represented by the black line.
- Initial simulation data — Represented by the blue deltas (Δ).

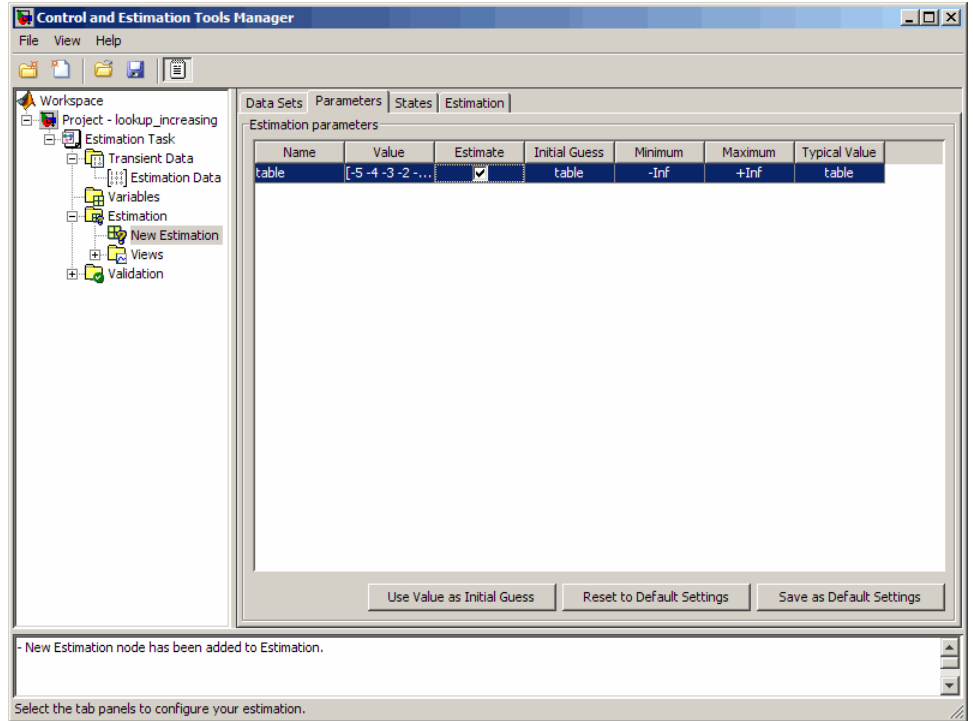
- 3 Select the table output values to estimate.
 - a In the Control and Estimation Tools Manager GUI, select the **Variables** node under the **Estimation Task** node.
 - b Click **Add** to open the Select Parameters dialog box, where you see the Simulink model parameters.
 - c Select **table**, and click **OK** to add the table values to the **Estimated Parameters** tab.



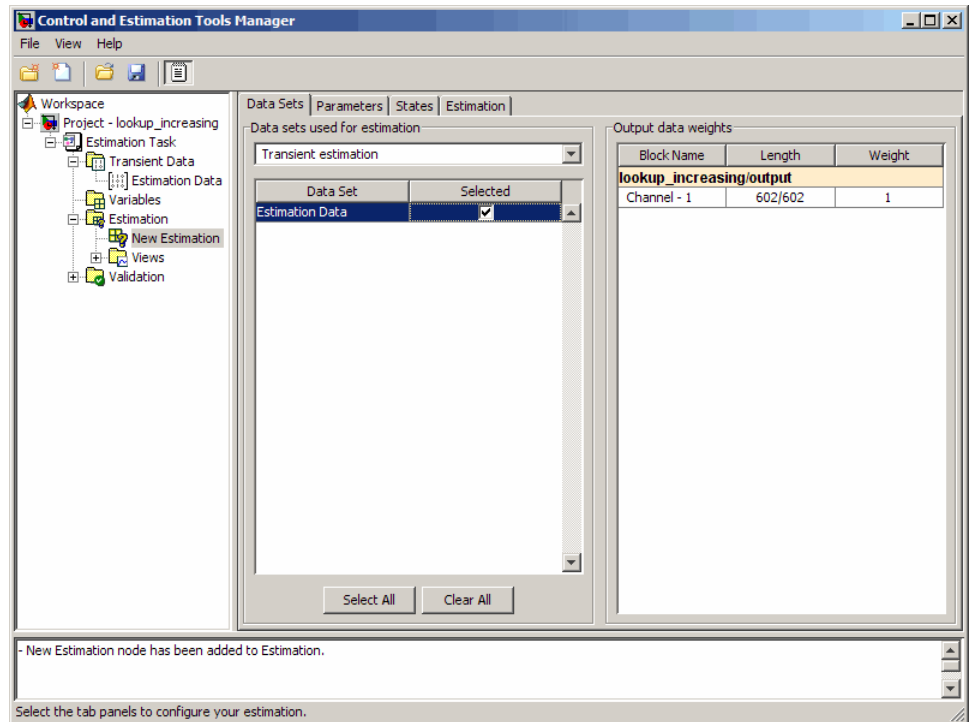
The **Default settings** area of the GUI displays the default settings for the table values. The **Value** field displays the initial table values.

- d Select the **Estimation** node, and click **New** to add a **New Estimation** node.

- e Select the **New Estimation** node. In the **Parameters** tab, select the **Estimate** check box to specify the lookup table values, table, for estimation.



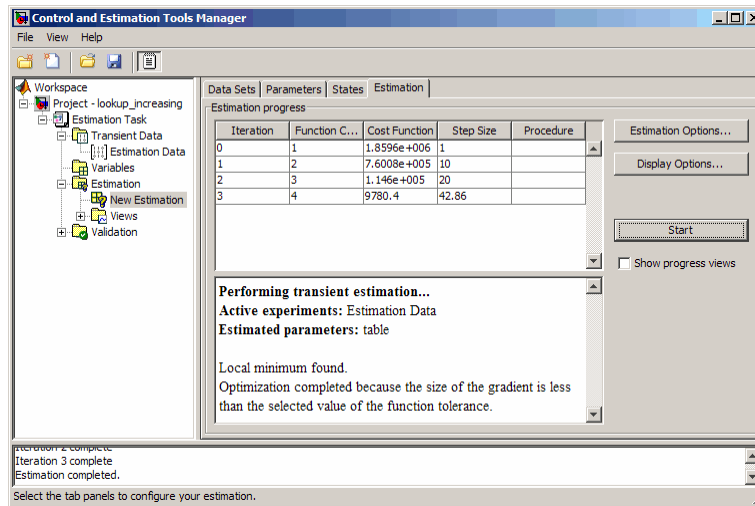
- 4 In the **Data Sets** tab of the **New Estimation** node, select the **Selected** check-box to specify the estimation data set.



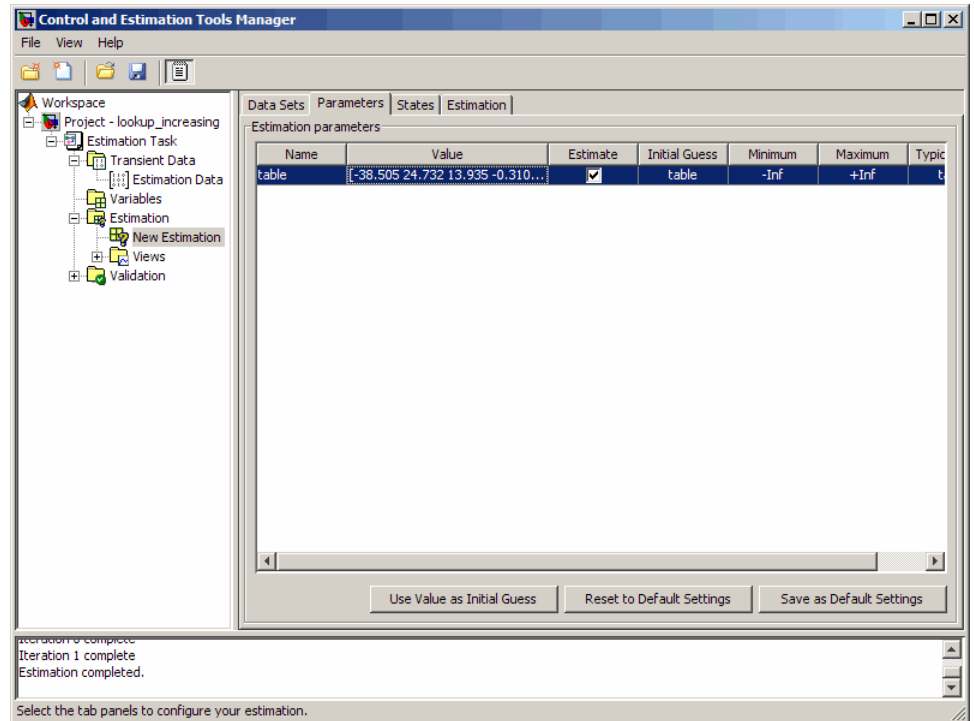
- 5 Estimate the parameters using the default settings.

- In the **Estimation** tab of the **New Estimation** node, click **Start** to start the estimation.

The Control and Tools Manager GUI updates at each iteration, and provides information about the estimation progress. After the estimation completes, the Control and Estimation Tools Manager GUI looks similar to the following figure.



- b** Select the **Parameters** tab of the **New Estimation** node to view the estimated table values. The **Value** field displays the estimated table values.



Validating the Estimation Results

After you estimate the table values, as described in “Estimating the Monotonically Increasing Table Values Using Default Settings” on page 5-24, you must use another data set to validate that you have not overfitted the model. You plot and examine the following plots to validate the estimation results:

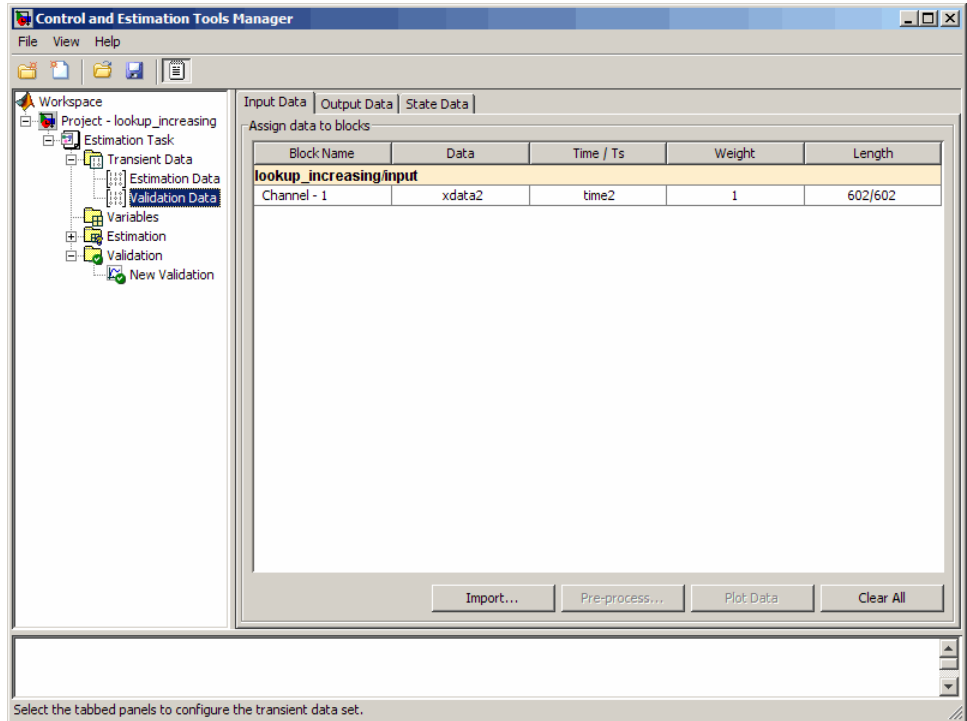
- Residuals plot
- Measured and simulated data plots

To validate the estimation results:

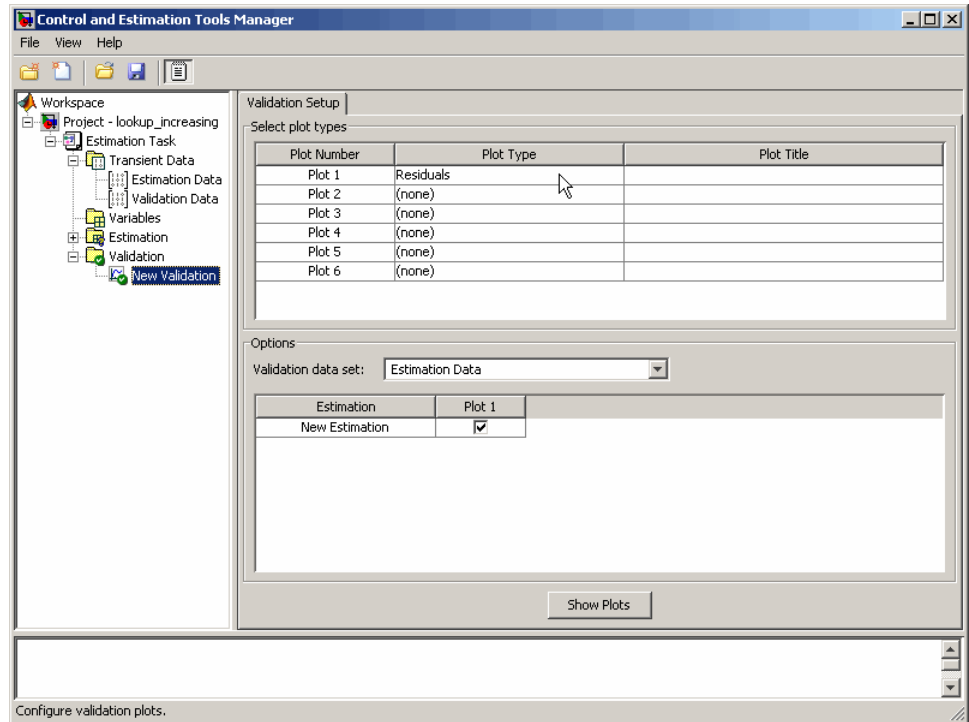
- 1 Import the validation I/O data, xdata2 and ydata2, and time vector, time2, in the Control and Estimation Tools Manager GUI.

You can load a project that already contains the estimated parameters, validation data set, and residuals plot. To do so, type the following commands at the MATLAB prompt:

```
lookup_increasing;
explorer.loadProject('lookup_increasing_val',...
    'Project - lookup_increasing', 'Validation Data')
```

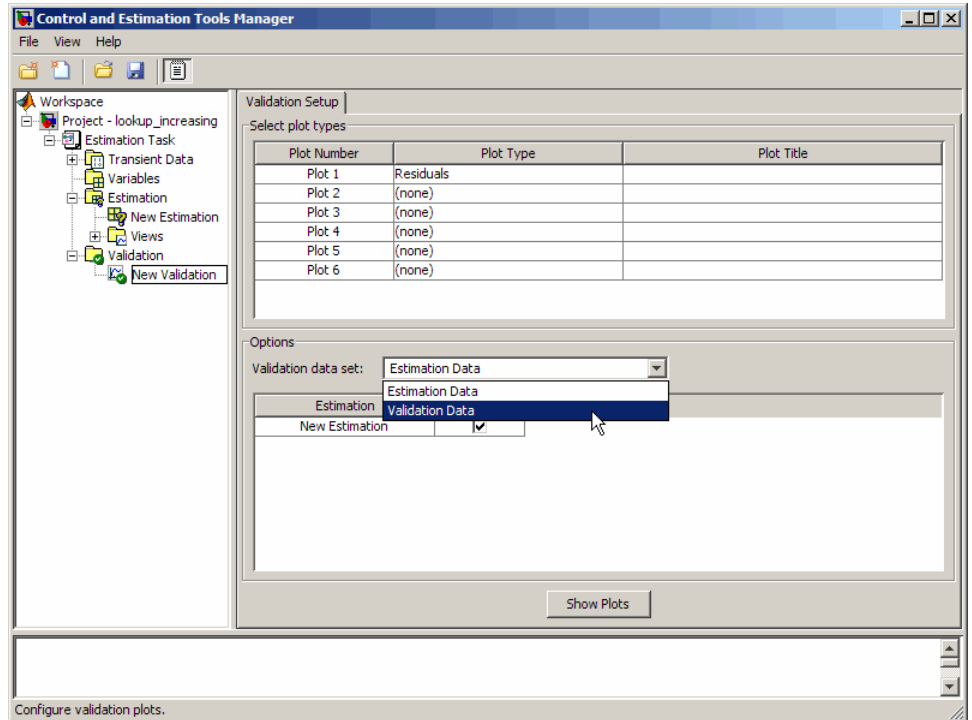


This project also contains the Residuals plot already configured in the **Select plot types** area of the GUI, as shown in the next figure. For more information on how to configure this plot, see “Performing Validation” on page 2-43.

