

# Simulink<sup>®</sup> Design Optimization<sup>™</sup>

## User's Guide

**R2012a**

**MATLAB<sup>®</sup>  
& SIMULINK<sup>®</sup>**

## How to Contact MathWorks



[www.mathworks.com](http://www.mathworks.com) Web  
[comp.soft-sys.matlab](mailto:comp.soft-sys.matlab) Newsgroup  
[www.mathworks.com/contact\\_TS.html](http://www.mathworks.com/contact_TS.html) Technical Support



[suggest@mathworks.com](mailto:suggest@mathworks.com) Product enhancement suggestions  
[bugs@mathworks.com](mailto:bugs@mathworks.com) Bug reports  
[doc@mathworks.com](mailto:doc@mathworks.com) Documentation error reports  
[service@mathworks.com](mailto:service@mathworks.com) Order status, license renewals, passcodes  
[info@mathworks.com](mailto:info@mathworks.com) Sales, pricing, and general information



508-647-7000 (Phone)



508-647-7001 (Fax)



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

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

*Simulink® Design Optimization™ User's Guide*

© COPYRIGHT 1993–2012 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](http://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

MathWorks products are protected by one or more U.S. patents. Please see [www.mathworks.com/patents](http://www.mathworks.com/patents) for more information.

**Revision History**

March 2009 Online only  
September 2009 Online only  
March 2010 Online only  
September 2010 Online only  
April 2011 Online only  
September 2011 Online only  
March 2012 Online only

New for Version 1 (Release 2009a)  
Revised for Version 1.1 (Release 2009b)  
Revised for Version 1.1.1 (Release 2010a)  
Revised for Version 1.2 (Release 2010b)  
Revised for Version 1.2.1 (Release 2011a)  
Revised for Version 2.0 (Release 2011b)  
Revised for Version 2.1 (Release 2012a)



## Data Analysis and Processing

**1**

<b>Model Requirements for Importing Data</b> .....	<b>1-2</b>
<b>Import Data (GUI)</b> .....	<b>1-4</b>
Creating an Estimation Project .....	<b>1-4</b>
Importing Time-Domain Data into the GUI .....	<b>1-6</b>
Importing Time-Series Data into the GUI .....	<b>1-11</b>
Importing Complex Data into the GUI .....	<b>1-11</b>
<b>Plot and Analyze Data (GUI)</b> .....	<b>1-12</b>
Why Plot the Data Before Parameter Estimation .....	<b>1-12</b>
How To Plot Data in the GUI .....	<b>1-12</b>
<b>Preprocess Data (GUI)</b> .....	<b>1-15</b>
Ways to Preprocess Data Using the Data Preprocessing Tool .....	<b>1-15</b>
Opening the Data Preprocessing Tool .....	<b>1-15</b>
Handling Missing Data .....	<b>1-17</b>
Handling Outliers .....	<b>1-19</b>
Detrending Data .....	<b>1-19</b>
Filtering Data .....	<b>1-19</b>
Selecting Data .....	<b>1-21</b>
<b>Add Preprocessed Data Sets to an Estimation Project (GUI)</b> .....	<b>1-30</b>
Overwriting an Existing Data Set .....	<b>1-30</b>
Creating a New Data Set .....	<b>1-31</b>
<b>Export Prepared Data to the MATLAB Workspace</b> ....	<b>1-33</b>

<b>Parameter Estimation Workflow</b> .....	2-2
<b>Specify Estimation Data (GUI)</b> .....	2-3
Creating an Estimation Task .....	2-3
How to Specify Data .....	2-4
<b>Specify Parameters to Estimate (GUI)</b> .....	2-7
Choosing Which Parameters to Estimate First .....	2-7
How to Specify Parameters for Estimation .....	2-7
Specifying Initial Guesses and Upper/Lower Bounds .....	2-12
Specifying Parameter Dependency .....	2-14
Specify Independent Parameters to Estimate (GUI) .....	2-15
<b>Specify Known Initial States (GUI)</b> .....	2-18
When to Specify Initial States Versus Estimate Initial States .....	2-18
How to Specify Initial States in the GUI .....	2-18
<b>Progress Plots</b> .....	2-21
Types of Plots .....	2-21
Basic Steps for Creating Plots .....	2-21
<b>Estimation Options</b> .....	2-25
Accessing Estimation Options .....	2-25
Supported Estimation Methods .....	2-27
Selecting Optimization Termination Options .....	2-29
Selecting Additional Optimization Options .....	2-29
Specifying Goodness of Fit Criteria (Cost Function) .....	2-30
How to Specify Estimation Options in the GUI .....	2-30
<b>Simulation Options</b> .....	2-32
Accessing Simulation Options .....	2-32
Selecting Simulation Time .....	2-33
Selecting Solvers .....	2-34
<b>Progress Display Options</b> .....	2-38

<b>Estimate Parameters (GUI)</b> .....	<b>2-39</b>
<b>Validate Parameters (GUI)</b> .....	<b>2-43</b>
Model Validation .....	<b>2-43</b>
Load and Import Validation Data (GUI) .....	<b>2-44</b>
Compare Measured and Simulated Responses (GUI) .....	<b>2-46</b>
Compare Residuals (GUI) .....	<b>2-50</b>
<b>Accelerating Model Simulations During Estimation</b> ..	<b>2-53</b>
About Accelerating Model Simulations During Estimation .....	<b>2-53</b>
Limitations .....	<b>2-53</b>
Setting the Accelerator Mode for Parameter Estimation ..	<b>2-53</b>
<b>Speedup Using Parallel Computing</b> .....	<b>2-55</b>
When to Use Parallel Computing for Parameter Estimation .....	<b>2-55</b>
How Parallel Computing Speeds Up Estimation .....	<b>2-56</b>
<b>How to Use Parallel Computing (GUI)</b> .....	<b>2-59</b>
Configure Your System for Parallel Computing .....	<b>2-59</b>
Specify Model Dependencies .....	<b>2-60</b>
Estimate Model Parameters .....	<b>2-61</b>
Troubleshooting .....	<b>2-65</b>
<b>Estimate Initial States</b> .....	<b>2-68</b>
How to Estimate Initial States (GUI) .....	<b>2-68</b>
Estimating Initial Conditions for Blocks with External Initial Conditions .....	<b>2-71</b>
Estimate Initial States of a Mass-Spring-Damper System (GUI) .....	<b>2-71</b>
<b>Estimation Projects</b> .....	<b>2-82</b>
Structure of an Estimation Project .....	<b>2-82</b>
Managing Multiple Projects and Tasks .....	<b>2-83</b>
Adding, Deleting and Renaming an Estimation Project ...	<b>2-84</b>
Saving Control and Estimation Tools Manager Projects ..	<b>2-85</b>
Loading Control and Estimation Tools Manager Projects ..	<b>2-86</b>
<b>Estimate Parameters (Code)</b> .....	<b>2-87</b>
How to Estimate Parameters (Code) .....	<b>2-87</b>

F14 Parameters and Initial State Estimation (Code) . . . . .	2-89
RC Circuit Parameters and Initial State Estimation (Code) . . . . .	2-100
Parameter Estimation Objects . . . . .	2-105
Creating Transient Experiment Data Objects . . . . .	2-106
Creating Estimation Objects . . . . .	2-109
Parameter Objects . . . . .	2-117
State Objects . . . . .	2-120
Transient Data Objects . . . . .	2-123
State Data Objects . . . . .	2-129
How to Use Parallel Computing (Code) . . . . .	2-132

## Response Optimization

# 3

<b>Response Optimization Workflow</b> . . . . .	3-3
<b>How the Optimization Algorithm Formulates</b>	
<b>Minimization Problems</b> . . . . .	3-4
Feasibility Problem and Constraint Formulation . . . . .	3-4
Tracking Problem . . . . .	3-7
Gradient Descent Method Problem Formulations . . . . .	3-8
Simplex Search Method Problem Formulations . . . . .	3-10
Pattern Search Method Problem Formulations . . . . .	3-11
Gradient Computations . . . . .	3-12
<b>Specify Signals to Log (GUI)</b> . . . . .	3-14
<b>Specifying Time Domain Design Requirements</b>	
<b>(GUI)</b> . . . . .	3-16
Specifying Piecewise-Linear Lower and Upper Bounds . . .	3-16
Specifying Step Response Characteristics . . . . .	3-21
Track Reference Signals . . . . .	3-25
Specifying Custom Requirements . . . . .	3-27
Editing Design Requirements . . . . .	3-29
<b>Specifying Frequency Domain Design Requirements</b>	
<b>(GUI)</b> . . . . .	3-32
Specifying Lower Bounds on Gain and Phase Margin . . . .	3-32



Specifying Piecewise-Linear Lower and Upper Bounds on Frequency Response .....	3-34
Specifying Bound on Closed-Loop Peak Gain .....	3-36
Specifying Lower Bound on Damping Ratio .....	3-38
Specifying Upper and Lower Bounds on Natural Frequency .....	3-40
Specifying Upper Bound on Approximate Settling Time ..	3-42
Specifying Piecewise-Linear Upper and Lower Bounds on Singular Values .....	3-44
Specifying Step Response Characteristics .....	3-47
Specifying Custom Requirements .....	3-50
<b>Design Variables (GUI) .....</b>	<b>3-54</b>
Specifying Design Variables .....	3-54
Specifying Independent Parameters .....	3-55
Specify Independent Parameters to Optimize .....	3-56
<b>General Options .....</b>	<b>3-60</b>
Accessing General Options .....	3-60
Progress Options .....	3-60
Result Options .....	3-61
<b>Optimization Options .....</b>	<b>3-64</b>
Accessing Optimization Options .....	3-64
Selecting Optimization Methods .....	3-65
Selecting Optimization Termination Options .....	3-66
Selecting Additional Optimization Options .....	3-67
<b>Creating Linearization I/O Sets (GUI) .....</b>	<b>3-69</b>
Create Linearization I/O Set .....	3-69
<b>Linearization Options .....</b>	<b>3-71</b>
Accessing Linearization Options .....	3-71
Configuring Linearization Options .....	3-71
<b>Plots .....</b>	<b>3-74</b>
Adding Plots in Design Optimization Tool .....	3-74
Plotting Current Response .....	3-74
Plotting Intermediate Steps .....	3-74
Modifying Plot Properties .....	3-74
Plot Types .....	3-76

<b>Compare Requirements and Design Variables Using Spider Plot</b> .....	<b>3-81</b>
<b>Run the Optimization (GUI)</b> .....	<b>3-85</b>
<b>Design Optimization to Meet Time- and Frequency-Domain Requirements</b> .....	<b>3-88</b>
<b>Design Optimization to Meet a Custom Objective Using the GUI</b> .....	<b>3-103</b>
<b>Design Optimization to Meet a Custom Objective at the Command Line</b> .....	<b>3-123</b>
<b>Design Optimization to Meet Custom Signal Requirements (GUI)</b> .....	<b>3-133</b>
<b>Design Optimization to Meet Frequency-Domain Requirements</b> .....	<b>3-138</b>
<b>Optimizing Parameters for Robustness</b> .....	<b>3-157</b>
What Is Robustness? .....	<b>3-157</b>
Sampling Methods for Uncertain Parameters .....	<b>3-158</b>
Optimize Parameters for Robustness (GUI) .....	<b>3-162</b>
<b>Accelerating Model Simulations During Optimization</b> .....	<b>3-171</b>
About Accelerating Optimization .....	<b>3-171</b>
Limitations .....	<b>3-171</b>
Setting Accelerator Mode for Response Optimization ....	<b>3-171</b>
<b>Speedup Using Parallel Computing</b> .....	<b>3-173</b>
When to Use Parallel Computing for Response Optimization .....	<b>3-173</b>
How Parallel Computing Speeds Up Optimization .....	<b>3-174</b>
<b>How to Use Parallel Computing</b> .....	<b>3-178</b>
Configure Your System for Parallel Computing .....	<b>3-178</b>
Specify Model Dependencies .....	<b>3-179</b>