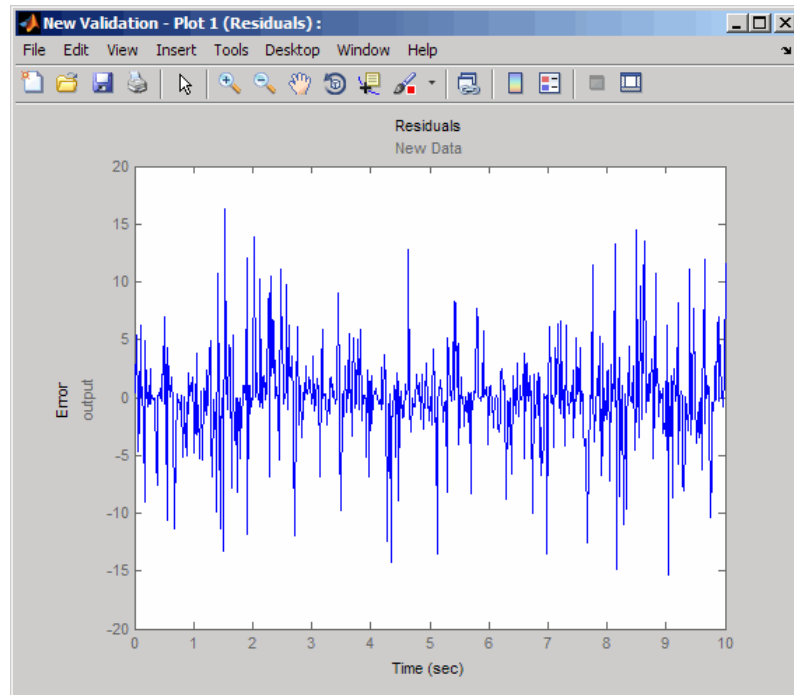


- 2 Plot and examine the residuals.
 - a Select the **New Validation** node under the **Validation** node.

- b In the **Options** area, select **Validation Data** from the **Validation data set** drop-down list.



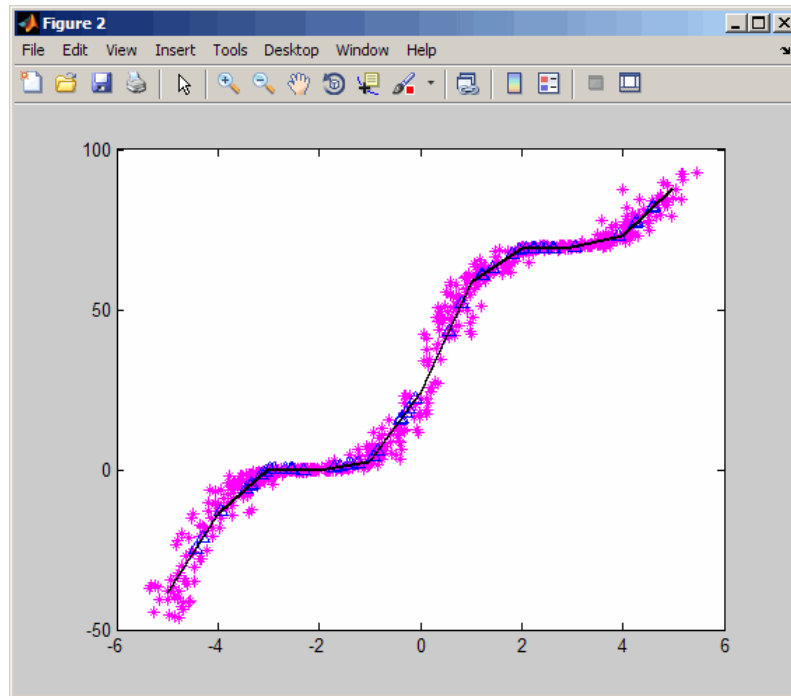
- c Click **Show Plots** to open the residuals plot.



The residuals, which show the difference between the simulated and measured data, lie the range $[-15,15]$ — within 20% of the maximum output variation. This indicates a good match between the measured and the simulated table data values.

- 3 Plot and examine the validation data, simulated data and estimated table values.

```
sim('lookup_increasing')  
figure(2); plot(xdata2,ydata2, 'm*', xout, yout, 'b^')  
hold on; plot(-5:5, cumsum(table), 'k', 'LineWidth', 2)
```



The plot shows that the table values, shown as the black line, match both the measured data and the simulated table values. The table data values cover the entire range of input values, which indicates that all the lookup table values have been estimated.

Capturing Time-Varying System Behavior Using Adaptive Lookup Tables

In this section...

“Building Models Using Adaptive Lookup Table Blocks” on page 5-37

“Configuring Adaptive Lookup Table Blocks” on page 5-41

“Example — Modeling an Engine Using n-D Adaptive Lookup Table” on page 5-44

“Using Adaptive Lookup Tables in Real-Time Environment” on page 5-60

Building Models Using Adaptive Lookup Table Blocks

Simulink Design Optimization software provides blocks for modeling systems as adaptive lookup tables. You can use the adaptive lookup table blocks to create lookup tables from measured or simulated data. You build a model using the adaptive lookup table blocks, and then simulate the model to adapt the lookup table values to the time-varying I/O data. During simulation, the software uses the input data to locate the table values, and then uses the output data to recalculate the table values. The updated table values are stored in the adaptive lookup table block. For more information, see “Adaptive Lookup Tables” on page 5-3.

The Adaptive Lookup Table library has the following three blocks:

- Adaptive Lookup Table (1D Stair-Fit) — One-dimensional adaptive lookup table
- Adaptive Lookup Table (2D Stair-Fit) — Two-dimensional adaptive lookup table
- Adaptive Lookup Table (nD Stair-Fit) — Multidimensional adaptive lookup table

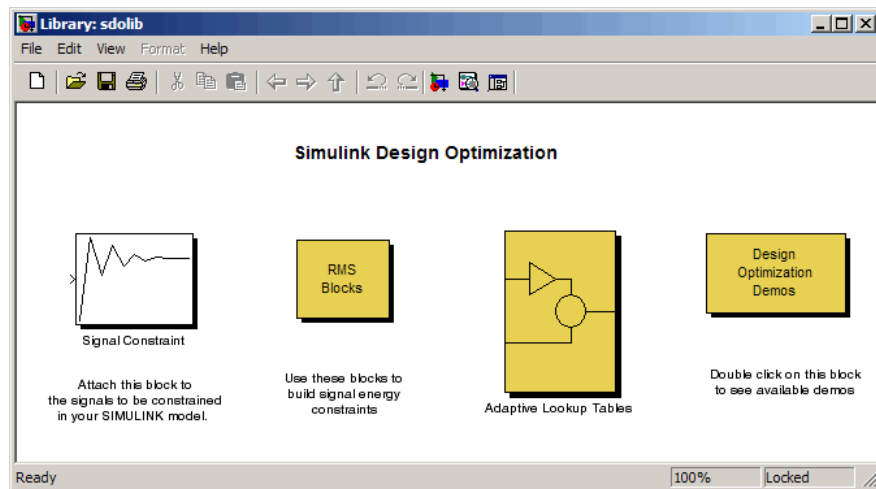
Note Use the n-D Adaptive Lookup Table block to create lookup tables of three or more dimensions.

To access the Adaptive Lookup Tables library:

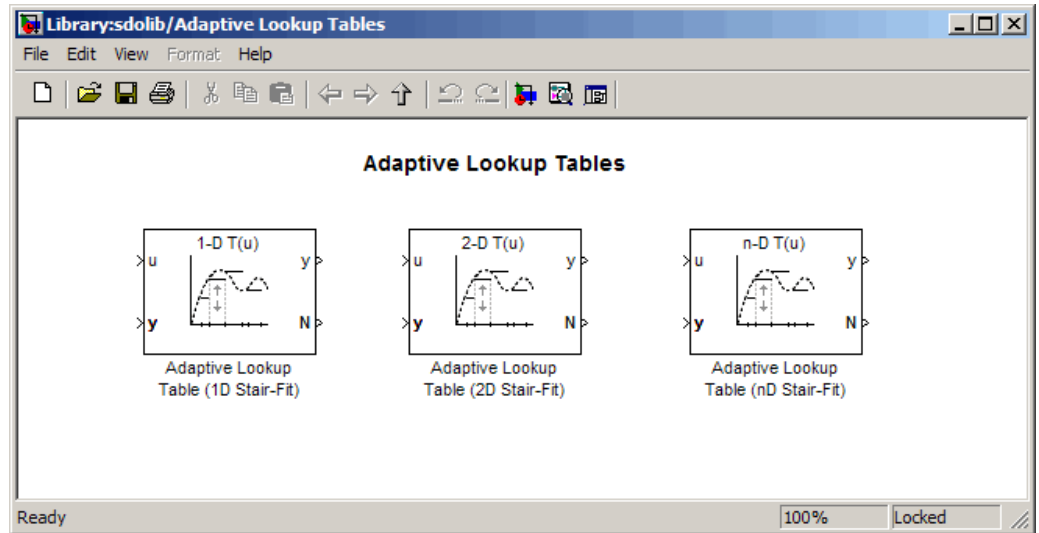
- 1 Type the following command at the MATLAB prompt:

```
sdolib
```

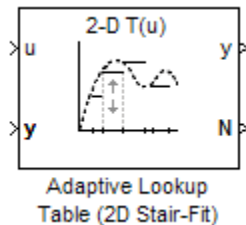
The Simulink Design Optimization library opens as shown in the next figure.



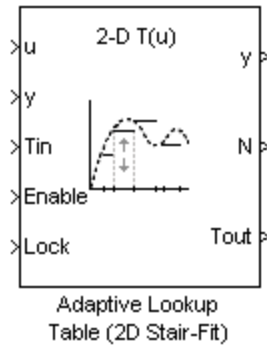
- 2 Double-click the Adaptive Lookup Tables block to open the Adaptive Lookup Tables library, as shown in the next figure.



By default, the Adaptive Lookup Table blocks have two inputs and outputs as shown in the next figure.



You can display additional inputs and outputs in a block by selecting the corresponding options in the Function Block Parameters dialog box. To learn more about the options, see Chapter 8, "Block Reference".



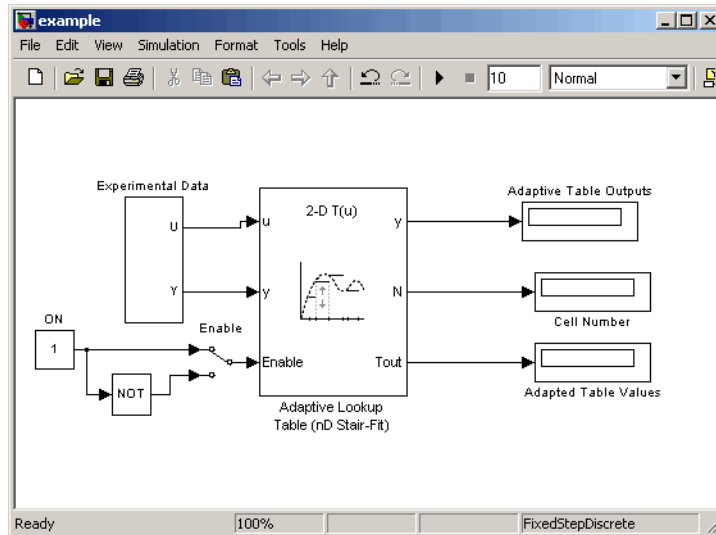
Adaptive Lookup Table Block Showing Inputs and Outputs

The 2-D Adaptive Lookup Table block has the following inputs and outputs:

- **u** and **y** — Input and output data of the system being modeled, respectively
 For example, to model an engine’s efficiency as a function of engine rpm and manifold pressure, specify **u** as the rpm, **y** as the pressure, and **y** as the efficiency signals.
- **Tin** — The initial table data
- **Enable** — Signal to enable, disable, or reset the adaptation process
- **Lock** — Signal to update only specified cells in the table
- **y** — Value of the cell currently being adapted
- **N** — Number of the cell currently being adapted
- **Tout** — Values of the adapted table data

For more information on how to use adaptive lookup tables, see “Example — Modeling an Engine Using n-D Adaptive Lookup Table” on page 5-44.

A typical Simulink diagram using an adaptive lookup table block is shown in the next figure.



Simulink® Diagram Using an Adaptive Lookup Table

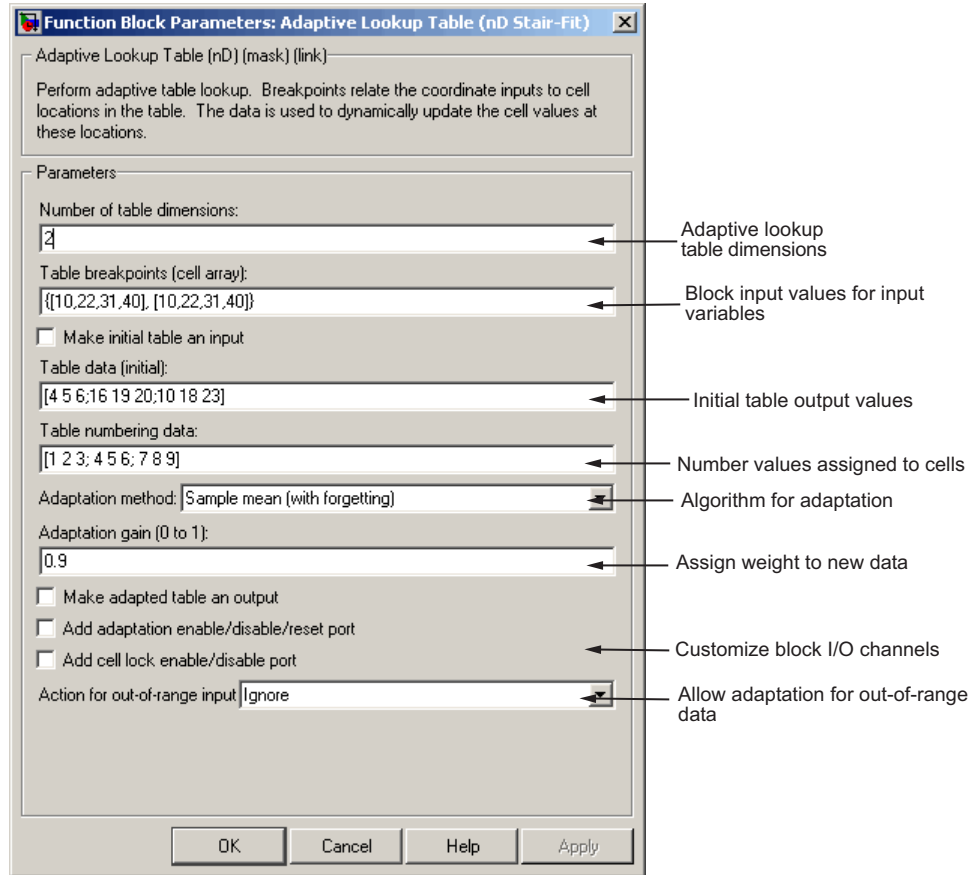
In this figure, the Experiment Data block imports a set of experimental data into Simulink through MATLAB workspace variables. The initial table is specified in the block mask parameters. When the simulation runs, the initial table begins to adapt to new data inputs and the resulting table is copied to the block's output.

Configuring Adaptive Lookup Table Blocks

- “Setting Adaptive Lookup Table Parameters” on page 5-41
- “Selecting an Adaptation Method” on page 5-42

Setting Adaptive Lookup Table Parameters

You can configure the Adaptive Lookup Table parameters in the Function Block Parameters dialog box. Double-click the block to open the dialog box shown in the next figure.



n-D Adaptive Lookup Table Dialog Box

For details on how to set these parameters, see the individual Chapter 8, “Block Reference” pages.

Selecting an Adaptation Method

You can select an adaptation algorithm from the **Adaptation Method** drop-down list in the Function Block Parameters dialog box. This section discusses the details of these algorithms.

Sample Mean. Sample mean provides the average value of n output data samples and is defined as:

$$\hat{y}(n) = \frac{1}{n} \sum_{i=1}^n y(i)$$

where $y(i)$ is the i^{th} measurement collected within a particular *cell*. For each input data u , the sample mean at the corresponding cell is updated using the output data measurement, y . Instead of accumulating n samples of data for each cell, a recursive relation is used to calculate the sample mean. The recursive expression is obtained by the following equation:

$$\hat{y}(n) = \frac{1}{n} \left[\sum_{i=1}^{n-1} y(i) + y(n) \right] = \frac{n-1}{n} \left[\frac{1}{n-1} \sum_{i=1}^{n-1} y(i) \right] + \frac{1}{n} y(n) = \frac{n-1}{n} \hat{y}(n-1) + \frac{1}{n} y(n)$$

where $y(n)$ is the n^{th} data sample.

Defining *a priori estimation error* as $e(n) = y(n) - \hat{y}(n-1)$, the recursive relation can be written as:

$$\hat{y}(n) = \hat{y}(n-1) + \frac{1}{n} e(n)$$

where $n \geq 1$ and the initial estimate $\hat{y}(0)$ is arbitrary.

In this expression, only the number of samples, n , for each cell—rather than n data samples—is stored in memory.

Sample Mean with Forgetting. The adaptation method “Sample Mean” on page 5-43 has an *infinite memory*. The past data samples have the same weight as the final sample in calculating the sample mean. Sample mean (with forgetting) uses an algorithm with a *forgetting factor* or **Adaptation gain** that puts more weight on the more recent samples. This algorithm provides robustness against initial response transients of the plant and an adjustable speed of adaptation. Sample mean (with forgetting) is defined as:

$$\begin{aligned}\hat{y}(n) &= \frac{1}{\sum_{i=1}^n \lambda^{n-i}} \sum_{i=1}^n \lambda^{n-i} y(i) \\ &= \frac{1}{\sum_{i=1}^n \lambda^{n-i}} \left[\sum_{i=1}^{n-1} \lambda^{n-i} y(i) + y(n) \right] = \frac{s(n-1)}{s(n)} \hat{y}(n-1) + \frac{1}{s(n)} y(n)\end{aligned}$$

where $\lambda \in [0,1]$ is the **Adaptation gain** and $s(k) = \sum_{i=1}^k \lambda^{n-i}$.

Defining *a priori estimation error* as $e(n) = y(n) - \hat{y}(n-1)$, where $n \geq 1$ and the initial estimate $\hat{y}(0)$ is arbitrary, the recursive relation can be written as:

$$\hat{y}(n) = \hat{y}(n-1) + \frac{1}{s(n)} e(n) = \hat{y}(n-1) + \frac{1-\lambda}{1-\lambda^n} e(n)$$

A small value of λ results in faster adaptation. A value of 0 indicates short memory (last data becomes the table value), and a value of 1 indicates long memory (average all data received in a cell).

Example – Modeling an Engine Using n-D Adaptive Lookup Table

- “Objectives” on page 5-44
- “About the Data” on page 5-45
- “Building a Model Using Adaptive Lookup Table Blocks” on page 5-45
- “Adapting the Lookup Table Values Using Time-Varying I/O Data” on page 5-57

Objectives

In this example, you learn how to capture the time-varying behavior of an engine using an n-D adaptive lookup table. You accomplish the following tasks using the Simulink software:

- Configure an adaptive lookup table block to model your system.
- Simulate the model to update the lookup table values dynamically.
- Export the adapted lookup table values to the MATLAB workspace.
- Lock a specific cell in the table during adaptation.
- Disable the adaptation process and use the adaptive lookup table as a static lookup table.

About the Data

In this example, you use the data in `vedata.mat` which contains the following variables measured from an engine:

- `X` — 10 input breakpoints for intake manifold pressure in the range [10,100]
- `Y` — 36 input breakpoints for engine speed in the range [0,7000]
- `Z` — 10x36 matrix of table data for engine volumetric efficiency

To learn more about breakpoints and table data, see “Anatomy of a Lookup Table” in the Simulink documentation.

The output volumetric efficiency of the engine is time varying, and a function of two inputs—intake manifold pressure and engine speed. The data in the MAT-file is used to generate the time-varying input and output (I/O) data for the engine.

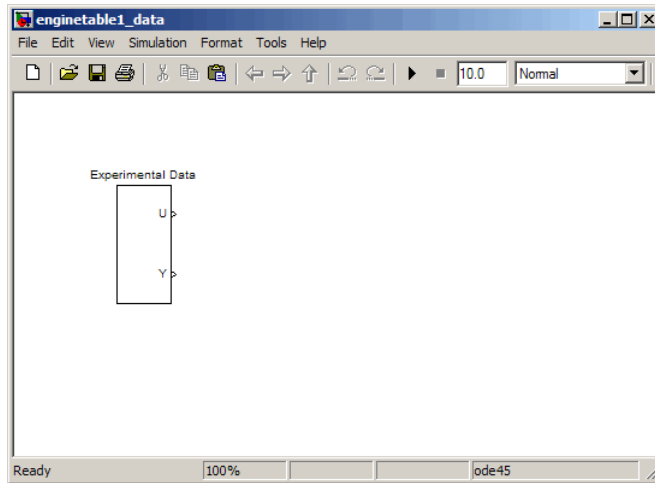
Building a Model Using Adaptive Lookup Table Blocks

In this portion of the tutorial, you learn how to build a model of an engine using an Adaptive Lookup Table block.

- 1 Open a preconfigured Simulink model by typing the model name at the MATLAB prompt:

```
enginetable1_data
```

The Experimental Data subsystem in the Simulink model generates time-varying I/O data during simulation.

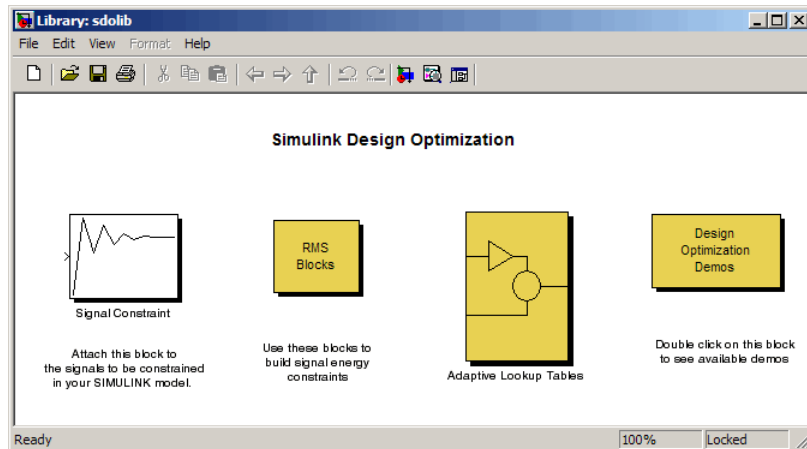


This command also loads the variables X, Y and Z into the MATLAB workspace. To learn more about this data, see “About the Data” on page 5-45.

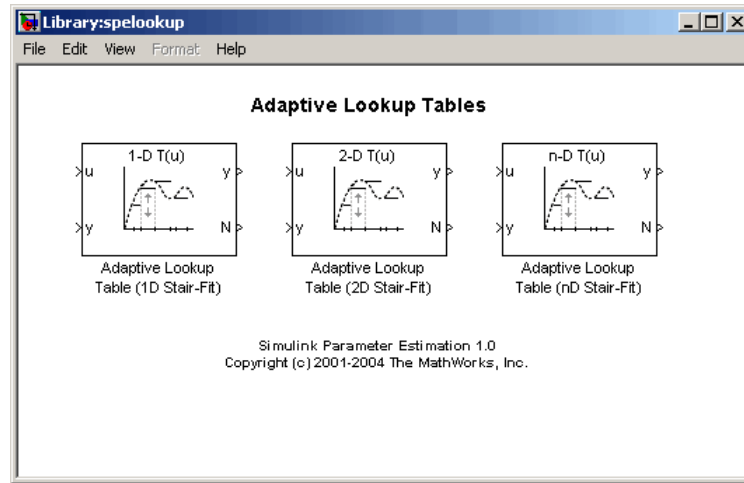
2 Add an Adaptive Lookup Table block to the Simulink model.

- Open the Simulink Design Optimization library by typing the following command at the MATLAB prompt:

```
sdolib
```

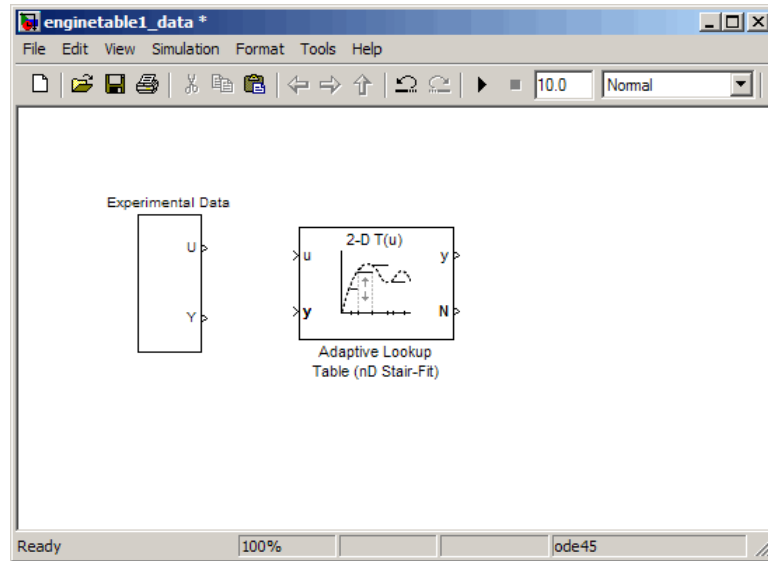


- b** Double-click the Adaptive Lookup Tables block to open the Adaptive Lookup Tables library.

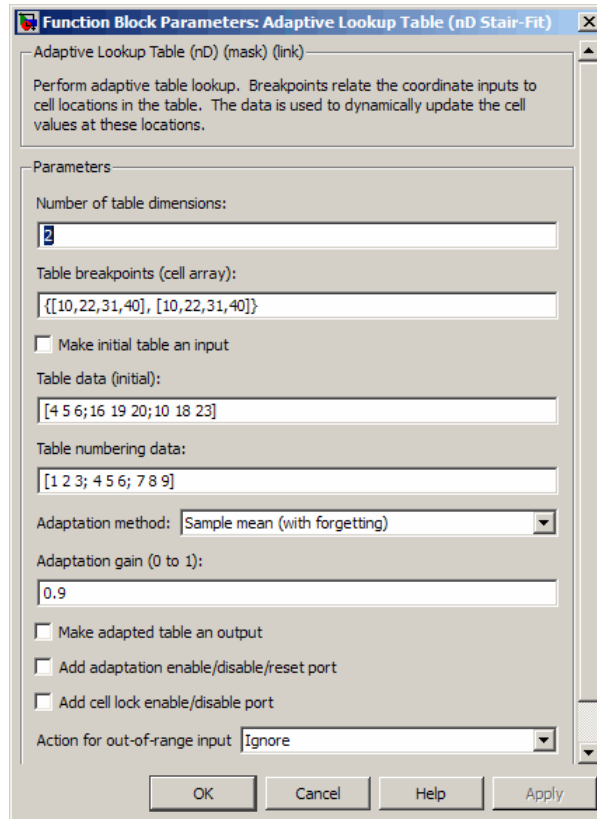


Simulink Design Optimization software provides three Adaptive Lookup Table blocks. In this example, you use the Adaptive Lookup Table (nD Stair-Fit) block to model your system. To learn more about the blocks, see Chapter 8, “Block Reference”.

- c** Drag and drop the Adaptive Lookup Table (nD Stair-Fit) block from the Simulink Design Optimization library to the Simulink model window.



- 3 Double-click the Adaptive Lookup Table (nD Stair-Fit) block to open the Function Block Parameters: Adaptive Lookup Table (nD Stair-Fit) dialog box.



4 In the Function Block Parameters dialog box:

- a Specify the following block parameters:
 - **Table breakpoints (cell array)** — Enter `{[X; 110], [Y; 7200]}` to specify the range of input breakpoints.
 - **Table data (initial)** — Enter `rand(10,36)` to specify random numbers as the initial table values for the volumetric efficiency.
 - **Table numbering data** — Enter `reshape(1:360,10,36)` to specify a numbering scheme for the table cells.
- b Verify that **Sample mean (with forgetting)** is selected in the **Adaptation method** drop-down list.

- c** Enter 0.98 in the **Adaptation gain (0 to 1)** field to specify the *forgetting factor* for the Sample mean (with forgetting) adaptation algorithm.

An adaptation gain close to 1 indicates high robustness of the lookup table values to input noise. To learn more about the adaptation gain, see “Sample Mean with Forgetting” on page 5-43 in “Selecting an Adaptation Method” on page 5-42.

- d** Select the **Make adapted table an output** check box.

This action adds a new port named `Tout` to the Adaptive Lookup Table block. You use this port to plot the table values as they are being adapted.

- e** Select the **Add adaptation enable/disable/reset port** check box.

This action adds a new port named `Enable` to the Adaptive Lookup Table block. You use this port to enable or disable the adaptation process.

- f** Select the **Add cell lock enable/disable port** check box.

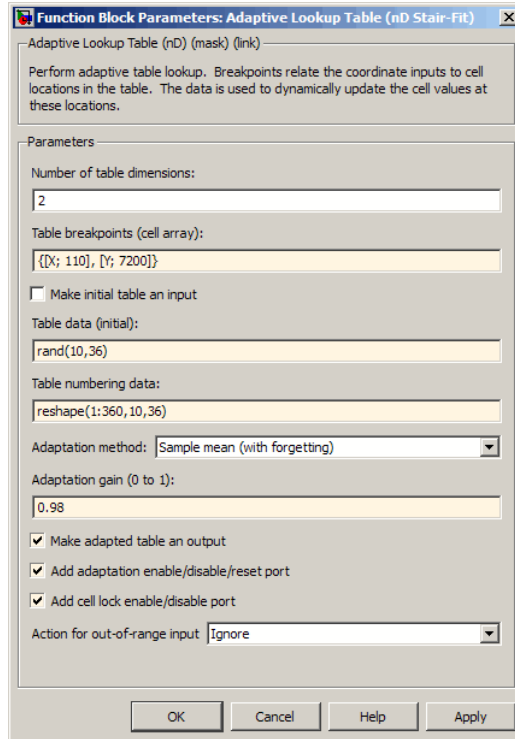
This action adds a new port named `Lock` to the Adaptive Lookup Table block. You use this port to lock a cell during the adaptation process.