Optimize Design Using Parallel Computing (GUI) . . . . .	3-180
Optimize Design Using Parallel Computing (Code) . . . . .	3-184
<b>Refining and Troubleshooting Optimization Results . .</b>	<b>3-186</b>
Troubleshooting Optimization Results . . . . .	3-186
<b>Managing Design Optimization Tool Session . . . . .</b>	<b>3-196</b>
Saving a Session . . . . .	3-196
Loading a Session . . . . .	3-196

## Optimization-Based Control Design

# 4

<b>Overview of Optimization-Based Compensator Design . . . . .</b>	<b>4-2</b>
<b>Time Domain Design Requirements in Simulink . . . . .</b>	<b>4-4</b>
Specifying Piecewise-Linear Lower and Upper Bounds . . .	4-4
Specifying Step Response Characteristics . . . . .	4-9
Track Reference Signals . . . . .	4-13
Specifying Custom Requirements . . . . .	4-15
Editing Design Requirements . . . . .	4-17
<b>Frequency-Domain Design Requirements in Simulink . . . . .</b>	<b>4-20</b>
Specifying Lower Bounds on Gain and Phase Margin . . . .	4-20
Specifying Piecewise-Linear Lower and Upper Bounds on Frequency Response . . . . .	4-22
Specifying Bound on Closed-Loop Peak Gain . . . . .	4-24
Specifying Lower Bound on Damping Ratio . . . . .	4-26
Specifying Upper and Lower Bounds on Natural Frequency . . . . .	4-28
Specifying Upper Bound on Approximate Settling Time . .	4-30
Specifying Piecewise-Linear Upper and Lower Bounds on Singular Values . . . . .	4-32
Specifying Step Response Characteristics . . . . .	4-35
Specifying Custom Requirements . . . . .	4-38

<b>Time- and Frequency-Domain Requirements in SISO</b>	
<b>Design Tool</b> .....	4-42
Root Locus Diagrams .....	4-42
Open-Loop and Prefilter Bode Diagrams .....	4-44
Open-Loop Nichols Plots .....	4-45
Step/Impulse Response Plots .....	4-46
<b>Time-Domain Simulations in SISO Design Tool</b> .....	4-47
<b>Designing Optimization-Based Controllers for LTI</b>	
<b>Systems</b> .....	4-48
How to Design Optimization-Based Controllers for LTI	
Systems .....	4-48
Example — Frequency-Domain Optimization for LTI	
System .....	4-49
<b>Designing Linear Controllers for Simulink Models</b> ...	4-70

## Lookup Tables

# 5

<b>What Are Lookup Tables?</b> .....	5-2
Static Lookup Tables .....	5-2
Adaptive Lookup Tables .....	5-3
<b>Estimating Values of Lookup Tables</b> .....	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
<b>Capturing Time-Varying System Behavior Using</b>	
<b>Adaptive Lookup Tables</b> .....	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 .....	5-59
--	------

## Examples

### A

Parameter Estimation .....	A-2
Parameter Optimization .....	A-3
Optimization-Based Linear Control Design .....	A-4
Lookup Tables .....	A-5

## Index



# Data Analysis and Processing

---

- “Model Requirements for Importing Data” on page 1-2
- “Import Data (GUI)” on page 1-4
- “Plot and Analyze Data (GUI)” on page 1-12
- “Preprocess Data (GUI)” on page 1-15
- “Add Preprocessed Data Sets to an Estimation Project (GUI)” on page 1-30
- “Export Prepared Data to the MATLAB Workspace” on page 1-33

## Model Requirements 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

---

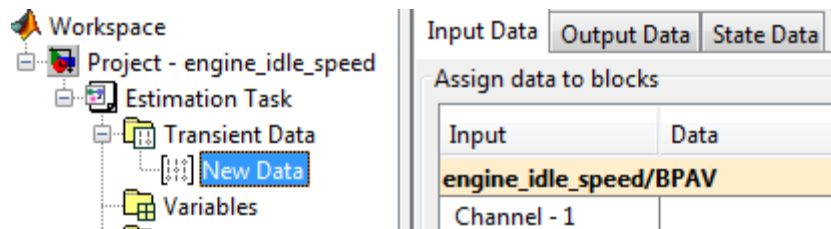
**Note** You do not need an Inport block if your model already contains a fixed input block, such as a Step block.

---

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

To enable signal logging, right-click the signal and select **Signal Properties**. In the **Signal Properties** dialog box, select the **Log signal data** check box. For more information, see “Exporting Signal Data Using Signal Logging” 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 Output blocks or logged signals in the model.

Adding an Inport or Output 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.

### **More About**

“Supported Data”

## Import Data (GUI)

In this section...
“Creating an Estimation Project” on page 1-4
“Importing Time-Domain Data into the GUI” on page 1-6
“Importing Time-Series Data into the GUI” on page 1-11
“Importing Complex Data into the GUI” on page 1-11

### 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.

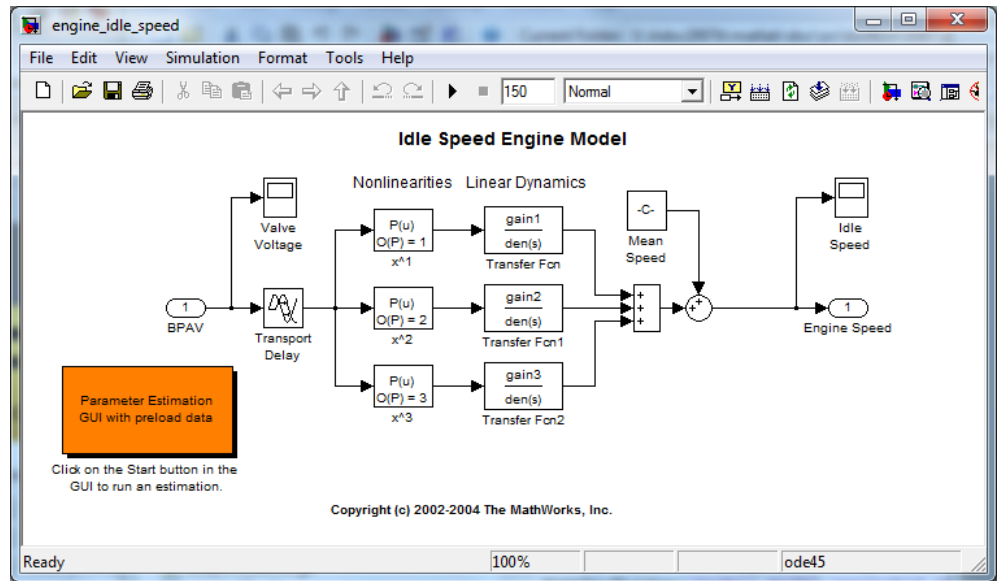
To create an estimation project:

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

```
engine_idle_speed
```

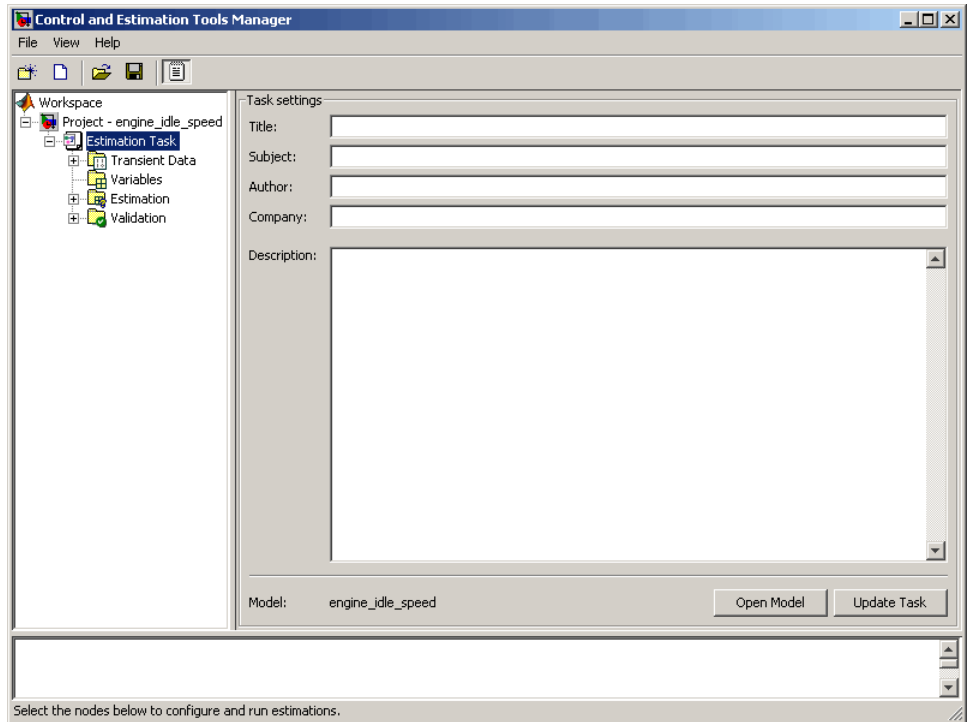
at the MATLAB® prompt.

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 “Model Requirements 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.

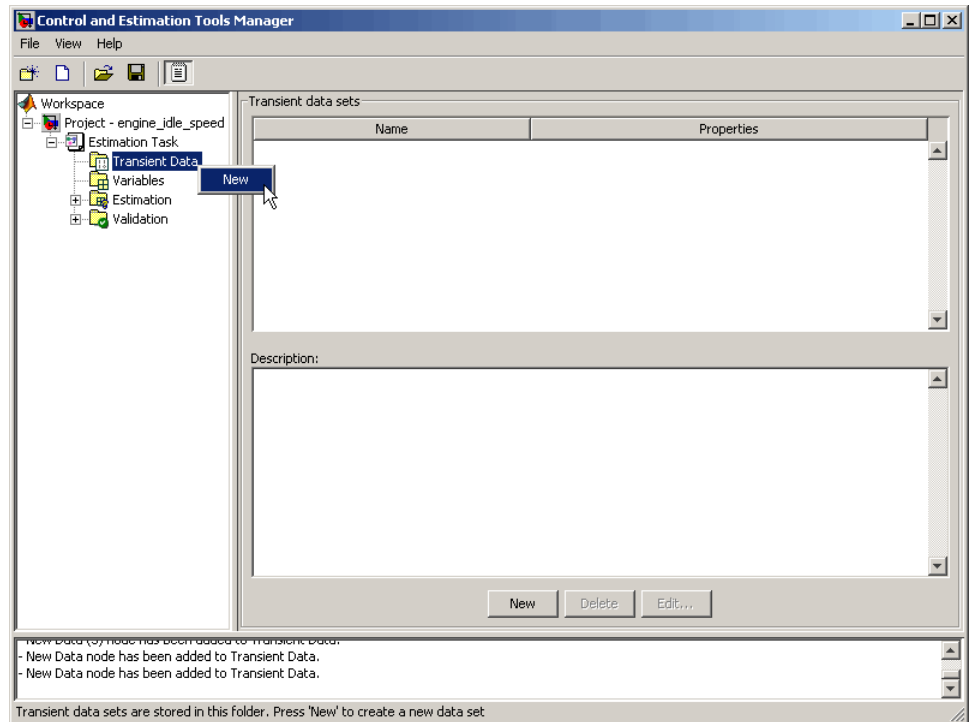
---

## Importing Time-Domain Data into the GUI

After you create an estimation project, as described in “Creating an Estimation Project” on page 1-4, you can import the estimation data into the GUI. To learn more about the types of data for parameter estimation, see “Supported Data” in the *Simulink Design Optimization Getting Started Guide*.