- g** Verify that `Ignore` is selected in the **Action for out-of-range** drop-down list.

This selection specifies that the software ignores any time-varying inputs outside the range of input breakpoints during adaptation.

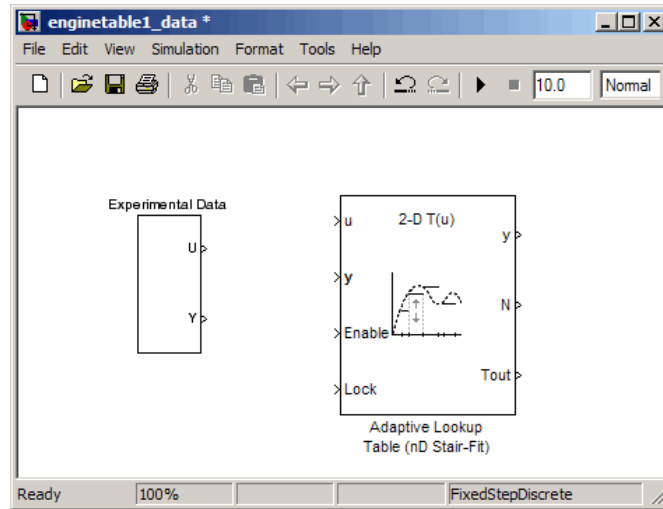
Tip To learn more about the Adaptive Lookup Table (nD Stair-Fit) block parameters, see the Adaptive Lookup Table (nD Stair-Fit) block reference page.

After you configure the parameters, the block parameters dialog box looks like the following figure.

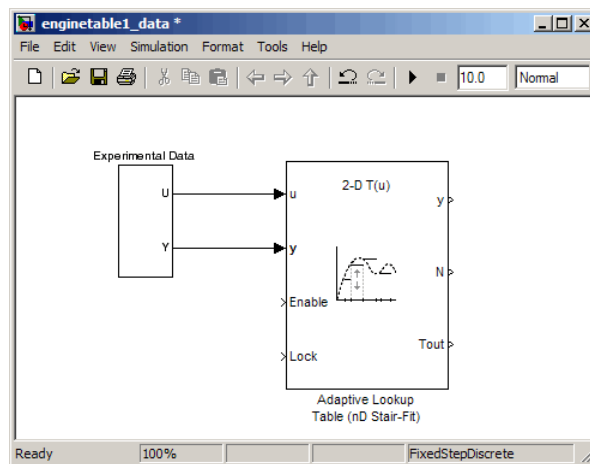


h Click **OK** to close the Function Block Parameters dialog box.

The Simulink model now looks similar to the following figure.

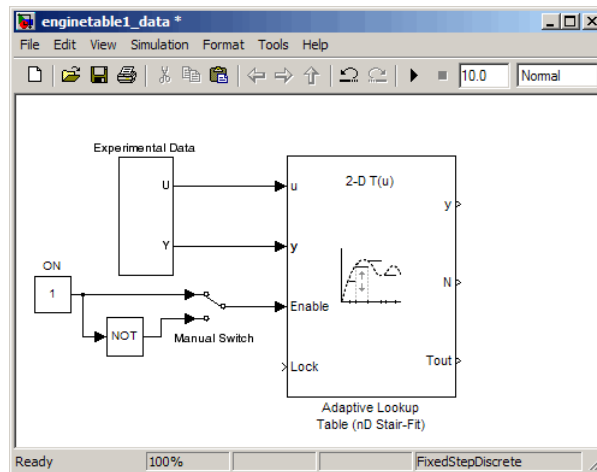


- 5 Assign the input and output data to the engine model by connecting the U and Y ports of the Experimental Data block to the u and y ports of the Adaptive Lookup Table block, respectively.



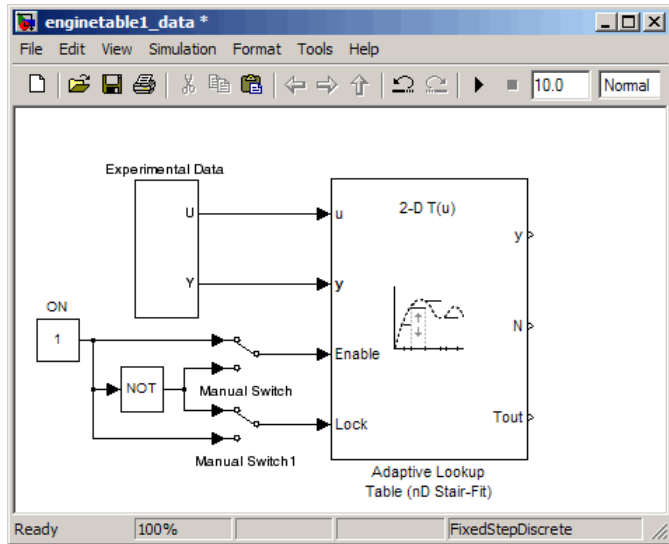
Tip To learn how to connect blocks in the Simulink model window, see “Connecting Blocks in the Model Window” in the Simulink documentation.

- 6 Design a logic using Simulink blocks to enable or disable the adaptation process. Connect the logic to the Adaptive Lookup Table block, as shown in the following figure.

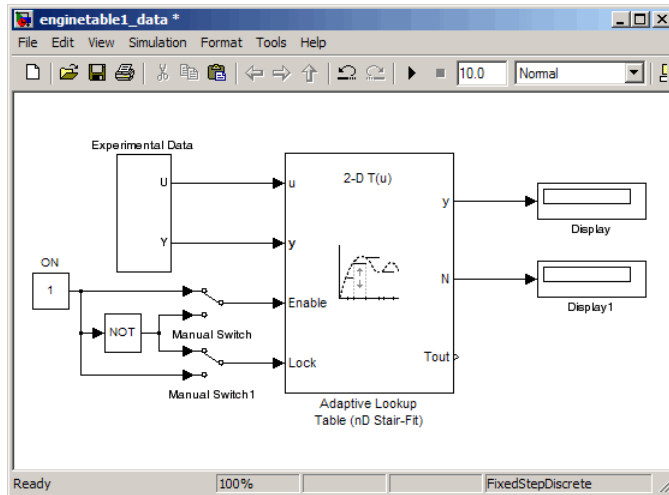


This logic outputs an initial value of 1 which enables the adaptation process.

- 7 Design a logic to lock a cell during adaptation. Connect the logic to the Adaptive Lookup Table block, as shown in the following figure.



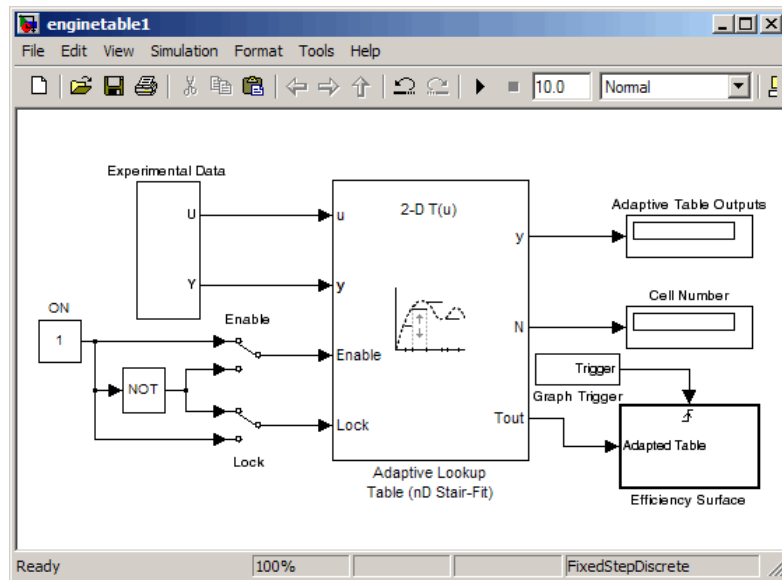
- 8 In the Simulink Library Browser, select the **Simulink > Sinks** library, and drag **Display** blocks to the model window. Connect the blocks, as shown in the following figure.



During simulation, the Display blocks show the following:

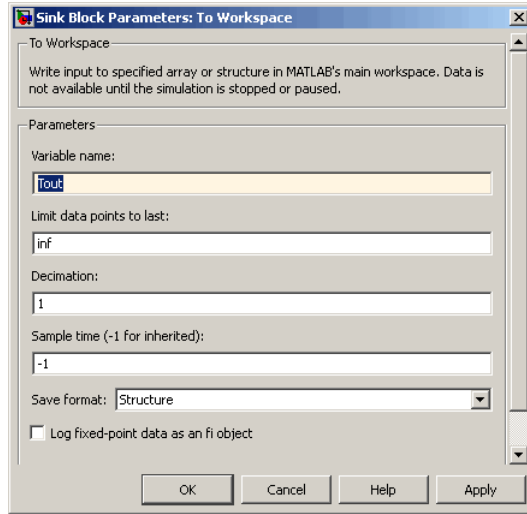
- Display block — Shows the value of the current cell being adapted.
 - Display1 block — Shows the number of the current cell being adapted.
- 9 Write a MATLAB function to plot the lookup table values as they adapt during simulation.

Alternatively, you can type `enginetable` at the MATLAB prompt to open a preconfigured Simulink model. The **Efficiency Surface** subsystem contains a function to plot the lookup table values, as shown in the next figure.

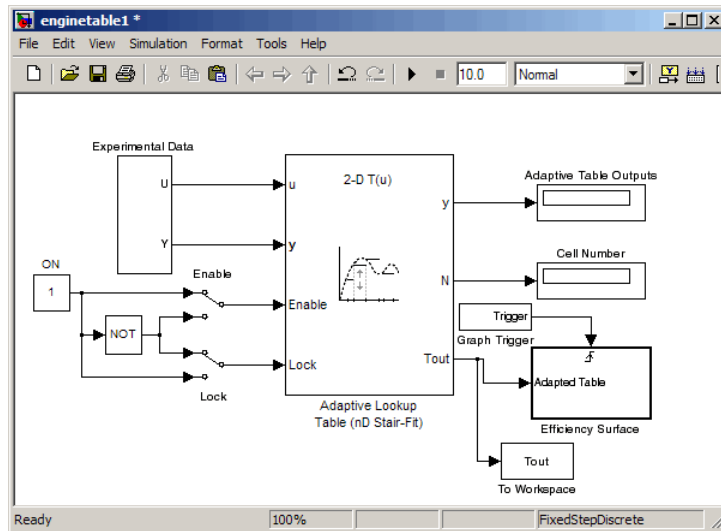


- 10 Connect a To Workspace block to export the adapted table values:
- a In the Simulink Library Browser, select the **Simulink > Sinks** library, and drag the To Workspace block to the model window.

To learn more about this block, see the To Workspace block reference page in the Simulink documentation.
 - b Double-click the To Workspace block to open the Sink Block Parameters dialog box, and type `Tout` in the **Variable name** field.



- c Click **OK**.
- d Connect the To Workspace block to the adaptive lookup table output signal **Tout**, as shown in the next figure.



You have now built the model for updating and viewing the adaptive lookup table values. You must now simulate the model to start the adaptation, as described in “Adapting the Lookup Table Values Using Time-Varying I/O Data” on page 5-57.

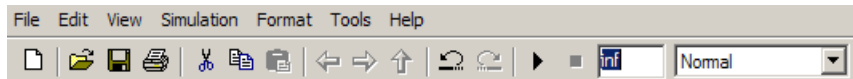
Adapting the Lookup Table Values Using Time-Varying I/O Data

In this portion of the tutorial, you learn how to update the lookup table values to adapt to the time-varying input and output values.

You must have already built the Simulink model, as described “Building a Model Using Adaptive Lookup Table Blocks” on page 5-45.

To perform the adaptation:

- 1 In the Simulink model window, enter `inf` as the simulation time.

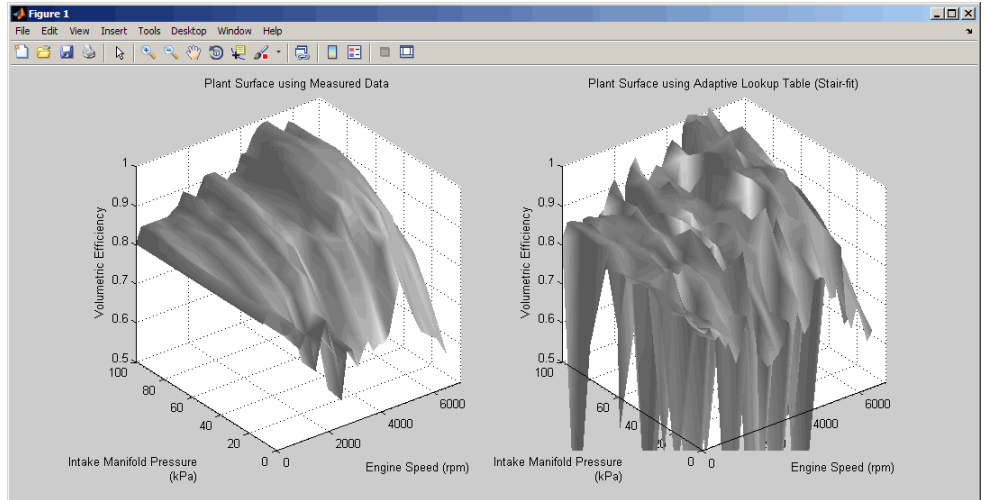


The simulation time of infinity specifies that the adaptation process continues as long as the input and output values of the engine change.

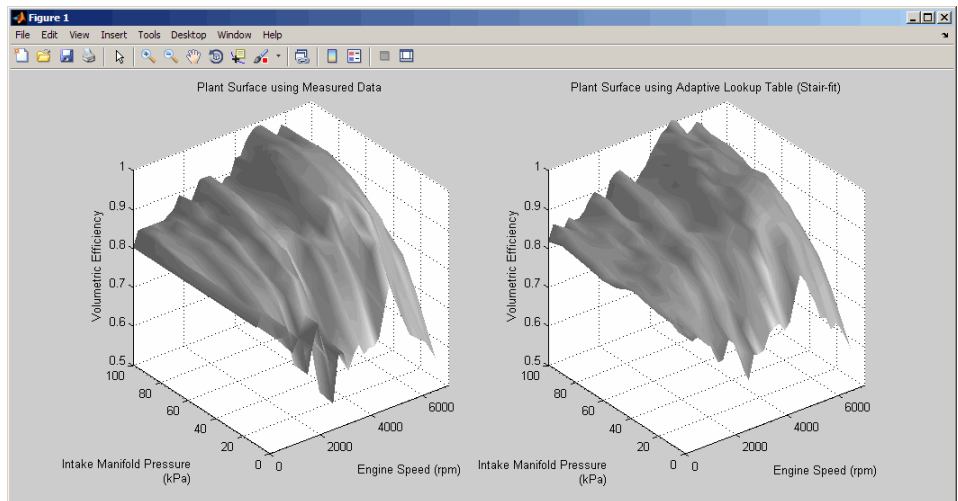
- 2 In the Simulink model window, select **Simulation > Start** to start the adaptation process.

A figure window opens that shows the volumetric efficiency of the engine as a function of the intake manifold pressure and engine speed:

- The left plot shows the measured volumetric efficiency as a function of intake manifold pressure and engine speed.
- The right plot shows the volumetric efficiency as it adapts with the time-varying intake manifold pressure and engine speed.



During simulation, the lookup table values displayed on the right plot adapt to the variations in the I/O data. The left and the right plots resemble each other after a few seconds, as shown in the next figure.




Tip During simulation, the `Cell Number` and `Adaptive Table Outputs` blocks in the Simulink model display the cell number, and the adapted lookup table value in the cell, respectively.

- 3 Pause the simulation by selecting **Simulation > Pause**.

This action also exports the adapted table values `Tout` to the MATLAB workspace.

Note After you pause the simulation, the adapted table values are stored in the `Adaptive Lookup Table` block.

- 4 Examine that the left and the right plots match. This resemblance indicates that the table values have adapted to the time-varying I/O data.
- 5 Lock a table cell so that only one cell adapts. You may find this feature useful if a portion of the data is highly erratic or otherwise difficult for the algorithm to handle.
 - a Select **Simulation > Start** to restart the simulation.

- b Double-click the `Lock` block. This action toggles the switch to .

You can view the number of the locked cell in the `Cell Number` block in the Simulink model.

- 6 After the table values adapt to the time-varying I/O data, you can continue to use the `Adaptive Lookup Table` block as a static lookup table:
 - a In the Simulink model window, double-click the `Enable` block. This action toggles the switch, and disables the adaptation.
 - b Select **Simulation > Start** to restart the simulation, if it is not already running.

During simulation, the `Adaptive Lookup Table` block works like a static lookup table, and continues to estimate the output values as the

input values change. You can see the current lookup table value in the Adaptive Table Outputs block in the Simulink model window.

Note After you disable the adaptation, the Adaptive Lookup Table block does not update the stored table values, and the figure that displays the table values does not update.

Using Adaptive Lookup Tables in Real-Time Environment

You can use experimental data from sensor measurements collected by running various tests on a system in real time. The measured data is then sent to the adaptive table block to generate a lookup table describing the relation between the system inputs and output.

You can also use the Adaptive Lookup Table block in a real-time environment, where some time-varying properties of a system need to be captured. To do so, generate C code using Real-Time Workshop® code generation software that can then be run in an xPC Target™ or dSPACE® software. Because you can start, stop, or reset the adaptation if you want, use logic to enable the adaptation of the table data only when it is desired. The cell number output N, and the Enable and Lock inputs facilitate this process. Use the Enable input to start and stop the adaptation and the Lock input to update only one of the table cells. The Lock input combined with some logic using the cell number output N provide the means for updating only the desired table cells during a simulation run.

Function Reference

Parameter Estimation (p. 6-2)

Parameter estimation using
measured data

Parameter Optimization (p. 6-3)

Optimize model parameters to meet
time-domain requirements

Parameter Estimation

`spetool`

Create Estimation Task in Control
and Estimation Tools Manager GUI

Parameter Optimization

Response Optimization Projects (p. 6-3)	Work with response optimization projects
Design Requirements (p. 6-3)	Manage design requirements
Parameters (p. 6-4)	Manage parameters to optimize
Model Dependencies (p. 6-4)	
Model Robustness (p. 6-4)	Specify uncertain parameter values
Optimization Options (p. 6-4)	View and modify optimization settings
Simulation Options (p. 6-4)	View and modify simulation settings

Response Optimization Projects

<code>getsro</code>	Extract response optimization project for given Simulink model
<code>ncdupdate</code>	Upgrade models with Nonlinear Control Design Blockset blocks
<code>newsro</code>	New response optimization project with default settings
<code>optimize</code>	Run response optimization project

Design Requirements

<code>findconstr</code>	Extract design requirements specified in Signal Constraint block
-------------------------	--

Parameters

<code>findpar</code>	Find specifications for given tuned parameter
<code>initpar</code>	Specify initial parameter values in response optimization project

Model Dependencies

<code>finddepend</code>	List of model path dependencies
-------------------------	---------------------------------

Model Robustness

<code>gridunc</code>	Multi-dimensional grid of uncertain parameter values
<code>randunc</code>	Random values of uncertain parameters
<code>setunc</code>	Specify parameter uncertainty in response optimization project

Optimization Options

<code>optimget</code>	Current optimizer settings
<code>optimset</code>	Modify optimization settings

Simulation Options

<code>simget</code>	Current simulation settings
<code>simset</code>	Modify simulation settings

Functions — Alphabetical List

findconstr

Purpose Extract design requirements specified in Signal Constraint block

Syntax `constraints=findconstr(proj,'blockname')`

Description `constraints=findconstr(proj,'blockname')` extracts design requirements specified in the Signal Constraint block `blockname` of the Simulink model associated with the response optimization project `proj`. `constraints` contains data that defines the design requirements. Modify the `UpperBoundX`, `UpperBoundY`, `LowerBoundX` and `LowerBoundY` properties of `constraints` to specify new design requirements, and the `ReferenceX` and `ReferenceY` properties to add a reference signal. The Simulink model associated with `proj` must be open.

Note The Signal Constraint block does not update to display the modified constraints and reference signal. However, the updated constraint bounds and reference signal specified in the constraint object are used when you optimize the parameters from the command line.

Example Retrieve the constraints specified in the Signal Constraint block of the model associated with the response optimization project:

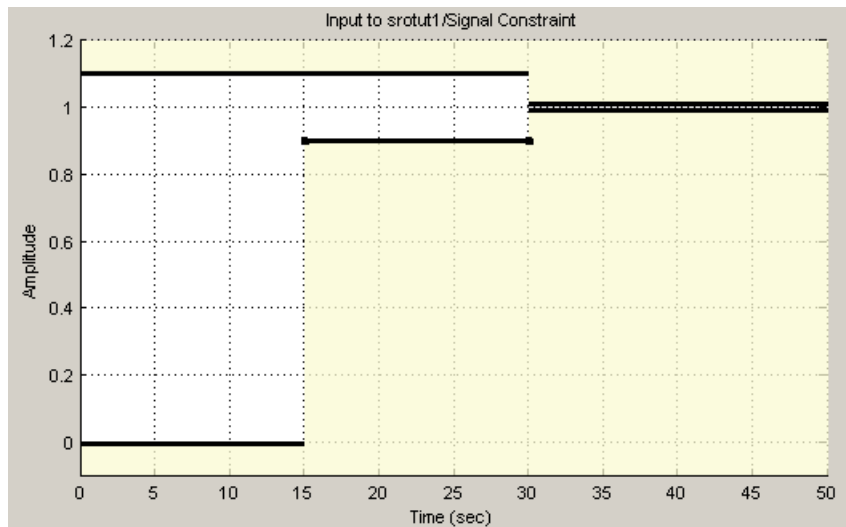
```
% Open the Simulink model
srotut1
% Create a response optimization project with default settings and
% parameters to optimize
proj=newsro('srotut1','Kint');
% Retrieve the constraints for this project
constraint=findconstr(proj,'srotut1/Signal Constraint');
```

Change the positioning of the constraint bounds by editing the upper- and lower-bound matrices:

```
constraint.UpperBoundY=[1.1 1.1;1.01 1.01];
constraint.UpperBoundX=[0 30;30 50];
constraint.LowerBoundY=[0 0;0.9 0.9;0.99 0.99];
```

```
constraint.LowerBoundX=[0 15;15 30;30 50];
```

If you add these constraints graphically in the Signal Constraint block, the constraints appear as shown in the following figure. To learn more about how to add constraints graphically, see “Specifying Design Requirements” on page 3-5.

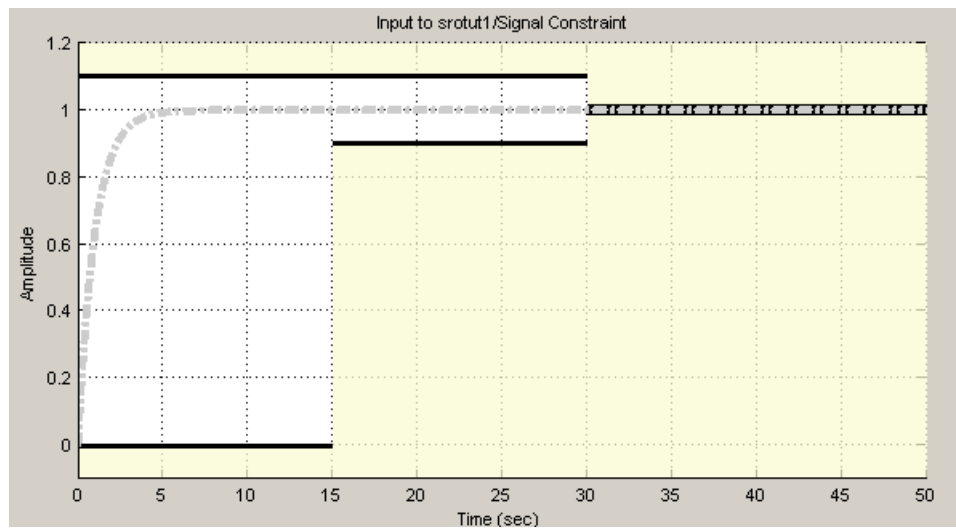


Include a reference signal using the following commands:

```
constraint.CostEnable='on';
constraint.ReferenceX=linspace(0,50,1000);
constraint.ReferenceY=1-exp(-linspace(0,50,1000));
```

If you include the reference signal graphically in the Signal Constraint block, the reference signal appears as shown in the following figure. To learn more about how to specify a reference signal, see “Tracking Reference Signals” on page 3-18.

findconstr



See Also

getsro, newsro, optimize

Purpose List of model path dependencies

Syntax `dirs=finddepend(proj)`

Description `dirs=finddepend(proj)` returns paths containing Simulink model dependencies required for parameter optimization using parallel computing. `proj` is a response optimization project for the model created using `newsro` or `getsro`. `dirs` is a cell array, where each element is a path string. `dirs` is empty when `finddepend` does not detect any model dependencies. Append paths to `dirs` when the list of paths is empty or incomplete.

`finddepend` does not return a complete list of model dependency paths when the dependencies are undetectable.

Examples List model path dependencies required for parallel computing:

```
% Copy Simulink boiler library to temporary folder.
pathToLib = boilerpressure_setup;
% Add folder to search path.
addpath(pathToLib);
% Open Simulink model.
boilerpressure_demo
% Extract response optimization project.
proj=getsro('boilerpressure_demo');
% Enable parallel computing.
optimset(proj,'UseParallel','always');
% Get model dependency paths.
dirs=finddepend(proj)
% Add paths to optimization project.
optimset(proj,'ParallelPathDependencies',dirs)
```

Make local paths accessible to remote workers:

```
% Copy Simulink boiler library to temporary folder.
pathToLib = boilerpressure_setup;
```

finddepend

```
% Add folder to search path.
addpath(pathToLib);
% Open Simulink model.
boilerpressure_demo
% Extract response optimization project.
proj=getsro('boilerpressure_demo');
% Enable parallel computing.
optimset(proj,'UseParallel','always');
% Get model dependency paths.
dirs=finddepend(proj)
% The resulting path is on a local drive, C:/.
% Replace C:/ with valid network path accessible to remote workers.
dirs = regexprep(dirs,'C:/','\\\\hostname\\C$\\')
% Add paths to optimization project.
optimset(proj,'ParallelPathDependencies',dirs)
```

Append path to model path dependency list:

```
% Copy Simulink boiler library to temporary folder.
pathToLib = boilerpressure_setup;
% Add folder to search path.
addpath(pathToLib);
% Open Simulink model.
boilerpressure_demo
% Extract response optimization project.
proj=getsro('boilerpressure_demo');
% Enable parallel computing.
optimset(proj,'UseParallel','always');
% Get model dependency paths.
dirs=finddepend(proj)
% Append an additional path.
dirs=vertcat(dirs,'\\hostname\C$\matlab\work')
% Add paths to optimization project.
optimset(proj,'ParallelPathDependencies',dirs)
```

Alternatives

Identify model path dependencies using the GUI:

- 1** In the Simulink model, double-click the Signal Constraint block to open the Block Parameters: Signal Constraint window.
- 2** In the Block Parameters window, select **Optimization > Parallel Options** to open the **Parallel Options** tab.
- 3** Select the **Use the matlabpool during optimization** option to identify the model dependencies automatically.

See Also

newsro | getsro | optimget | optimset

Tutorials

- Improving Optimization Performance Using Parallel Computing

How To

- “Speeding Up Response Optimization Using Parallel Computing” on page 3-60
- “Model Dependencies”

findpar

Purpose Find specifications for given tuned parameter

Syntax `p=findpar(proj, 'param')`

Description `p=findpar(proj, 'param')` returns a tuned parameters object for the parameter with the name `param` within the response optimization project, `proj`. The tuned parameters object defines specifications for each tuned parameter that the optimization method uses, such as initial guesses, lower bounds, etc.

The properties of each tuned parameter object are

Name	A string giving the parameter's name.
Value	The current value of the parameter. This changes during the optimization.
InitialGuess	The initial guess for the parameter value for the optimization.
Minimum	The minimum value this parameter can take. By default, it is set to <code>-Inf</code> .
Maximum	The maximum value this parameter can take. By default, it is set to <code>Inf</code> .
TypicalValue	A value that the tuned parameter is scaled by during the optimization.
ReferencedBy	The block, or blocks, in which the parameter appears.
Description	An optional string giving a description of the parameter.
Tuned	Set to 1 or 0 to indicate if this parameter is to be tuned or not.

Edit these properties to specify additional information about your parameters.

Example

Create a response optimization project for srotut1.

```
proj=newsro('srotut1','Kint');
```

Find the tuned parameters object for the parameter Kint.

```
p=findpar(proj,'Kint')
```

This command returns the following result:

```
      Name: 'Kint'  
      Value: 0  
InitialGuess: 0  
      Minimum: -Inf  
      Maximum: Inf  
TypicalValue: 0  
ReferencedBy: {0x1 cell}  
Description: ''  
      Tuned: 1
```

Tuned parameter.

Change the initial guess to 0.5, and the minimum value to 0 with the set function.

```
set(p,'InitialGuess',0.5,'Minimum',0)
```

See Also

getsro, newsro, optimize

Purpose Extract response optimization project for given Simulink model

Syntax `proj=getsro('modelName')`

Description `proj=getsro('modelName')` returns the response optimization project, `proj`, currently associated with the Simulink model with name, `modelName`. The model should be open and contain Simulink Design Optimization blocks. Use the project with the `optimize` function to optimize response signals in the model by tuning specified parameters.

Example Open the model `pidtune_demo` by typing the model name at the MATLAB prompt:

```
pidtune_demo
```

Extract the response optimization project from this model by typing the following command at the MATLAB prompt:

```
proj=getsro('pidtune_demo')
```

This command returns the following result:

```
                Name: 'pidtune_demo'  
Parameters: [3x1 ResponseOptimizer.Parameter]  
OptimOptions: [1x1 sroengine.OptimOptions]  
Tests: [1x1 ResponseOptimizer.SimTest]  
Model: 'pidtune_demo'
```

```
Response Optimization Project.
```

Use the `findpar` and `findconstr` functions to specify signal constraints and tuned parameters.