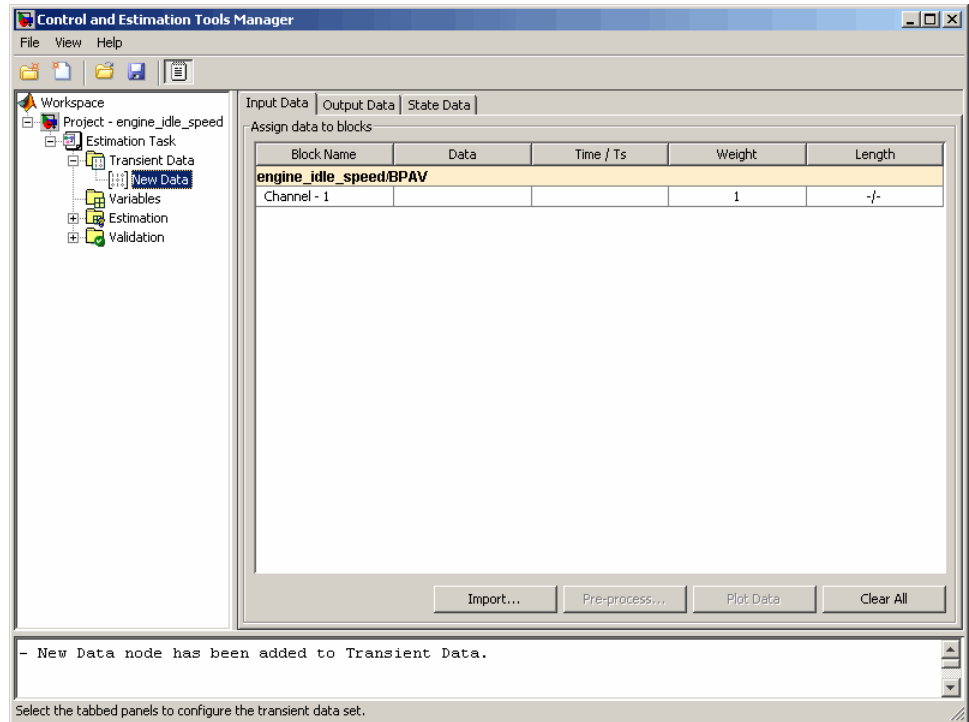
To import *transient* (measured) data for your dynamic system:

- 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 “Model Requirements 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:

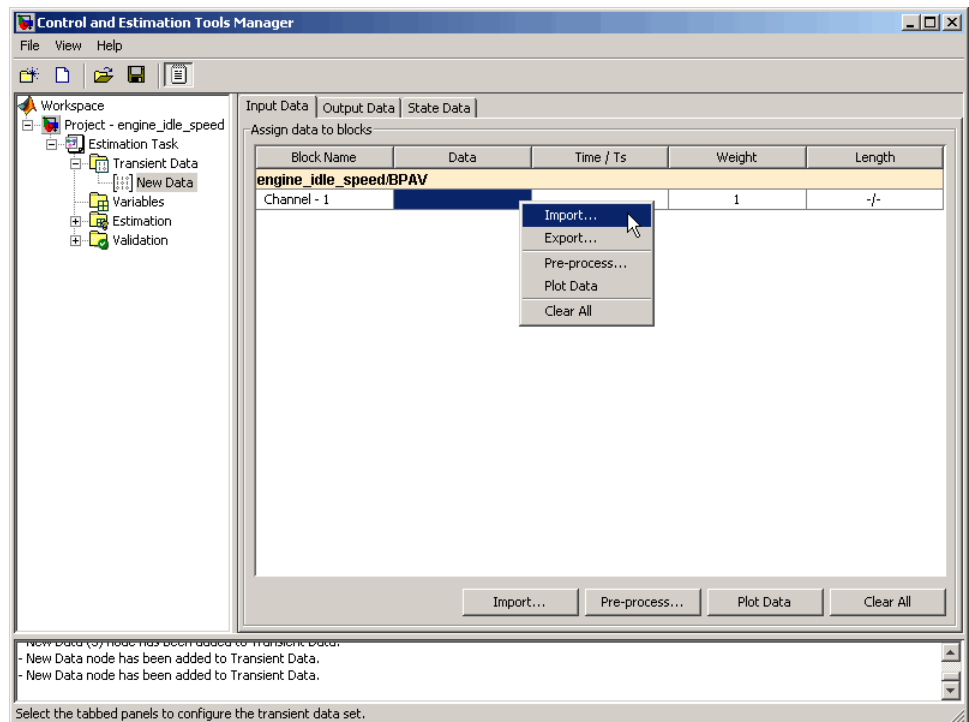
- “Importing Input Data and Time Vector” on page 1-9

- “Importing Output Data and Time Vector” on page 1-10

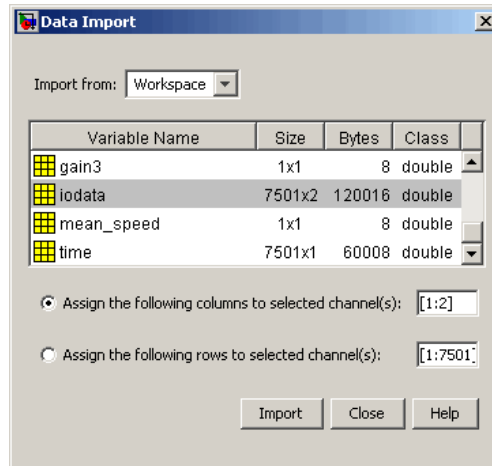
## Importing Input Data and Time Vector

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 1-6.

## 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 1-6.
- 3 Specify both the data sets together as estimation data, as described in “Specify Estimation Data (GUI)” on page 2-3.
- 4 Estimate the parameters, as described in “Estimate Parameters (GUI)” on page 2-39.

## Plot and Analyze Data (GUI)

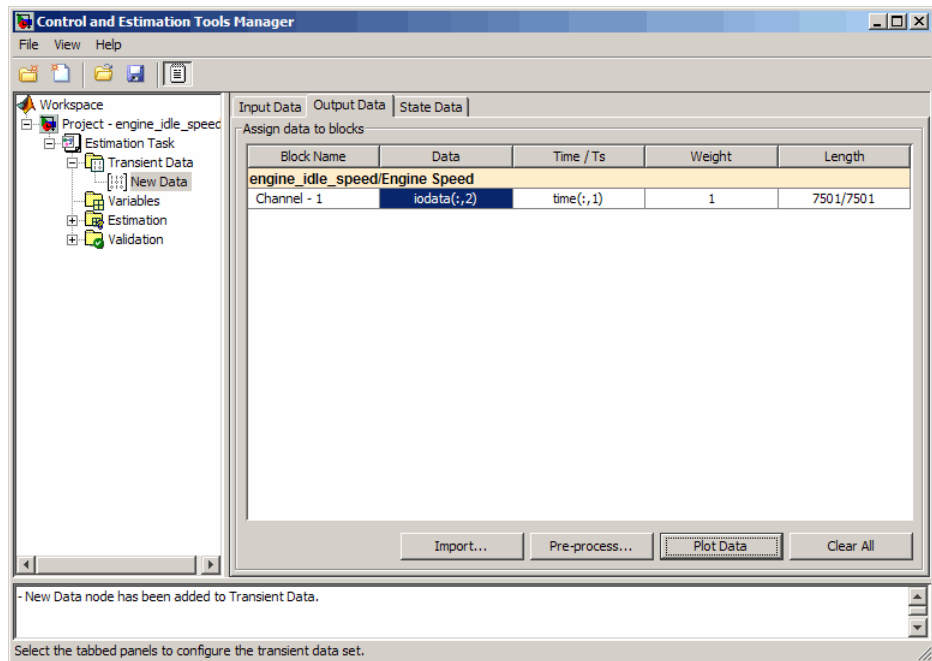
In this section...
“Why Plot the Data Before Parameter Estimation” on page 1-12
“How To Plot Data in the GUI” on page 1-12

### Why Plot the Data Before Parameter Estimation

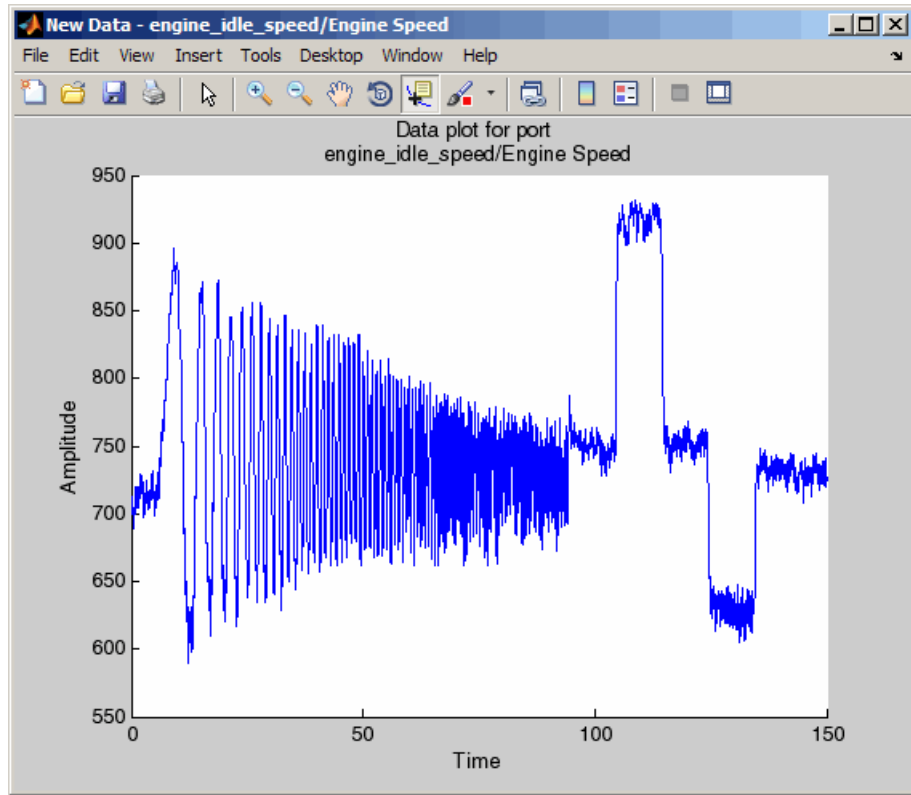
After you import the estimation data, as described in “Import Data (GUI)” on page 1-4, it is useful to remove outliers, smooth, detrend, or otherwise treat the data to make it more tractable for analysis and estimation purposes. To view and analyze the data characteristics, you must plot the data on 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 preprocess the data as described in “Preprocess Data (GUI)” on page 1-15.