See Also `findconstr`, `findpar`, `newsro`, `optimize`

Purpose	Multi-dimensional grid of uncertain parameter values
Syntax	<code>uncpar=gridunc('ParameterName',Values,...)</code>
Description	<code>uncpar=gridunc('ParameterName',Values,...)</code> constructs a multi-dimensional grid that contains all value combinations of uncertain parameters <i>ParameterName</i> . <i>uncpar</i> contains the uncertain parameter values.
Inputs	<p>ParameterName_n Name of the uncertain parameter in a response optimization project.</p> <p>Values_n Values of corresponding uncertain parameter <i>ParameterName_n</i>:</p> <ul style="list-style-type: none"> • <i>{first_value,...,last_value}</i> for vector- or matrix-valued parameters. Dimensions of each cell element of <i>{first_value,...,last_value}</i> must match the dimensions of the parameter. • <i>{first_value,...,last_value}</i> or <i>[first_value,...,last_value]</i> for scalar-valued parameters.
Examples	<p>Create a grid of values for vector-valued uncertain parameters:</p> <pre>uset=gridunc('Kp',{[1;4],[1;4]},'Kd',... {[0.1;0.3;0.2],[0.1;0.3;0.2],[0.1;0.3;0.2]});</pre> <p>Create and enable a grid of values for uncertain parameters to test model robustness:</p> <pre>% Create a grid of values for the uncertain parameters P, I and D. uset=gridunc('P',[1,2,3,4],'I',[0.1,0.2,0.3],'D',[30,35,40]); % View values for the uncertain parameter P. uset.P % The default value of the Optimized property of uset is false.</pre>

```
% This implies that the uncertain parameter values are not
% enabled for testing model robustness.
% Set the value to true to enable uncertain parameter values.
uset.Optimized(1:end)=true;
```

Algorithm

For parameters with values specified as $\{first_value, \dots, last_value\}$ or $[first_value, \dots, last_value]$, gridunc creates a hypercube which consists of:

- 2^S vertices, where S is the number of parameters. $first_value$ and $last_value$ of each parameter form the vertices of the hypercube.
- Number of samples equal to the product of number of values specified for each parameter.

Alternatives

To create a grid of uncertain parameter values using the GUI:

- 1** In the Simulink model, double-click the Signal Constraint block to open the Block Parameters: Signal Constraint window.
- 2** In the Block Parameters window, select **Optimization > Uncertain Parameters** to open the Uncertain Parameters dialog box.
- 3** Select Grid as the **Sampling method**.
- 4** Specify the uncertain parameters and their values:
 - a** Click **Add** to open the Add Parameters dialog box.
 - b** Select the uncertain parameters and click **OK** to add them to the Uncertain Parameters dialog box.
 - c** Specify the values for the corresponding parameter in the **Sample Values** column.

See Also

setunc | randunc

How To

- “Sampling Methods for Computing Uncertain Parameter Values” on page 3-43

- “Optimizing Parameters for Model Robustness” on page 3-42

initpar

Purpose Specify initial parameter values in response optimization project

Syntax `initpar(proj)`

Description `initpar(proj)` updates the parameters' initial guess in *proj* with the current parameter values in the model or base workspace. *proj* is a response optimization project, created using `getsro` or `newsro`.

Before using `initpar`, open the Simulink model associated with *proj*.

Examples Initialize a response optimization project with updated parameter values:

1 Run an optimization to meet step response requirements.

```
% Open Simulink model.
sldo_model1_constrblk
% Create response optimization project to optimize
% parameters Kd, Ki and Kp.
proj=newsro('sldo_model1_constrblk',{'Kd','Ki','Kp'})
% Extract design requirements from Signal Constraint block.
constr=findconstr(proj,...
'sldo_model1_constrblk/Signal Constraint')
% Specify step response requirements.
constr.LowerBoundX=[0 10;10 15;15 50];
constr.LowerBoundY=[0 10;10 30;30 50];
constr.UpperBoundX=[0 30;30 50];
constr.UpperBoundY=[1.1 1.1;1.01 1.01];
% View initial value of Kd.
proj.parameter(1).InitialGuess
% Optimize parameters to meet step response requirements.
optimize(proj);
```

2 Specify a reference signal, and initialize *proj* with updated parameter values before running a new optimization.

```
% Specify reference signal.
```

```
constr.CostEnable='on';
constr.ReferenceX=linspace(0,50,200)';
constr.ReferenceY=1-exp(-0.3*linspace(0,50,200))';
% The values of Kd, Ki and Kp are not updated
% in proj. For example:
proj.parameter(1).InitialGuess
% Update proj with updated parameter values.
initpar(proj);
% View the updated initial value of Kd, which is now updated.
% The same is true for Ki and Kp.
proj.parameter(1).InitialGuess
% Rerun optimization to meet step requirements and
% track the reference signal simultaneously.
optimize(proj);
```

See Also

newsro | findpar | findconstr | optimize

Tutorials

- “Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the Command Line”

How To

- “Optimizing Parameters at the Command Line” on page 3-87

ncdupdate

Purpose	Upgrade models with Nonlinear Control Design Blockset blocks
Syntax	<code>ncdupdate('modelName')</code>
Description	<p><code>ncdupdate('modelName')</code> searches the Simulink model specified by the string 'modelName' for Nonlinear Control Design Blockset Outputport blocks and replaces them by the equivalent Signal Constraint block from Simulink Design Optimization library. The model must be open prior to calling <code>ncdupdate</code>.</p> <p>When your Nonlinear Control Design Blockset settings are stored in a <code>ncdStruct</code> variable that automatically load when you open the model, this variable changes in the workspace during the update so that it is compatible with Simulink Design Optimization software. You must resave this variable after the update.</p> <p>When your Nonlinear Control Design Blockset settings are stored in an <code>ncdStruct</code> variable that do not automatically load with the model, first load the <code>ncdStruct</code> variable into the workspace before calling <code>ncdupdate</code>. Then resave the variable after the update.</p> <p>To retain the upgraded blocks, save the model after running <code>ncdupdate</code>.</p>
See Also	<code>slupdate</code>

Purpose	New response optimization project with default settings
Syntax	<code>proj=newsro('modelName',params)</code>
Description	<code>proj=newsro('modelName',params)</code> creates a response optimization project <i>proj</i> for the Simulink model <i>modelName</i> . <i>params</i> is a cell array of strings that specifies model parameters for optimization. <i>proj</i> contains default design requirements, parameter settings, and optimization options in the Signal Constraint block. The model must contain at least one Signal Constraint block.
Examples	<p>Create a response optimization project with default settings and specify parameters for optimization:</p> <pre>% Create a new response optimization project and specify parameters % Kp, Ki and Kd for optimization. proj = newsro('pidtune_demo',{'Kp' 'Ki' 'Kd'})</pre>
Alternatives	<p>Create a response optimization project with default settings using the GUI:</p> <ol style="list-style-type: none">1 Drag and drop a Signal Constraint block from the Simulink Design Optimization library into the Simulink model.2 Connect the Signal Constraint block to the signal that must meet specific design requirements.3 Double-click the Signal Constraint block to create a response optimization project with default settings.
See Also	<code>findconstr</code> <code>findpar</code> <code>getsro</code> <code>optimize</code>
Tutorials	<ul style="list-style-type: none">• “Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the Command Line”
How To	<ul style="list-style-type: none">• “Constraining Model Signals” on page 3-3

- “Optimizing Parameters at the Command Line” on page 3-87

Purpose Current optimizer settings

Syntax `opt_settings=optimget(proj)`

Description `opt_settings=optimget(proj)` returns the current optimization settings object, `opt_settings`, for the response optimization project `proj`.

Use `optimset` to modify the optimization options.

For more information on the settings and their possible values, see the `optimset` reference page.

Example Create a new default response optimization project for the model `srotut1`.

```
proj=newsro('srotut1','Kint');
```

Get the optimization settings for this project.

```
opt_settings=optimget(proj)
```

This command returns the following list of optimization settings and their current values.

```

Method: 'fmincon'
Algorithm: 'active-set'
Display: 'iter'
GradientType: 'basic'
MaximallyFeasible: 0
MaxIter: 100
TolCon: 1.0000e-003
TolFun: 1.0000e-003
TolX: 1.0000e-003
Restarts: 0
UseParallel: 'never'
ParallelPathDependencies: {0x1 cell}
SearchMethod: []

```

optimget

See Also

optimset, simget, simset

Purpose	Run response optimization project
Syntax	<code>result=optimize(proj)</code>
Description	<p><code>result=optimize(proj)</code> optimizes the responses specified in the response optimization project, <code>proj</code>, with the constraints, parameters, and settings. The response optimization results are displayed after each iteration. The tuned parameters are changed in the workspace. Enter the parameter name at the MATLAB prompt to see its new value.</p> <p>A results object, <code>result</code>, is also returned. The properties of this object are</p> <ul style="list-style-type: none">• Cost: The final value of the cost function.• ExitFlag: 1 if the optimization terminated successfully, 0 if it did not.• Iteration: The number of iterations. <p>For more information on the results properties, see the reference pages for the Optimization Toolbox functions <code>fmincon</code> and <code>fminsearch</code> and the Global Optimization Toolbox function <code>patternsearch</code>.</p>
Example	<p>Open the <code>pitchrate_demo</code> model.</p> <pre>pitchrate_demo</pre> <p>Create a response optimization project based on the current settings in the model.</p> <pre>proj=getsro('pitchrate_demo');</pre> <p>Run the optimization with the following command.</p> <pre>results=optimize(proj)</pre>

The expected results are displayed as follows.

Iter	S-count	f(x)	max		Directional derivative	First-order optimality	Procedure
			constraint	Step-size			
0	1	0	1803				
1	14	0	160	0.0287	0	0.0152	
2	21	0	0.2607	0.0327	0	0.00598	Hessian modified
3	28	0	0.04203	0.071	0	0.0122	Hessian modified
4	35	0	0.001894	0.0164	0	0.00112	Hessian modified
5	42	0	7.631e-006	0.000804	0	5.01e-006	Hessian modified

Successful termination.

Found a feasible or optimal solution within the specified tolerances.

k1 =

0.8674

k2 =

-0.1513

k3 =

-0.5003

results =

Cost: 0

X: [4x1 double]

ExitFlag: 1

Iteration: 5

See Also

findconstr, findpar, getsro, newsro, optimget, optimset

Purpose Modify optimization settings

Syntax `optimset(proj, 'Property1', Value1, 'Property2', Value2, ...)`

Description `optimset(proj, 'Property1', Value1, 'Property2', Value2, ...)` modifies the optimization settings within the response optimization project, `proj`. The value of the optimization setting, `Property1`, is set to `Value1`, `Property2` is set to `Value2`, etc.

Property	Description	Possible Settings
Method	The optimization method used. The following methods are available: <ul style="list-style-type: none"> • <code>fmincon</code> — Optimization Toolbox function <code>fmincon</code> • <code>patternsearch</code> — Global Optimization Toolbox function <code>patternsearch</code> • <code>fminsearch</code> — Optimization Toolbox function <code>fminsearch</code> 	{'fmincon'} 'patternsearch' 'fminsearch'
Algorithm	The algorithm used by the optimization method.	For <code>fmincon</code> optimization method: <ul style="list-style-type: none"> • {'active-set'} • 'trust-region-reflective' • 'interior-point'

optimset

Property	Description	Possible Settings
Display	<p>The level of information that the optimization displays:</p> <ul style="list-style-type: none">• <code>off</code> — No output• <code>iter</code> — Output at each iteration• <code>final</code> — Final output only• <code>notify</code> — Output only if the function does not converge	<code>'off'</code> <code>{'iter'}</code> <code>'final'</code> <code>'notify'</code>
GradientType	<p>Method used to calculate gradients when using <code>'fmincon'</code> as the Method. Use one of the following finite difference methods for gradient calculation:</p> <ul style="list-style-type: none">• <code>basic</code> — Default method for computing the gradients• <code>refined</code> — Offers a more robust and less noisy gradient calculation method than <code>'basic'</code> <p>The <code>refined</code> method is sometimes more expensive, and does not work with certain models such as SimPowerSystems models.</p>	<code>{'basic'}</code> <code>'refined'</code>

Property	Description	Possible Settings
MaximallyFeasible	<p>Option to specify that the optimization continue after an initial, feasible solution has been found:</p> <ul style="list-style-type: none"> • 0 — Terminate the optimization as soon as an initial solution that satisfies the constraints is found. The resulting response signal may lie very close to the constraint segment. • 1 — Continue the optimization after an initial solution is found. The optimization can continue to search for a maximally feasible solution that is typically located further inside the constraint region. <hr/> <p>Note The software ignores this option when you track a reference signal.</p> <hr/>	{0} 1
MaxIter	Maximum number of iterations allowed	Positive integer value
To1Con	Termination tolerance on the constraints	Positive scalar value
To1Fun	Termination tolerance on the function value	Positive scalar value
To1X	Termination tolerance on the parameter values	Positive scalar value

optimset

Property	Description	Possible Settings
Restarts	<p>In some optimizations, the Hessian may become ill-conditioned, and the optimization does not converge. In these cases, it is sometimes useful to restart the optimization after it stops, using the endpoint of the previous optimization as the starting point for the next one. To automatically restart the optimization, use this option to indicate the number of times you want to restart.</p>	Nonnegative integer value
UseParallel	<p>Parallel computing option for the following optimization methods:</p> <ul style="list-style-type: none">• <code>fmincon</code>• <code>patternsearch</code> <hr/> <p>Note Parallel Computing Toolbox software must be installed to enable parallel computing for the optimization methods.</p> <hr/> <p>When set to <code>'always'</code>, the methods compute the following in parallel:</p> <ul style="list-style-type: none">• <code>fmincon</code> — Computes finite difference gradients	<code>'always' {'never'}</code>

Property	Description	Possible Settings
	<ul style="list-style-type: none"> patternsearch — Performs population evaluation Disable the option by setting to 'never'.	
ParallelPathDependencies	Option to store model path dependencies when using parallel computing	Cell array of strings
SearchMethod	Search options for use with the patternsearch method	See “Search Options” in the Global Optimization Toolbox documentation.

For more information on the possible settings and the values they can take, see the reference page for `optimset` in the MATLAB documentation.

Example

Create a default response optimization project for the model `srotut1`.

```
proj=newsro('srotut1','Kint');
```

Get the optimization settings for this project.

```
opt_settings=optimget(proj)
```

This command returns the following list of optimization settings and their current values.

```
Method: 'fmincon'
Algorithm: 'active-set'
Display: 'iter'
GradientType: 'basic'
MaximallyFeasible: 0
MaxIter: 100
TolCon: 1.0000e-003
TolFun: 1.0000e-003
```

optimset

```
TolX: 1.0000e-003
Restarts: 0
UseParallel: 'never'
ParallelPathDependencies: {0x1 cell}
SearchMethod: []
```

Use `optimset` to change the maximum number of iterations to 150.

```
optimset(proj, 'MaxIter', 150)
```

To view the changes to `opt_settings`, enter the variable name at the MATLAB prompt.

```
opt_settings
```

This command returns

```
Method: 'fmincon'
Algorithm: 'active-set'
Display: 'iter'
GradientType: 'basic'
MaximallyFeasible: 0
MaxIter: 150
TolCon: 1.0000e-003
TolFun: 1.0000e-003
TolX: 1.0000e-003
Restarts: 0
UseParallel: 'never'
ParallelPathDependencies: {0x1 cell}
SearchMethod: []
```

See Also

`optimget`, `simget`, `simset`

Purpose	Random values of uncertain parameters
Syntax	<code>uncpar=randunc(N,'ParameterName',Range,...)</code>
Description	<code>uncpar=randunc(N,'ParameterName',Range,...)</code> generates random values of uncertain parameters, specified as comma separated <code>'ParameterName'</code> and <code>Range</code> value pairs. <code>Range</code> specifies the lower and upper bounds for the uncertain parameter. Enter <code>Range</code> as a cell array <code>{[Min],[Max]}</code> for vector- and scalar-valued parameters or vector <code>[Min,Max]</code> for scalar-valued parameters. Dimensions of each cell element must match the corresponding parameter dimension. <code>N</code> is the number of samples inside the hypercube formed by <code>Min</code> and <code>Max</code> of each parameter. <code>uncpar</code> contains the uncertain parameter values.
Examples	<p>Generate random values for vector-valued uncertain parameters <code>Kp</code> and <code>Kd</code>:</p> <pre>uset=randunc(10,'Kp',{[1,4,3,0],[4,10,7.5,6]},'Kd',{[2,2],[8,7]})</pre> <hr/> <p>Generate random values for scalar-valued uncertain parameters <code>w0</code> and <code>zeta</code>:</p> <pre>uset=randunc(4,'w0',[0.45,0.55],'zeta',[0.45,0.55])</pre> <hr/> <p>Generate random values for uncertain parameters and test model robustness:</p> <pre>% Open the Simulink model. sldo_model1_desreq_optim % Extract response optimization project from the model. proj=getsro('sldo_model1_desreq_optim'); % Create random values for uncertain parameters w0 and zeta. uset=randunc(4,'w0',[0.45,0.55],'zeta',[0.45,0.55]) % View values for the uncertain parameter w0.</pre>

```
uset.w0
% The default value of the Optimized property of uset is false.
% This value implies that the uncertain parameter values are not
% enabled for testing model robustness.
% Set the value to true to enable all uncertain parameter
% values for testing model robustness.
uset.Optimized(1:end)=true;
% Specify parameter uncertainty in response optimization project.
setunc(proj,uset);
% Test model robustness.
optimize(proj);
```

Algorithm

For parameters p with range specified as $[Min, Max]$ or $\{[Min], [Max]\}$, `randunc` interprets the range of the uncertain parameters as:

$$\text{Min}(i,j) \leq p(i,j) \leq \text{Max}(i,j)$$

`randunc` generates a set of uncertain parameter values consists of the following:

- All vertices of the hypercube specified by Min and Max values of the parameters. The total number of vertices of the hypercube is 2^S , where S is the number of uncertain parameters.
- N random samples inside the hypercube.

Alternatives

To generate random values of uncertain parameters using the GUI:

- 1** In the Simulink model, double-click the Signal Constraint block to open the Block Parameters: Signal Constraint window.
- 2** In the Block Parameters window, select **Optimization > Uncertain Parameters** to open the Uncertain Parameters dialog box.
- 3** Select Random (Monte Carlo) as the **Sampling method**.
- 4** Specify the number of samples in the **Number of samples** field.

- 5 Specify the uncertain parameters and their range:
 - a Click **Add** to open the Add Parameters dialog box.
 - b Select the uncertain parameters and click **OK** to add them to the Uncertain Parameters dialog box.
 - c Specify the range for the corresponding parameter in the **Min** and **Max** columns.

See Also

setunc | gridunc

How To

- “Sampling Methods for Computing Uncertain Parameter Values” on page 3-43
- “Optimizing Parameters for Model Robustness” on page 3-42

setunc

Purpose Specify parameter uncertainty in response optimization project

Syntax `setunc(proj,unc_settings)`

Description `setunc(proj,unc_settings)` sets the parameter uncertainty specifications for the response optimization project, `proj`. Use the function `gridunc` or `randunc` to specify the uncertainty settings, `unc_settings`.

Example Create a response optimization project.

```
proj=newsro('srotut1','Kint');
```

Specify uncertain parameter settings using `gridunc`.

```
uset=gridunc('zeta',[0.9,1,1.1],'w0',[0.95,1,1.05]);
```

Set the uncertain parameters in the project.

```
setunc(proj,uset)
```

See Also `gridunc` | `randunc`

Purpose Current simulation settings

Syntax `simoptions=simget('proj')`

Description `simoptions=simget('proj')` returns a object containing the current simulation settings, `simoptions`, used by the response optimization project, `proj`. To modify the project's simulation settings, use the `simset` function.

The project's simulation settings are a subset of the parameters that you can set for Simulink models and blocks. For a detailed list of these settings and the possible values they can take, see “About Model Parameters” in the Simulink documentation. The default values of the simulation settings are the same as those used by the Simulink model the project is associated with. Changes that are made to the project's simulation settings are only used during simulations that are run as part of the optimization, and they do not affect the simulation settings for the model.

Example Create a response optimization project for the `srotut1` model:

```
proj=newsro('srotut1','Kint');
```

Get the simulation settings for this project:

```
simoptions = simget(proj)
```

This returns

```
simoptions =  
    AbsTol: 1.0000e-006  
    FixedStep: 'auto'  
    InitialStep: 'auto'  
    MaxStep: 'auto'  
    MinStep: 'auto'  
    RelTol: 1.0000e-003  
    Solver: 'auto'  
    ZeroCross: 'on'
```

simget

```
StartTime: '0.0'  
StopTime: '50'
```

See Also `optimget` | `optimset` | `simset`

How To • “Specifying the Simulation Options” on page 3-32

Purpose	Modify simulation settings
Syntax	<code>simset(proj, 'setting1', value1, 'setting2', value2, ...)</code>
Description	<code>simset(proj, 'setting1', value1, 'setting2', value2, ...)</code> modifies the simulation settings within the response optimization project, <code>proj</code> . The value of the simulation setting, <code>setting1</code> , is set to <code>value1</code> , <code>setting2</code> is set to <code>value2</code> , etc.

The project's simulation settings are a subset of the parameters that you can set for Simulink models and blocks. For a detailed list of these settings and the possible values they can take, see “About Model Parameters” in the Simulink documentation. The default values of the simulation options for the project are the same as those used by the Simulink model the project is associated with. Changes that are made to the project's simulation settings are only used during simulations that are run as part of the optimization, and they do not affect the simulation settings for the model.

Example Create a response optimization project for the `srotut1` model:

```
proj=newsro('srotut1', 'Kint');
```

Get the simulation settings for this project:

```
simoptions = simget(proj)
```

This returns

```
simoptions =  
    AbsTol: 1.0000e-006  
    FixedStep: 'auto'  
    InitialStep: 'auto'  
    MaxStep: 'auto'  
    MinStep: 'auto'  
    RelTol: 1.0000e-003  
    Solver: 'auto'  
    ZeroCross: 'on'
```

simset

```
StartTime: '0.0'  
StopTime: '50'
```

Use `simset` to change the solver type to `ode23` and the absolute tolerance to `1e-7`.

```
simset(proj, 'Solver', 'ode23', 'AbsTol', 1e-7)
```

Check the new values:

```
sim_settings=simget(proj);  
sim_settings.Solver
```

This shows that the solver is now set to `ode23`.

```
ans =  
ode23
```

Check the absolute tolerance:

```
sim_settings.AbsTol
```

This value is now set to `1e-7`.

```
ans =  
1.0000e-007
```

See Also

`optimget` | `optimset` | `simget`

How To

- “Specifying the Simulation Options” on page 3-32

Purpose Create Estimation Task in Control and Estimation Tools Manager GUI

Syntax `spetool('modelname')`

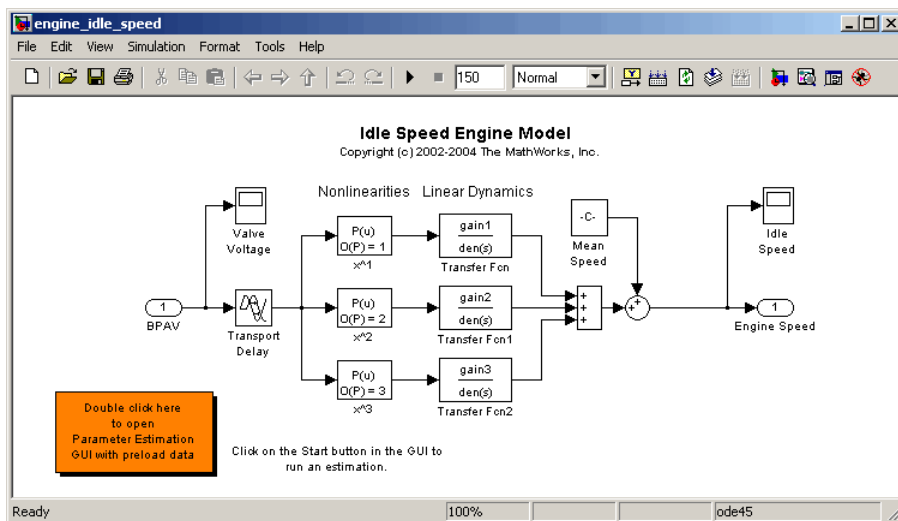
Description `spetool('modelname')` opens the Simulink model with the name `modelname` and creates an estimation task in the Control and Estimation Tools Manager GUI.

Example Create an estimation task by typing the following command at the MATLAB prompt:

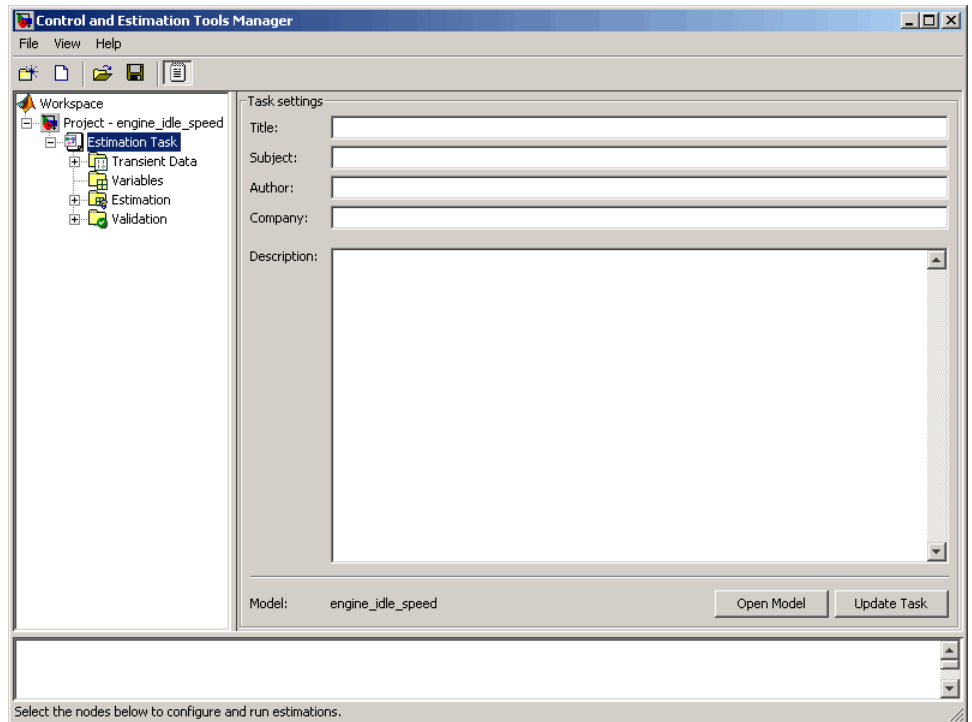
```
spetool('engine_idle_speed')
```

This command opens the following:

- Simulink model



- Control and Estimation Tools Manager containing a project with an estimation task



See Also

“Importing Data into the GUI” on page 1-5, “Configuring Parameter Estimation in the GUI” on page 2-3

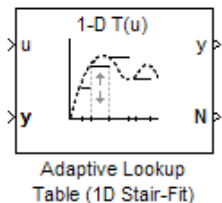
Block Reference

Adaptive Lookup Table (1D Stair-Fit)

Purpose Perform one-dimensional adaptive table lookup

Library Simulink Design Optimization

Description



The Adaptive Lookup Table (1D Stair-Fit) block creates a one-dimensional adaptive lookup table by dynamically updating the underlying lookup table. The block uses the outputs, y , of your system to do the adaptations.

Each indexing parameter u may take a value within a set of adapting data points, which are called *breakpoints*. Two breakpoints in each dimension define a *cell*. The set of all breakpoints in one of the dimensions defines a *grid*. In the one-dimensional case, each cell has two breakpoints, and the cell is a line segment.

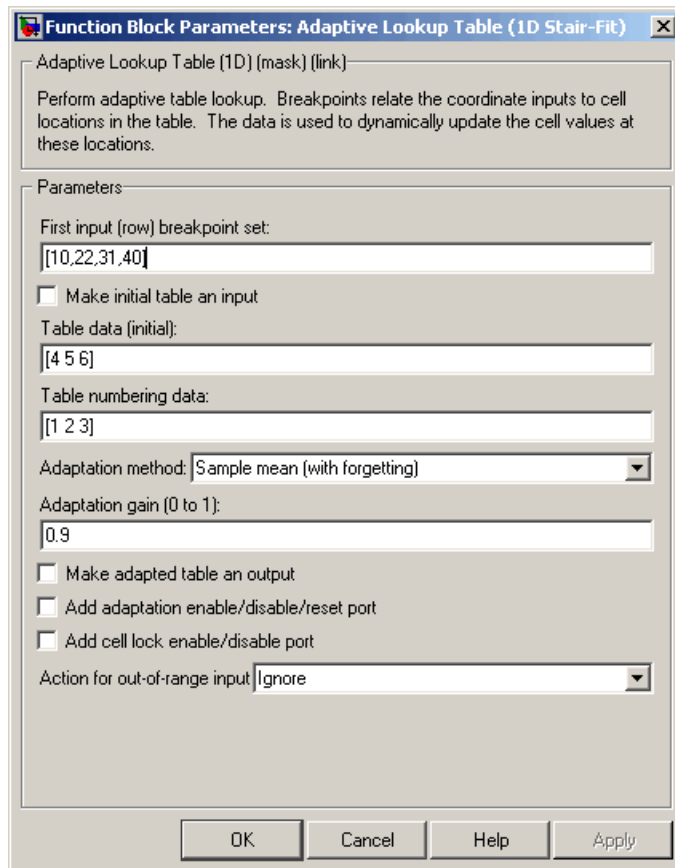
You can use the Adaptive Lookup Table (1D Stair Fit) block to model time-varying systems with one input.

Data Type Support

Doubles only

Adaptive Lookup Table (1D Stair-Fit)

Dialog Box



First input (row) breakpoint set

The vector of values containing possible block input values. The input vector must be monotonically increasing.

Make initial table an input

Selecting this check box forces the Adaptive Lookup Table (1D Stair-Fit) block to ignore the **Table data (initial)** parameter, and creates a new input port **Tin**. Use this port to input the table data.

Adaptive Lookup Table (1D Stair-Fit)

Table data (initial)

The initial table output values. This vector must be of size $N-1$, where N is the number of breakpoints.

Table numbering data

Number values assigned to cells. This vector must be the same size as the table data vector, and each value must be unique.

Adaptation method

Choose `Sample mean` or `Sample mean (with forgetting)`. `Sample mean` averages all the values received within a cell. `Sample mean with forgetting` gives more weight to the new data. How much weight is determined by the **Adaptation gain** parameter. For more information, see ..

Adaptation gain (0 to 1)

A number between 0 and 1 that regulates the weight given to new data during the adaptation. A 0 means short memory (last data becomes the table value), and 1 means long memory (average all data received in a cell).

Make adapted table an output

Selecting this check box creates an additional output port `Tout` for the adapted table.

Add adaptation enable/disable/reset port

Selecting this check box creates an additional input port `Enable` that enables, disables, or resets the adaptive lookup table. A signal value of 0 applied to the port disables the adaptation, and signal value of 1 enables the adaptation. Setting the signal value to 2 resets the table values to the initial table data.

Add cell lock enable/disable port

Selecting this check box creates an additional input port `Lock` that provides the means for updating only specified cells during a simulation run. A signal value of 0 unlocks the specified cells and signal value of 1 locks the specified cells.

Action for out-of-range input

Ignore or Adapt by extrapolating beyond the extreme breakpoints.