## Preprocess Data (GUI)

### In this section...

“Ways to Preprocess Data Using the Data Preprocessing Tool” on page 1-15

“Opening the Data Preprocessing Tool” on page 1-15

“Handling Missing Data” on page 1-17

“Handling Outliers” on page 1-19

“Detrending Data” on page 1-19

“Filtering Data” on page 1-19

“Selecting Data” on page 1-21

## Ways to Preprocess Data Using the Data Preprocessing Tool

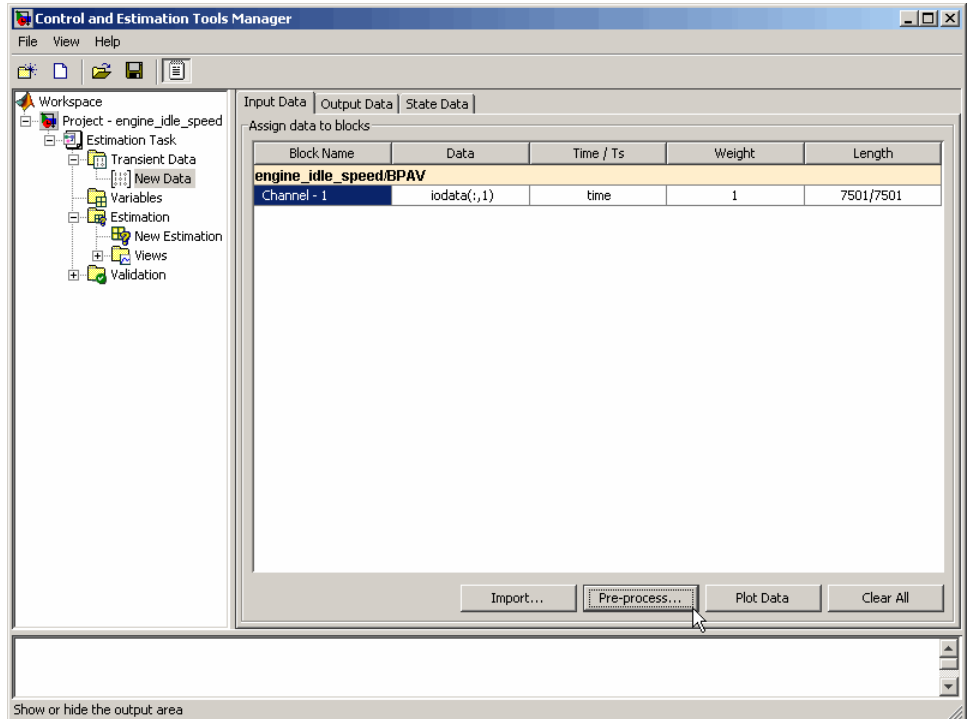
After you import the estimation data, as described in “Import Data (GUI)” on page 1-4, you can perform the following preprocessing operations using the Data Preprocessing Tool in Simulink Design Optimization software:

- 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.

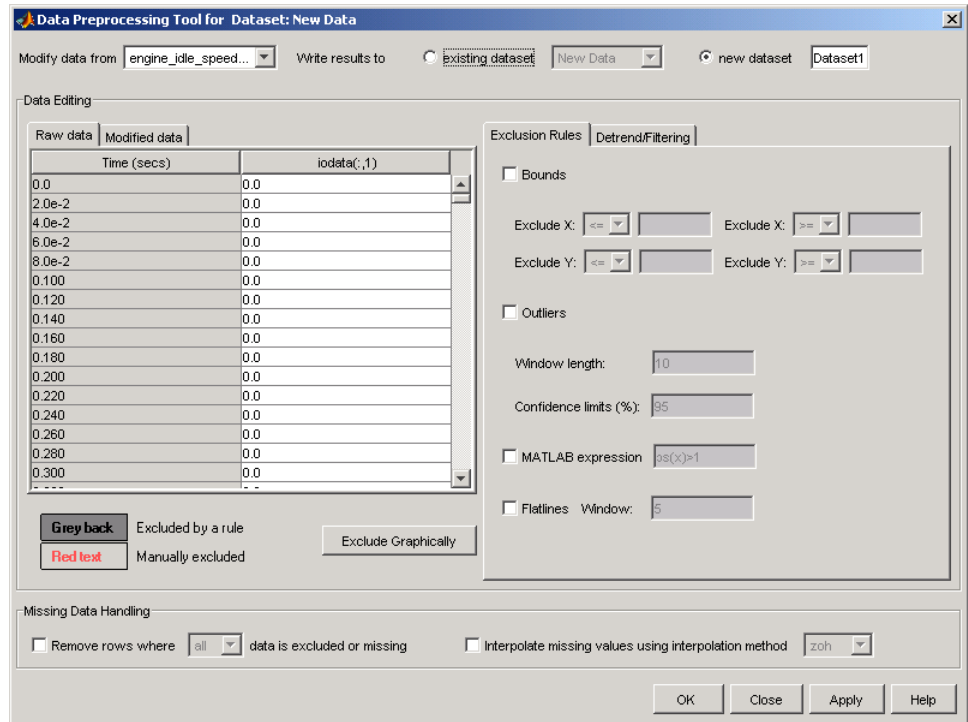
## Opening the Data Preprocessing Tool

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 “Model Requirements for Importing Data” on page 1-2, and “Creating an Estimation Project” on page 1-4.

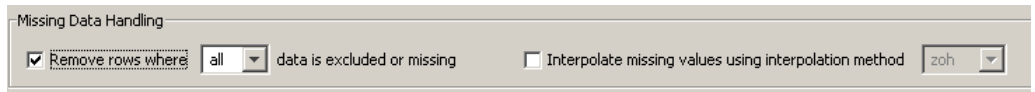
## Handling Missing Data

- “Removing Missing Data” on page 1-18

- “Interpolating Missing Data” on page 1-18

## 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 1-18.

## 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 1-19
- “How to Filter Data” on page 1-20

### Types of Filters

You have these choices for filtering your data:

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

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

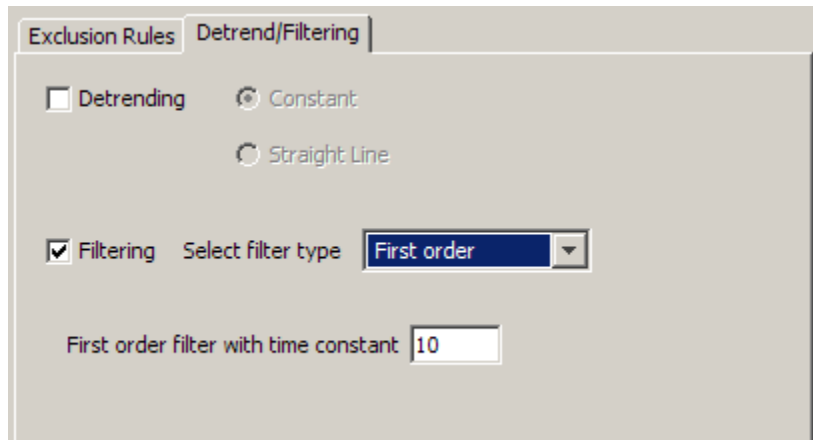
$$\frac{a_n s^n + a_{n-1} s^{n-1} + \dots + a_0}{b_m s^m + b_{m-1} s^{m-1} + \dots + b_0}$$

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 1-21
- “Graphically Selecting Data” on page 1-21
- “Using Rules to Select Data Samples” on page 1-24
- “Using the Data Table to Select Data Samples” on page 1-26

## 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.

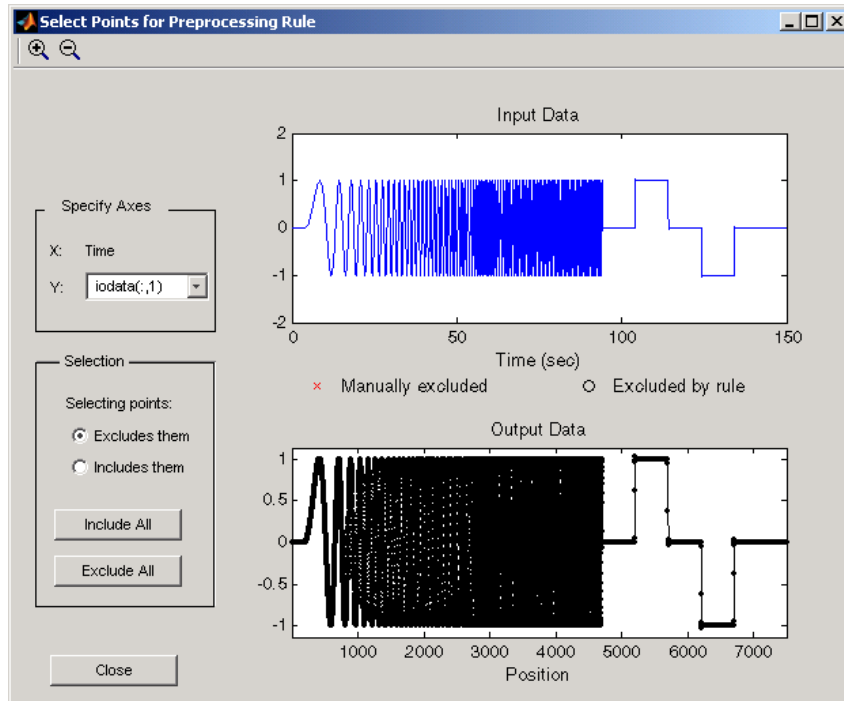
---

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

---

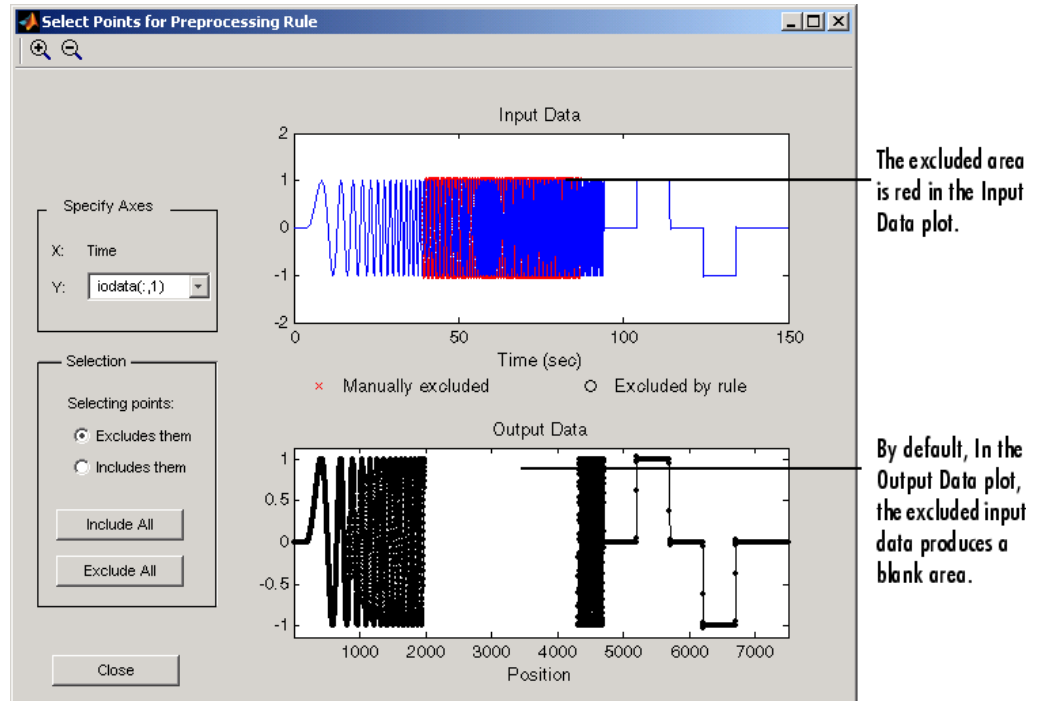
## Graphically Selecting Data

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.

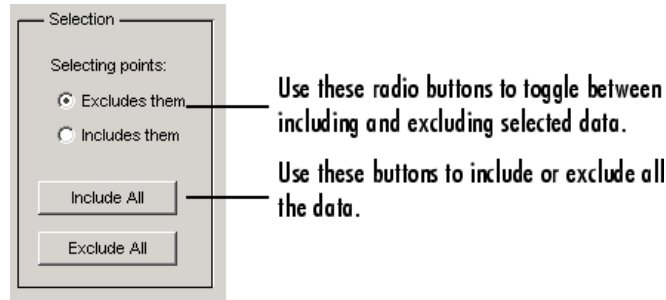
This figure shows an example of resulting data exclusion in the input data.



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.

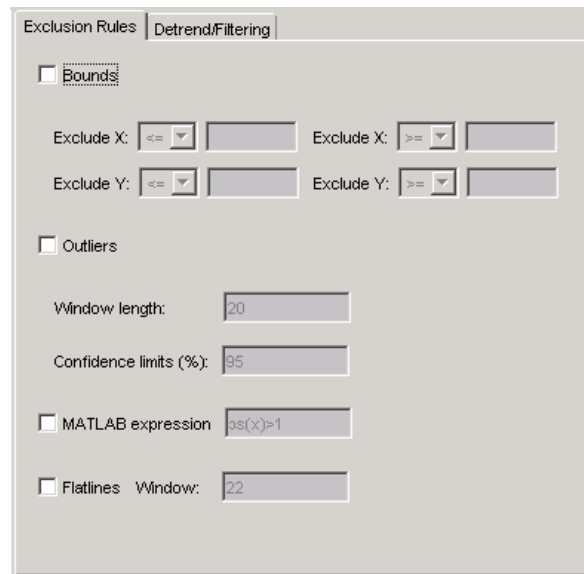
When you make changes in the Select Points for Preprocessing Rule window, they immediately appear in the **Data Editing** pane, and vice versa.

**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 1-25
- “MATLAB Expressions” on page 1-25

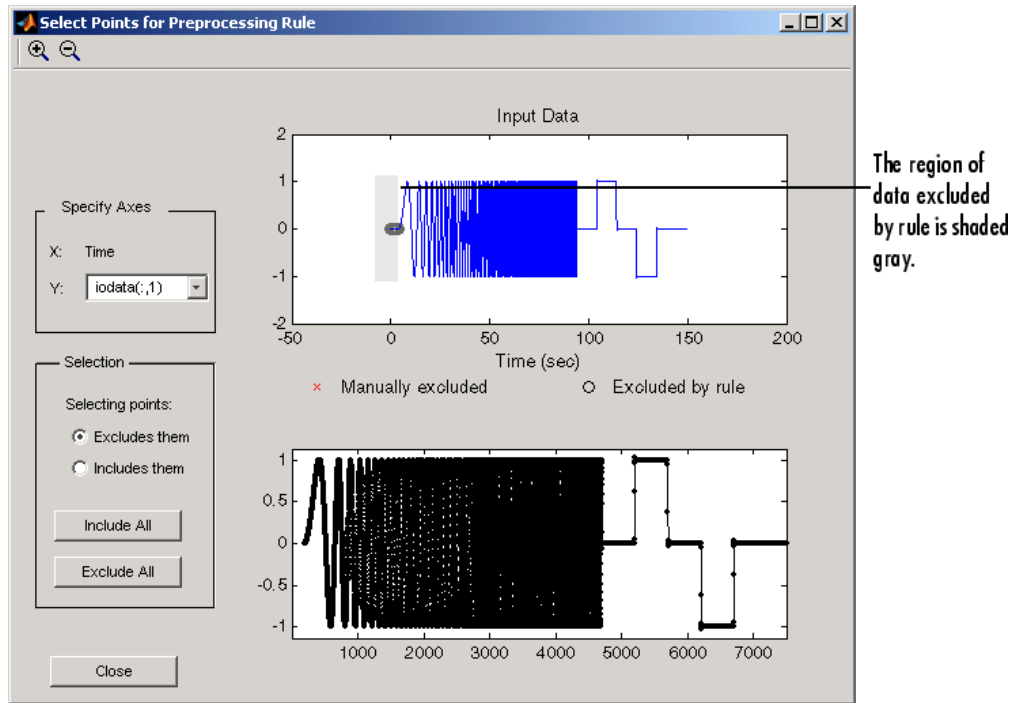
- “Flatlines” on page 1-25

**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.

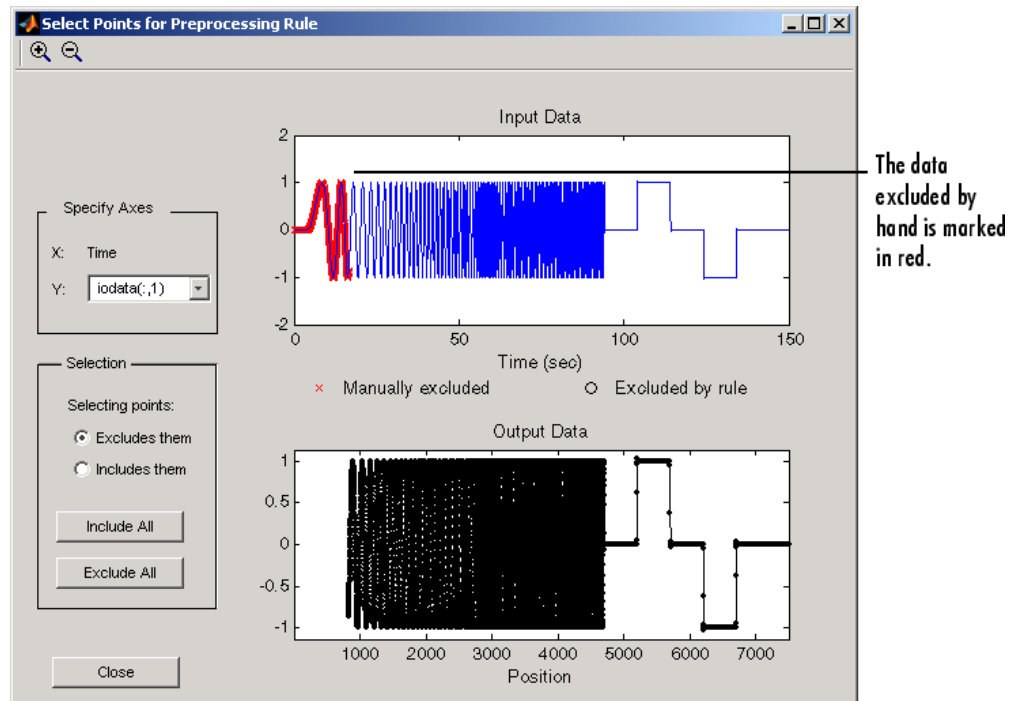
Time (secs)	iodata(:,1)*
8.760	0.979
8.780	0.976
8.800	0.972
8.820	0.969
8.840	NaN
8.860	NaN
8.880	NaN
8.900	NaN
8.920	NaN
8.940	NaN
8.960	NaN
8.980	NaN
9.0	NaN
9.20	NaN
9.40	NaN
9.60	NaN

Buttons: **Grey back** Excluded by a rule, **Red text** Manually excluded, **Exclude Graphically**

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 1-17 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.

## Add Preprocessed Data Sets to an Estimation Project (GUI)

After you preprocess the data using the techniques described in “Ways to Preprocess Data Using the Data Preprocessing Tool” on page 1-15, you can add the data set to an estimation project either by overwriting an existing data set or creating a new data set.

### In this section...

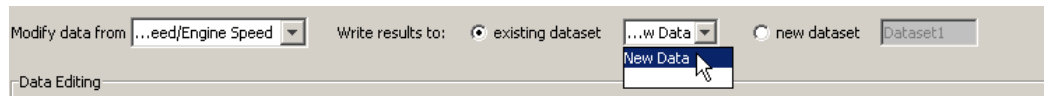
“Overwriting an Existing Data Set” on page 1-30

“Creating a New Data Set” on page 1-31

### Overwriting an Existing Data Set

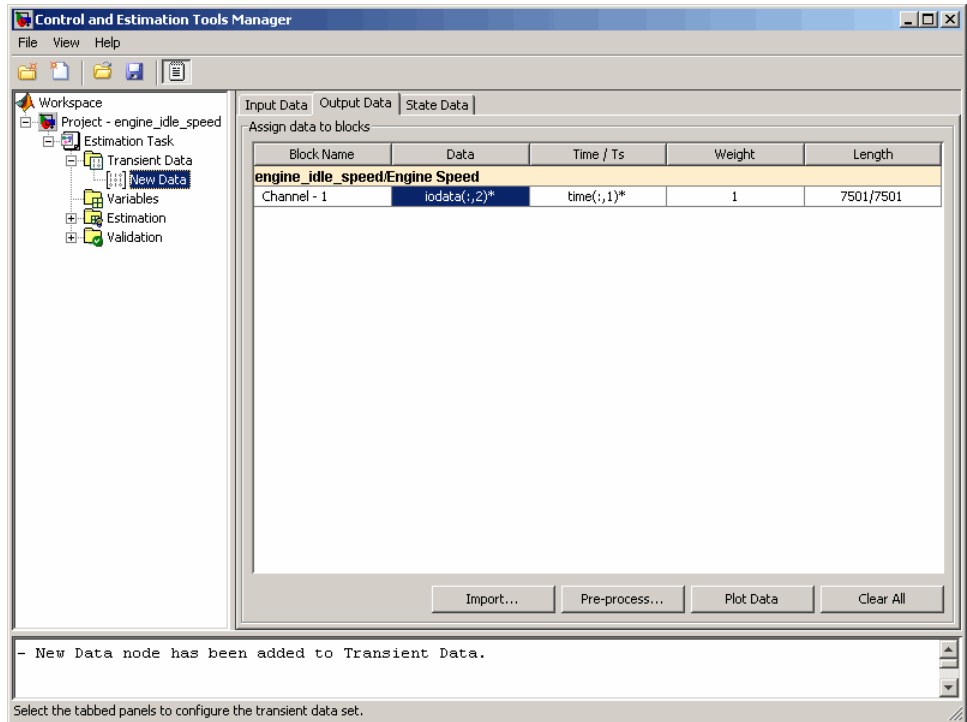
To overwrite an existing data set with the preprocessed data:

- 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 “Export Prepared Data to the MATLAB Workspace” on page 1-33.

## Creating a New Data Set

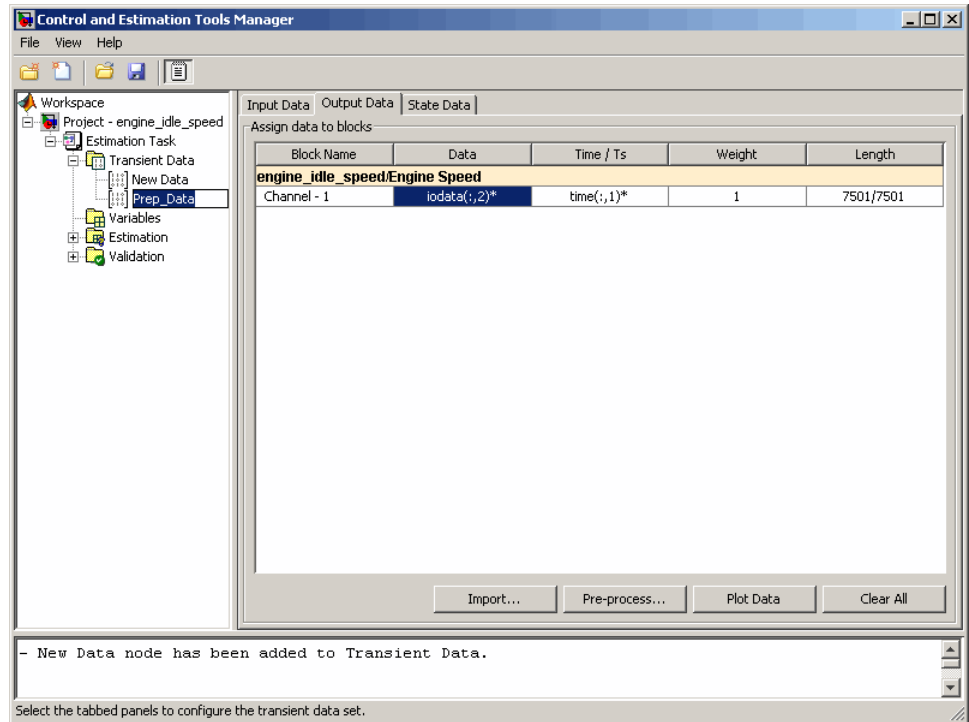
If you do not want to overwrite an existing data set with the preprocessed data, as described in “Overwriting an Existing Data Set” on page 1-30, you can create a new data set for the preprocessed data:

- 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 “Export Prepared Data to the MATLAB Workspace” on page 1-33.