Adaptive Lookup Table (1D Stair-Fit)

See Also

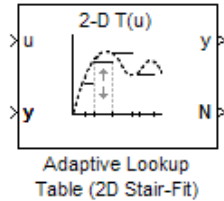
Adaptive Lookup Table (2D Stair-Fit), Adaptive Lookup Table (nD Stair-Fit), “Capturing Time-Varying System Behavior Using Adaptive Lookup Tables” on page 5-37

Adaptive Lookup Table (2D Stair-Fit)

Purpose Perform two-dimensional adaptive table lookup

Library Simulink Design Optimization

Description



The Adaptive Lookup Table (2D Stair-Fit) block creates a two-dimensional adaptive lookup table by dynamically updating the underlying lookup table. The block uses the outputs, y , of your system to do the adaptations.

Each indexing parameter u may take a value within a set of adapting data points, which are called *breakpoints*. Two breakpoints in each dimension define a *cell*. The set of all breakpoints in one of the dimensions defines a *grid*. In the two-dimensional case, each cell has four breakpoints and is a flat surface.

You can use the Adaptive Lookup Table (2D Stair-Fit) block to model time-varying systems with two inputs.

Data Type Support

Doubles only

Adaptive Lookup Table (2D Stair-Fit)

Dialog Box

Function Block Parameters: Adaptive Lookup Table (2D Stair-Fit)

Adaptive Lookup Table (2D) (mask) (link)

Perform adaptive table lookup. Breakpoints relate the coordinate inputs to cell locations in the table. The data is used to dynamically update the cell values at these locations.

Parameters

First input (row) breakpoint set:
[10,22,31,40]

Second input (column) breakpoint set:
[10,22,31,40]

Make initial table an input

Table data (initial):
[4 5 6;16 19 20;10 18 23]

Table numbering data:
[1 2 3; 4 5 6; 7 8 9]

Adaptation method: Sample mean (with forgetting)

Adaptation gain (0 to 1):
0.9

Make adapted table an output

Add adaptation enable/disable/reset port

Add cell lock enable/disable port

Action for out-of-range input: Ignore

OK Cancel Help Apply

First input (row) breakpoint set

The vector of values containing possible block input values for the first input variable. The first input vector must be monotonically increasing.

Adaptive Lookup Table (2D Stair-Fit)

Second input (column) breakpoint set

The vector of values containing possible block input values for the second input variable. The second input vector must be monotonically increasing.

Make initial table an input

Selecting this check box forces the Adaptive Lookup Table (2D Stair-Fit) block to ignore the **Table data (initial)** parameter, and creates a new input port T_{in} . Use this port to input the table data.

Table data (initial)

The initial table output values. This 2-by-2 matrix must be of size $(n-1)$ -by- $(m-1)$, where n is the number of first input breakpoints and m is the number of second input breakpoints.

Table numbering data

Number values assigned to cells. This matrix must be the same size as the table data matrix, and each value must be unique.

Adaptation method

Choose **Sample mean** or **Sample mean with forgetting**. **Sample mean** averages all the values received within a cell. **Sample mean with forgetting** gives more weight to the new data. How much weight is determined by the **Adaptation gain** parameter. For more information, see “Selecting an Adaptation Method” on page 5-42.

Adaptation gain (0 to 1)

A number from 0 to 1 that regulates the weight given to new data during the adaptation. A 0 means short memory (last data becomes the table value), and 1 means long memory (average all data received in a cell).

Make adapted table an output

Selecting this check box creates an additional output port T_{out} for the adapted table.

Add adaptation enable/disable/reset port

Selecting this check box creates an additional input port **Enable** that enables, disables, or resets the adaptive lookup table. A

Adaptive Lookup Table (2D Stair-Fit)

signal value of 0 applied to the port disables the adaptation, and signal value of 1 enables the adaptation. Setting the signal value to 2 resets the table values to the initial table data.

Add cell lock enable/disable port

Selecting this check box creates an additional input port **Lock** that provides the means for updating only specified cells during a simulation run. A signal value of 0 unlocks the specified cells and signal value of 1 locks the specified cells.

Action for out-of-range input

Ignore or Adapt by extrapolating beyond the extreme breakpoints.

See Also

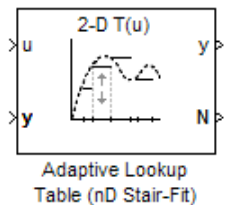
Adaptive Lookup Table (1D Stair-Fit), Adaptive Lookup Table (nD Stair-Fit), “Capturing Time-Varying System Behavior Using Adaptive Lookup Tables” on page 5-37

Adaptive Lookup Table (nD Stair-Fit)

Purpose Create adaptive lookup table of arbitrary dimension

Library Simulink Design Optimization

Description The Adaptive Lookup Table (nD Stair-Fit) block creates an adaptive lookup table of arbitrary dimension by dynamically updating the underlying lookup table. The block uses the outputs of your system to do the adaptations.



Each indexing parameter may take a value within a set of adapting data points, which are called *breakpoints*. Breakpoints in each dimension define a *cell*. The set of all breakpoints in one of the dimensions defines a *grid*. In the n-dimensional case, each cell has two n breakpoints and is an (n-1) hypersurface.

You can use the Adaptive Lookup Table (nD Stair-Fit) block to model time-varying systems with 2 or more inputs.

Data Type Support Doubles only

Adaptive Lookup Table (nD Stair-Fit)

Dialog Box

Function Block Parameters: Adaptive Lookup Table (nD Stair-Fit) [X]

Adaptive Lookup Table (nD) (mask) (link)

Perform adaptive table lookup. Breakpoints relate the coordinate inputs to cell locations in the table. The data is used to dynamically update the cell values at these locations.

Parameters:

Number of table dimensions:
2

Table breakpoints (cell array):
{[10,22,31,40], [10,22,31,40]}

Make initial table an input

Table data (initial):
[4 5 6;16 19 20;10 18 23]

Table numbering data:
[1 2 3; 4 5 6; 7 8 9]

Adaptation method: Sample mean (with forgetting)

Adaptation gain (0 to 1):
0.9

Make adapted table an output

Add adaptation enable/disable/reset port

Add cell lock enable/disable port

Action for out-of-range input: ignore

OK Cancel Help Apply

Number of table dimensions

The number of dimensions for the adaptive lookup table.

Table breakpoints (cell array)

A set of one-dimensional vectors that contains possible block input values for the input variables. Each input row must be monotonically increasing, but the rows do not have to be the same

Adaptive Lookup Table (nD Stair-Fit)

length. For example, if the **Number of table dimensions** is 3, you can set the table breakpoints as follows:

```
{[1 2 3], [5 7], [1 3 5 7]}
```

Make initial table an input

Selecting this check box forces the Adaptive Lookup Table (nD Stair-Fit) block to ignore the **Table data (initial)** parameter, and creates a new input port T_{in} . Use this port to input the table data.

Table data (initial)

The initial table output values. This (n-D) array must be of size (n-1)-by-(n-1) ... -by- (n-1), (D times), where D is the number of dimensions and n is the number of input breakpoints.

Table numbering data

Number values assigned to cells. This vector must be the same size as the table data array, and each value must be unique.

Adaptation method

Choose **Sample mean** or **Sample mean with forgetting**. **Sample mean** averages all the values received within a cell. **Sample mean with forgetting** gives more weight to the new data. How much weight is determined by the **Adaptation gain** parameter. For more information, see “Selecting an Adaptation Method” on page 5-42.

Adaptation gain (0 to 1)

A number from 0 to 1 that regulates the weight given to new data during the adaptation. A 0 means short memory (last data becomes the table value), and 1 means long memory (average all data received in a cell).

Make adapted table an output

Selecting this check box creates an additional output port T_{out} for the adapted table.

Adaptive Lookup Table (nD Stair-Fit)

Note The Adaptive Lookup Table (n-D Stair Fit) block cannot output a table of 3 or more dimensions.

Add adaptation enable/disable/reset port

Selecting this check box creates an additional input port **Enable** that enables, disables, or resets the adaptive lookup table. A signal value of 0 applied to the port disables the adaptation, and signal value of 1 enables the adaptation. Setting the signal value to 2 resets the table values to the initial table data.

Add cell lock enable/disable port

Selecting this check box creates an additional input port **Lock** that provides the means for updating only specified cells during a simulation run. A signal value of 0 unlocks the specified cells and signal value of 1 locks the specified cells.

Action for out-of-range input

Ignore or Adapt by extrapolating beyond the extreme breakpoints.

See Also

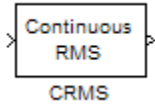
Adaptive Lookup Table (1D Stair-Fit), Adaptive Lookup Table (2D Stair-Fit), “Capturing Time-Varying System Behavior Using Adaptive Lookup Tables” on page 5-37

Purpose

Compute continuous-time, cumulative root mean square (CRMS) of signal

Library

Simulink Design Optimization

Description

Attach the CRMS block to a signal to compute its continuous-time, cumulative root mean square value. Use in conjunction with the Signal Constraint block to optimize the signal energy.

The continuous-time, cumulative root mean square value of a signal $u(t)$ is defined as

$$R.M.S = \sqrt{\frac{1}{T} \int_0^T \|u(t)\|^2 dt}$$

The R.M.S value gives a measure of the average energy in the signal.

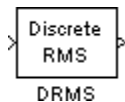
See Also

DRMS, Signal Constraint

Purpose Compute discrete-time, cumulative root mean square (DRMS) of signal

Library Simulink Design Optimization

Description



Attach the DRMS block to a signal to compute its discrete-time, cumulative root mean square value. Use in conjunction with the Signal Constraint block to optimize the signal energy.

The discrete-time, cumulative root mean square value of a signal $u(t_i)$ is defined as

$$R.M.S = \sqrt{\frac{1}{N} \sum_{i=1}^N \|u(t_i)\|^2}$$

The R.M.S value gives a measure of the average energy in the signal.

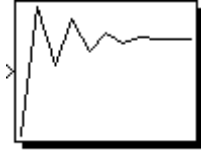
See Also CRMS, Signal Constraint

Signal Constraint

Purpose Specify desired signal response

Library Simulink Design Optimization

Description



Signal Constraint

Attach a Signal Constraint block to the signal in a Simulink model to optimize the model response to known inputs. Simulink Design Optimization software tunes parameters in the model to meet specified constraints. The constraints include bounds on signal amplitudes and matching of reference signals. The constraints are applicable to vector- and matrix-valued ports, in which case the signal bounds and reference signals apply to all entries of the signal/matrix.

For more information on how to use this block, see “Optimizing Parameters Using the GUI” on page 3-3.

See Also CRMS, DRMS

Examples

Use this list to find examples in the documentation.

Parameter Estimation

“Example — Estimating Initial States of a Mass-Spring-Damper System”
on page 2-67

“Example — Estimating Parameters and Initial States at the Command
Line” on page 2-84

Parameter Optimization

“Example — Optimizing Parameters for Model Robustness Using the GUI”
on page 3-49

Optimization-Based Linear Control Design

“Example — Frequency-Domain Optimization for LTI System” on page 4-9

Lookup Tables

“Example — Estimating Lookup Table Values from Data” on page 5-6

“Example — Estimating Constrained Values of a Lookup Table” on page
5-20

“Example — Modeling an Engine Using n-D Adaptive Lookup Table” on
page 5-44

Symbols and Numerics

1D adaptive lookup table 8-2

A

acceleration 2-50
adaptation gain 5-43
adaptive lookup tables 5-2
adding data sets 1-3
algorithms
 response optimization 3-28

C

command-line estimation 2-83
common questions 3-72
constraint bounds 3-19
 positioning exactly 3-8
 splitting 3-14
constraint segments
 moving 3-6
constraints
 positioning bounds 3-19
 scaling 3-14
 weightings 3-9
CRMS block 8-14
current responses
 plotting 3-36

D

data
 detrending 1-18
 filtering 1-18
 preprocessing 1-14
Data Import dialog box 1-9
data sets
 adding 1-3
design and analysis plots
 creating 4-11

design requirements
 example 4-17
desired responses
 constraint bounds 3-19
 reference signals 3-18
 step responses 3-15
detrending data 1-18
different time lengths 2-30
display options for estimation 2-35
DRMS block 8-15

E

Edit Design Requirement dialog box 3-8
estimation
 display options 2-35
 example of command-line estimation 2-84
 from the command line 2-83
 running 2-36
 selecting parameters 2-6
 selecting states 2-65
 setting up a project 1-3
Estimation Task 7-37

F

filtering data 1-18
findconstr function 7-2
finddepend function 7-5
findpar function 7-8

G

getsro function 7-10
gridlines
 response plots 3-8
gridunc function 7-11

H

handling outliers 1-18

I

- importing
 - initial conditions 2-17
 - transient data 1-5
- initial conditions
 - example of estimating 2-67
 - importing 2-17
- initial guesses 2-11
- initial responses
 - plotting 3-36
- initpar function 7-14
- intermediate responses
 - plotting 3-37

L

- loading
 - response optimization projects 3-86
- lookup tables
 - adaptive 5-2

M

- multiple projects and tasks 2-79

N

- ncdupdate function 7-16
- newsro function 7-17

O

- optimget function 7-19
- optimization
 - setting options for 2-23
- optimize function 7-21
- optimset function 7-23

P

- parallel computing

- speed up 3-60
- parameter estimation 2-50
- parameter estimation accelerator mode 2-50
- parameters
 - selecting for estimation 2-6
 - specification of 2-9
 - tuned 3-19
 - uncertain 3-42
- positioning constraints
 - snapping to horizontal 3-8
 - snapping to vertical 3-8
- preprocessing data 1-14
- projects
 - definition of 2-78
 - saving 2-81

R

- randunc function 7-29
- reference signals
 - plotting 3-36
 - specifying 3-18
 - tracking 3-18
- response optimization
 - accelerating 3-58
 - choosing signals 3-3
 - creating projects 3-3
 - example with control design in SISO Design Task 4-9
 - gradient type 3-29
 - maximally feasible solutions 3-29
 - running 3-39
 - simulation options 3-32
 - simulation solvers 3-33
 - starting 3-39
 - task within Control and Estimation Tools Manager 4-14
 - termination criteria 3-29
 - tolerances 3-29
- response optimization accelerator mode 3-58

- response optimization projects
 - properties 3-3
 - reloading 3-86
 - saving 3-82
- response optimization results
 - Optimization Progress dialog box 3-41
 - plots 3-40
 - tuning 3-27
- response plots
 - editing properties 3-37
- running an estimation 2-36

S

- Sample mean 5-43
- Sample mean with forgetting 5-43
- saving
 - response optimization projects 3-82
- saving projects 2-81
- selecting views 2-19
- setting options
 - for optimization 2-23
 - for simulation 2-23
 - upper/lower bounds 2-11
- setunc function 7-32
- Signal Constraint block 8-16
- signal responses
 - plotting 3-36
- simget function 7-33
- simset function 7-35
- simulation
 - setting options for 2-23
- simulation options
 - response optimization 3-32
- simulation solvers
 - response optimization 3-33
- simulation with 2-29

- specifying parameters 2-9
- states
 - selecting for estimation 2-65
- step responses
 - specifications 3-15

T

- transient data
 - importing 1-5
- troubleshooting
 - response optimization 3-72
- tuned compensator elements
 - example 4-16
- tuned parameters
 - adding 3-21
 - results 3-40
 - specifications 3-22
 - specifying 3-19
 - undoing 3-40

U

- uncertain parameters
 - adding 3-46
 - grid 3-43
 - random 3-43
 - specifications 3-43
 - specifying 3-42
 - using in response optimization projects 3-42
- upper/lower bounds
 - setting 2-11

V

- views
 - selecting 2-19