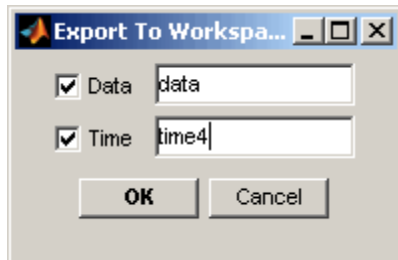
## Export Prepared Data to the MATLAB Workspace

After you add the preprocessed data to an estimation project, as described in “Add Preprocessed Data Sets to an Estimation Project (GUI)” on page 1-30, 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

---

- “Parameter Estimation Workflow” on page 2-2
- “Specify Estimation Data (GUI)” on page 2-3
- “Specify Parameters to Estimate (GUI)” on page 2-7
- “Specify Known Initial States (GUI)” on page 2-18
- “Progress Plots” on page 2-21
- “Estimation Options” on page 2-25
- “Simulation Options” on page 2-32
- “Progress Display Options” on page 2-38
- “Estimate Parameters (GUI)” on page 2-39
- “Validate Parameters (GUI)” on page 2-43
- “Accelerating Model Simulations During Estimation” on page 2-53
- “Speedup Using Parallel Computing” on page 2-55
- “How to Use Parallel Computing (GUI)” on page 2-59
- “Estimate Initial States” on page 2-68
- “Estimation Projects” on page 2-82
- “Estimate Parameters (Code)” on page 2-87

## Parameter Estimation Workflow

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 “Import Data (GUI)” on page 1-4 and “Preprocess Data (GUI)” on page 1-15, follow these steps to estimate model parameters:

- 1** Specify estimation data
- 2** Specify parameters to estimate
- 3** Specify known initial states or estimate the initial states
- 4** (Optional) Specify plots to view estimation progress
- 5** (Optional) Specify estimation and simulation options
- 6** Estimate the parameters
- 7** Validate the estimated parameters and states

---

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

---

To learn how to estimate parameters at the command line, see “Estimate Parameters (Code)” on page 2-87.

## Specify Estimation Data (GUI)

In this section...
“Creating an Estimation Task” on page 2-3
“How to Specify Data” on page 2-4

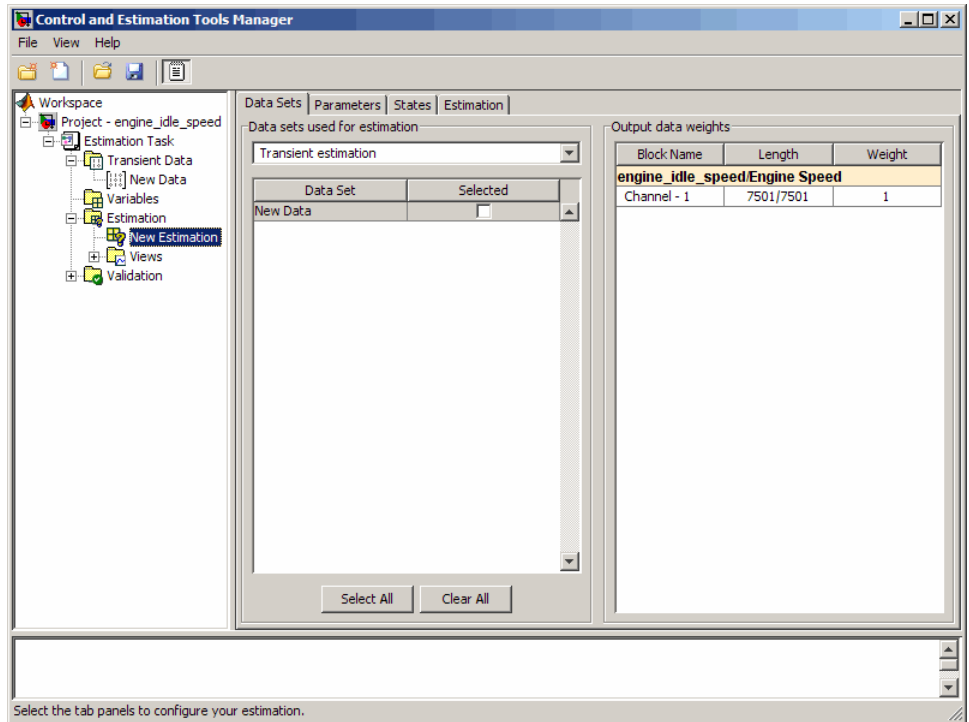
### Creating an Estimation Task

After you import the transient data, as described in “Import Data (GUI)” on page 1-4, you must create an estimation task and configure the estimation settings. If your data contains noise or outliers, you must also preprocess the data, as described in “Preprocess Data (GUI)” on page 1-15.

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.

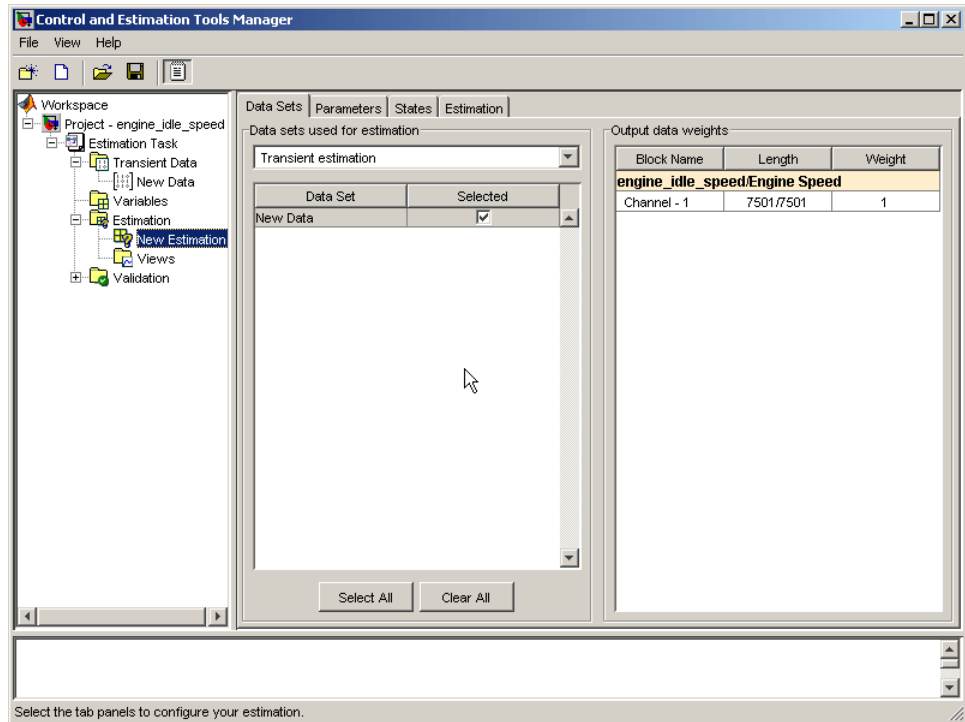
The Control and Estimation Tools Manager now resembles the next figure.



### How to Specify Data

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.

## Specify Parameters to Estimate (GUI)

### In this section...

“Choosing Which Parameters to Estimate First” on page 2-7

“How to Specify Parameters for Estimation” on page 2-7

“Specifying Initial Guesses and Upper/Lower Bounds” on page 2-12

“Specifying Parameter Dependency” on page 2-14

“Specify Independent Parameters to Estimate (GUI)” on page 2-15

### Choosing Which Parameters to Estimate First

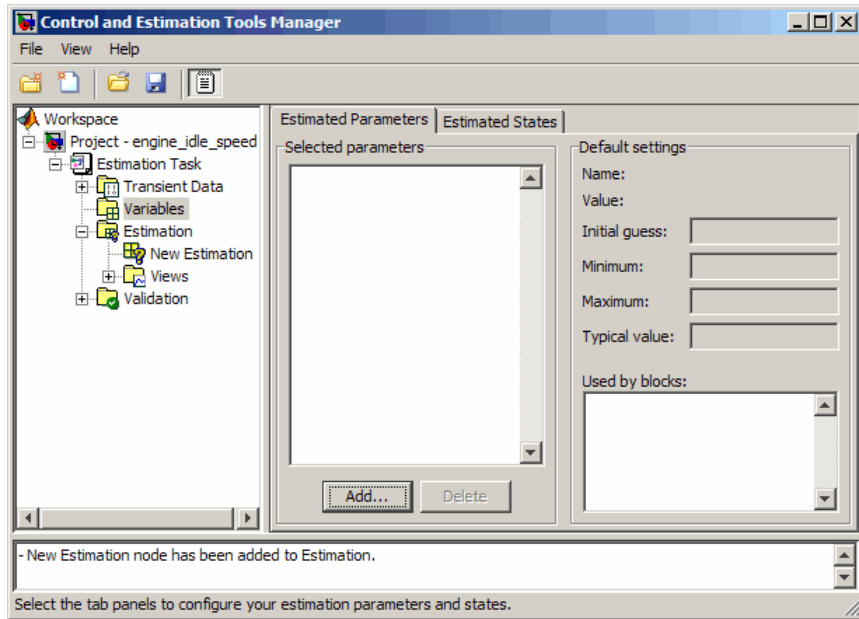
Simulink Design Optimization software lets you estimate scalar, vector and matrix parameters. Estimating model parameters is an iterative process. Often, it is more practical to estimate a small group of parameters and use the final estimated values as a starting point for further estimation of parameters 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

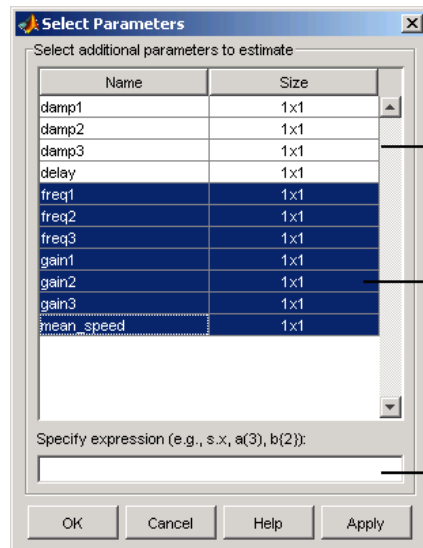
To select parameters for estimation:

- 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 2-14.

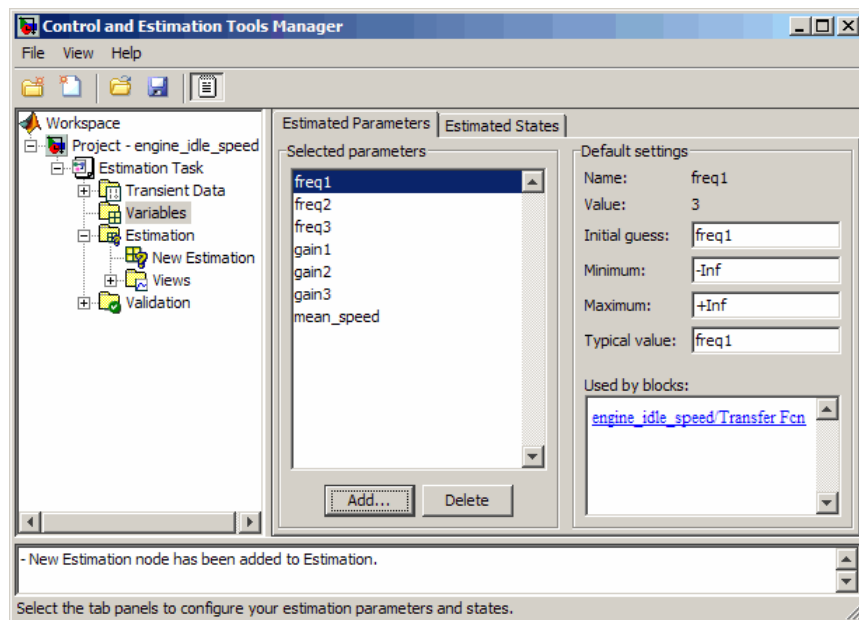
- 3 Select the last seven parameters: `freq1`, `freq2`, `freq3`, `gain1`, `gain2`, `gain3`, and `mean_speed`, and then click **OK**.

---

**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 2-12.

- 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 2-12.

---

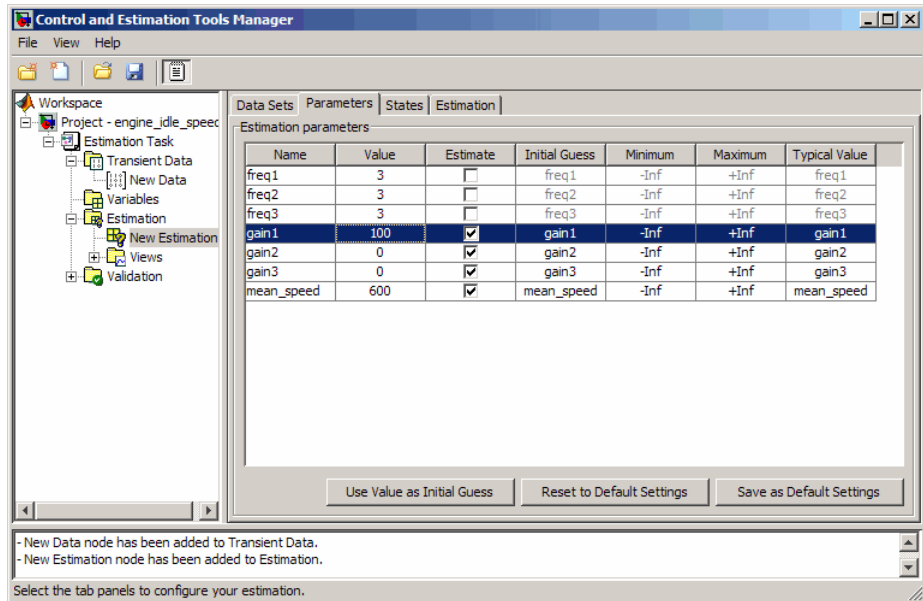
**Note** When you specify the **Minimum** and **Maximum** values for the parameters here, it does not affect your settings in the **Variables** node. You make these choices on a per estimation basis. You can move data to and from the **Variables** node into the **Estimation** node.

---

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.

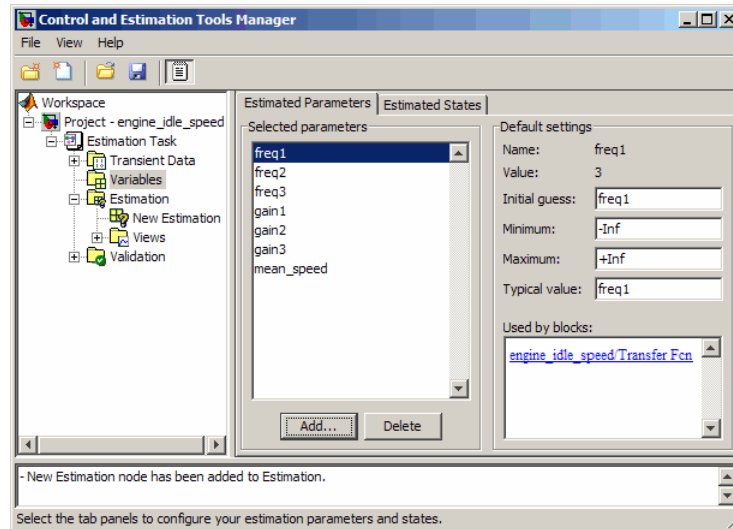
If you have good reason to believe a parameter lies within a finite range, it is usually best not to use the default minimum and maximum values. Often, there are computational advantages in specifying finite bounds if you can. It can be very important to specify lower and upper bounds. For example, if 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 or 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 “Specify Independent Parameters to Estimate (GUI)” on page 2-15.

## Specify Independent Parameters to Estimate (GUI)

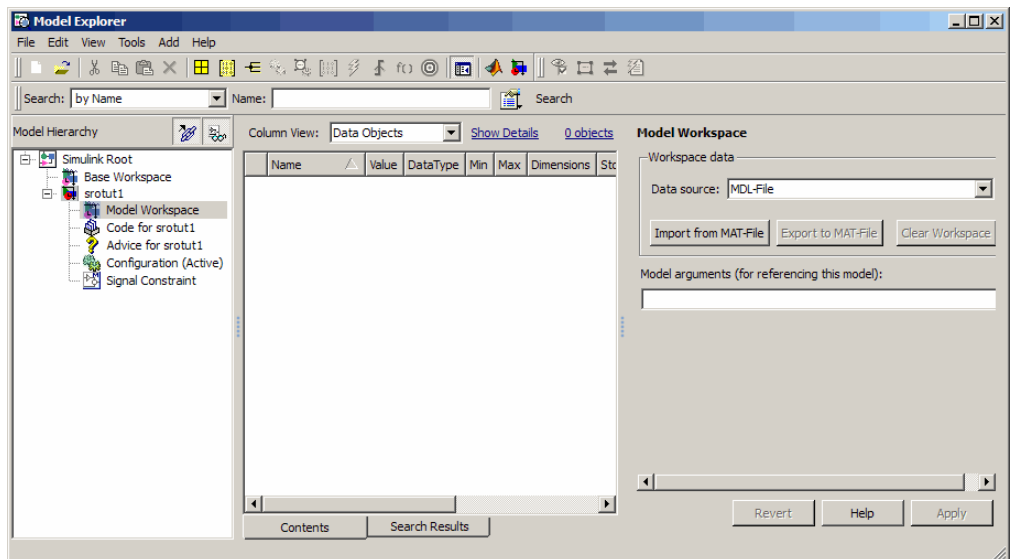
Assume that the parameter `Kint` in the model `srotut1` is related to the parameters `x` and `y` according to the relationship  $Kint = x + y$ . Also assume that the initial values of `x` and `y` are 1 and -0.7 respectively. To estimate `x` and `y` instead of `Kint`, first define these parameters in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

- 1 At the MATLAB prompt, type

```
srotut1
```

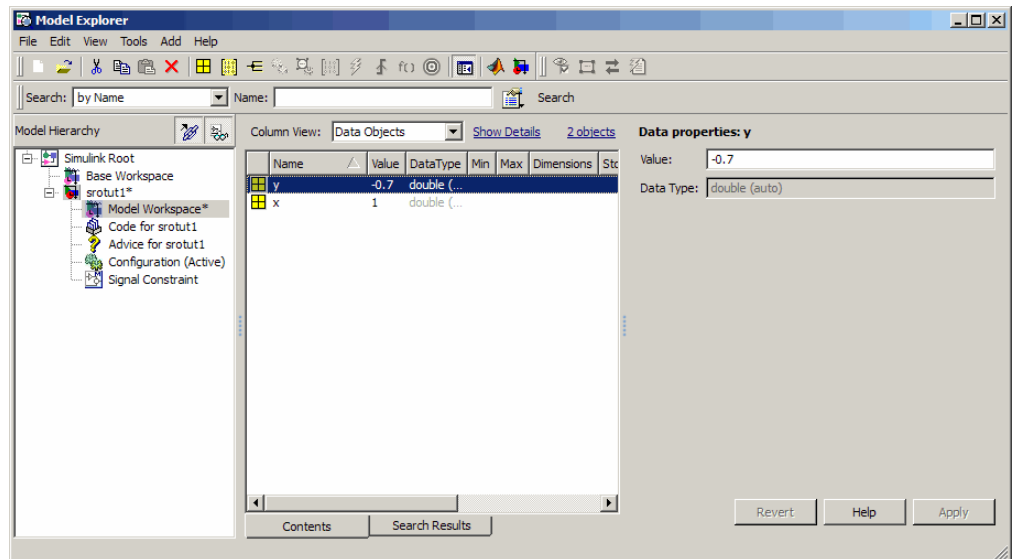
This opens the `srotut1` model window.

- 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:

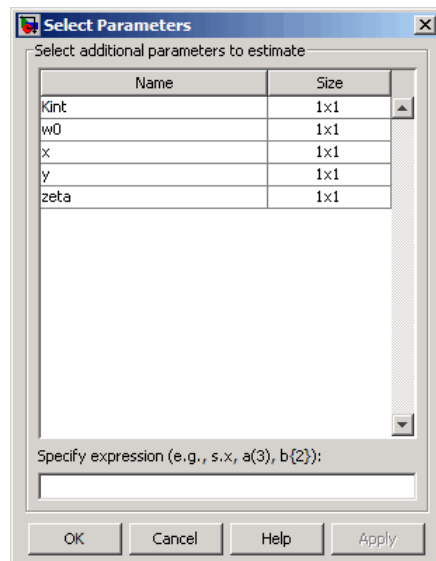
```
wks = get_param(gcs, 'ModelWorkspace')
x = wks.evalin('x')
y = wks.evalin('y')
Kint = x+y;
```

---

**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.



## Specify Known Initial States (GUI)

In this section...
“When to Specify Initial States Versus Estimate Initial States” on page 2-18
“How to Specify Initial States in the GUI” on page 2-18

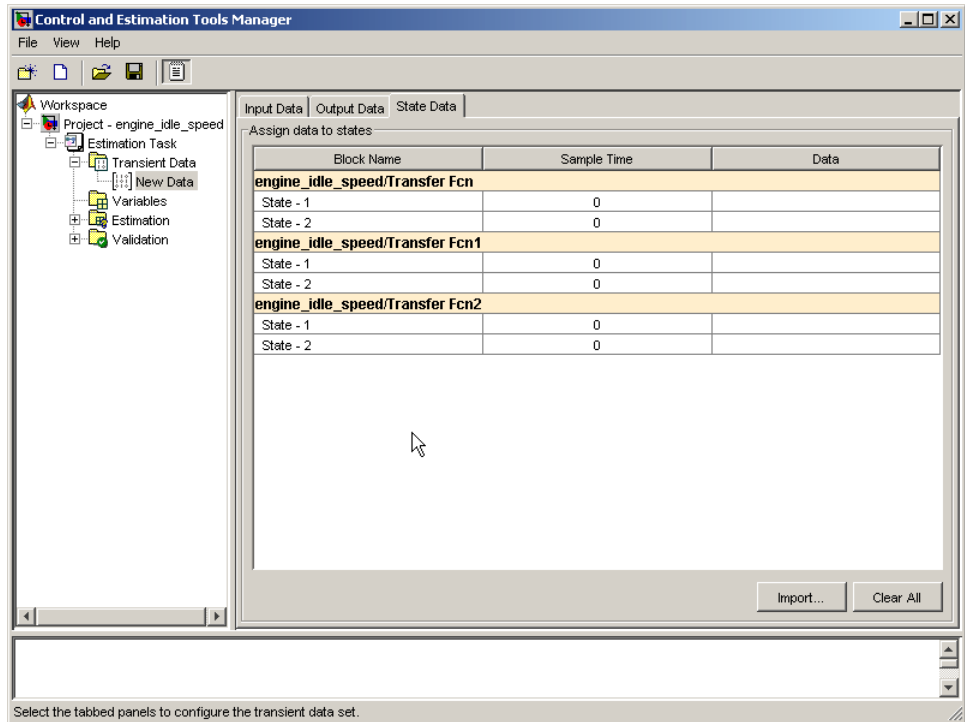
### When to Specify Initial States Versus Estimate Initial States

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 “Estimate Initial States” on page 2-68.

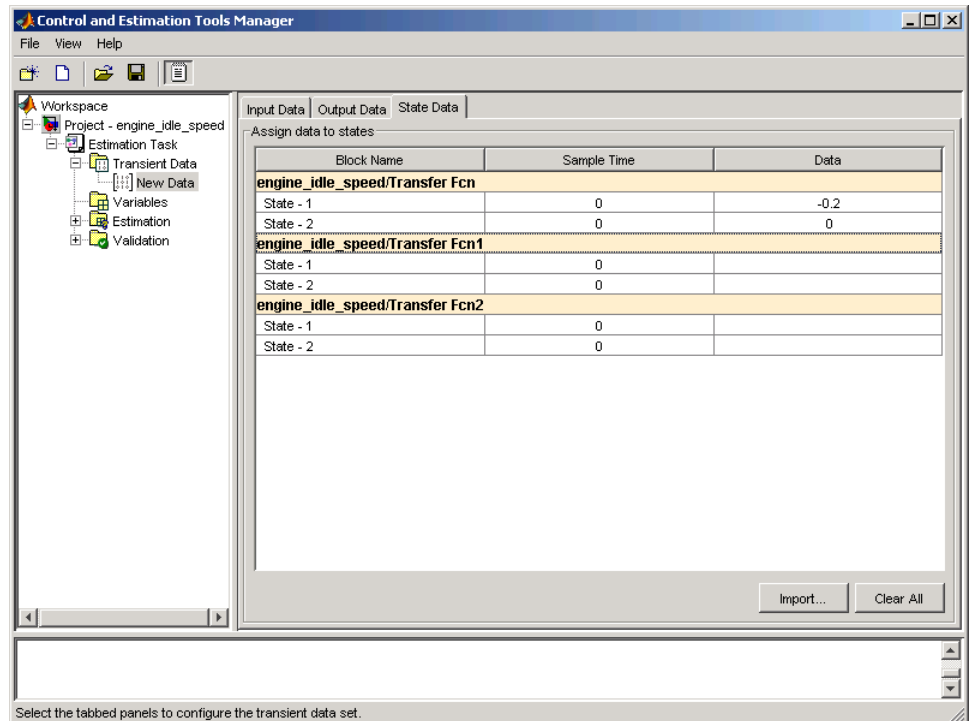
### How to Specify Initial States in the GUI

After you select parameters for estimation, as described in “Specify Parameters to Estimate (GUI)” on page 2-7, 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`.



## Progress Plots

In this section...
“Types of Plots” on page 2-21
“Basic Steps for Creating Plots” on page 2-21

### 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.

---

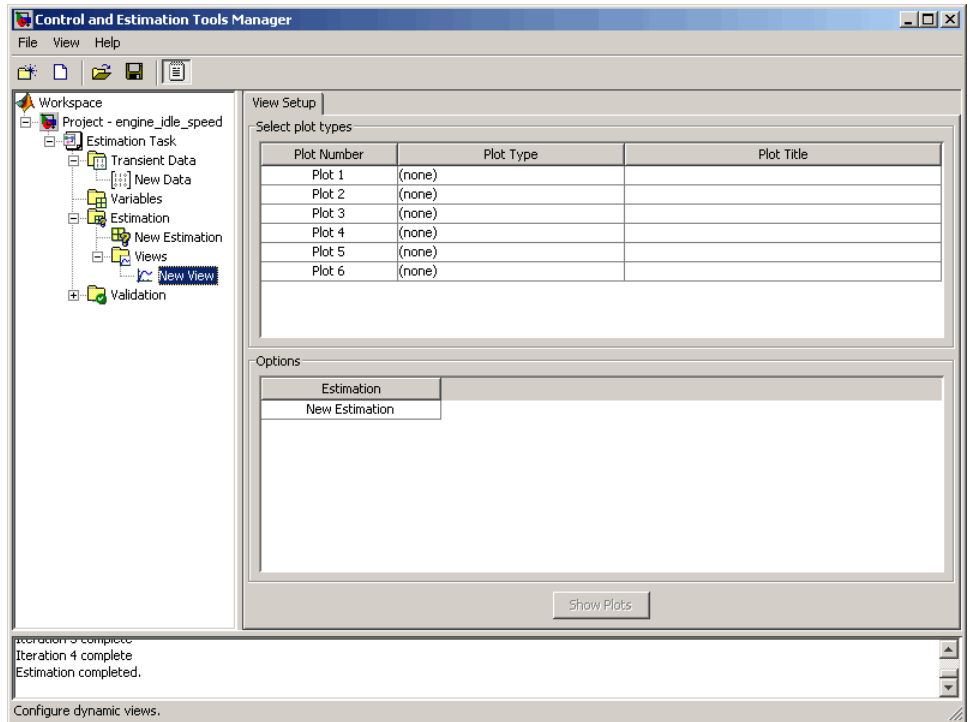
**Note** An estimation must be created before creating views. Otherwise, the **Options** table will be empty. To learn more, see “Creating an Estimation Task” on page 2-3.

---

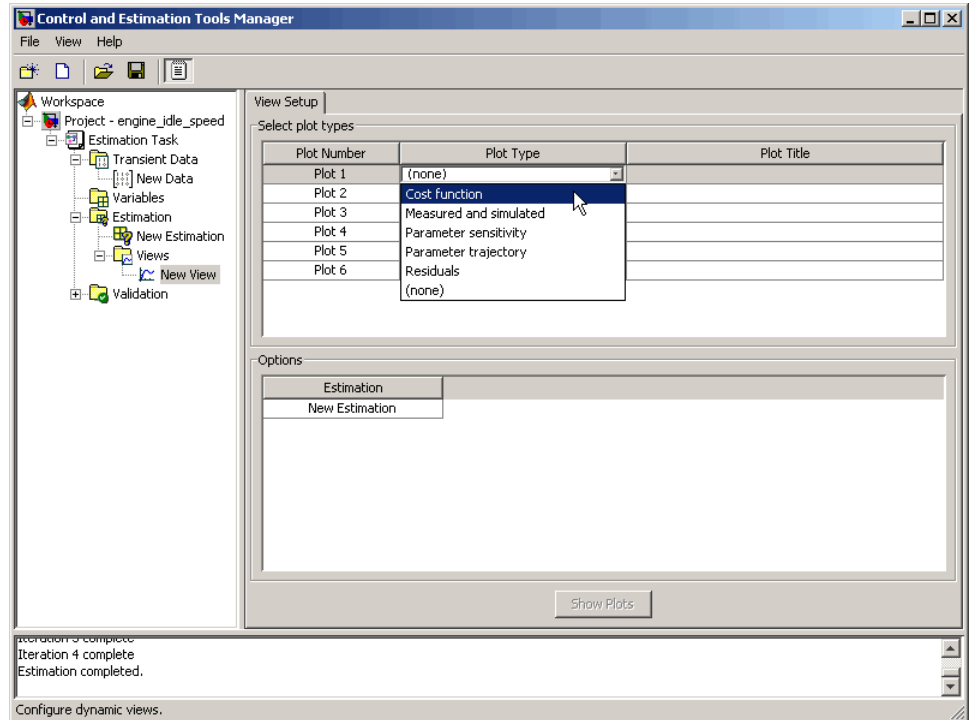
To create plots for viewing the estimation progress, follow the steps below:

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

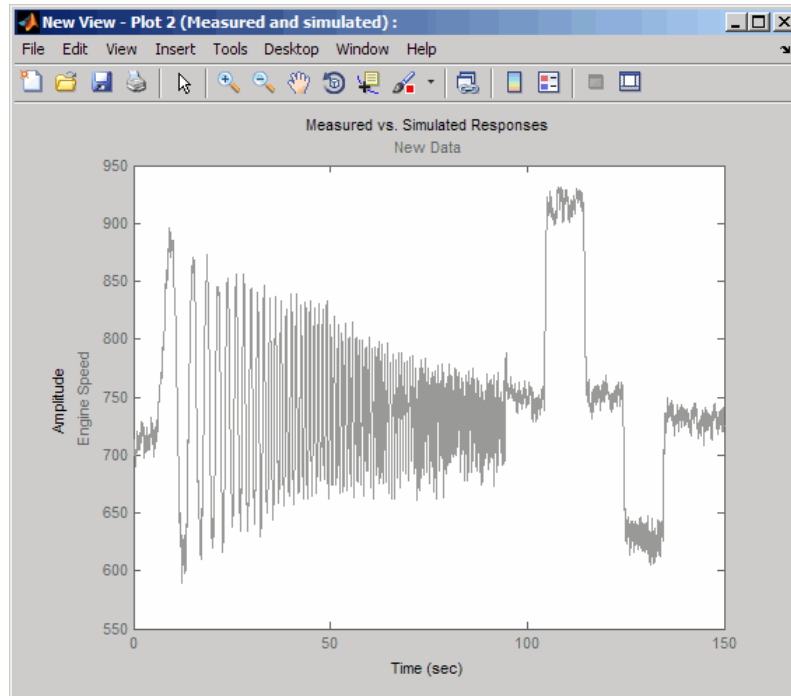
**2** In the **Workspace** tree, select **New View** to open the **View Setup** pane.



- 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.



## Estimation Options

### In this section...

“Accessing Estimation Options” on page 2-25

“Supported Estimation Methods” on page 2-27

“Selecting Optimization Termination Options” on page 2-29

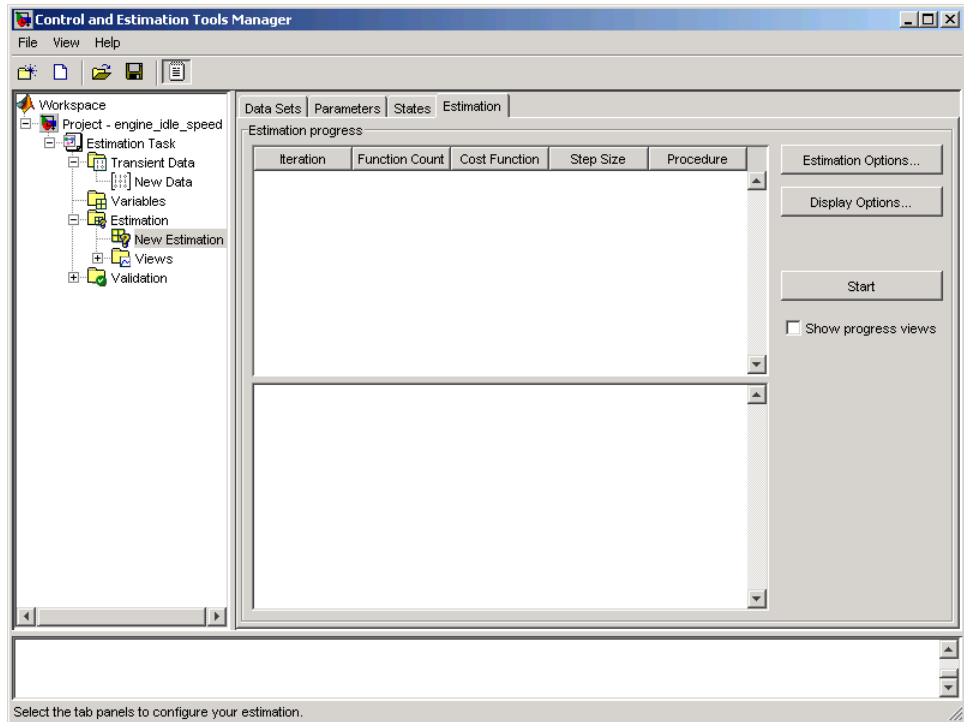
“Selecting Additional Optimization Options” on page 2-29

“Specifying Goodness of Fit Criteria (Cost Function)” on page 2-30

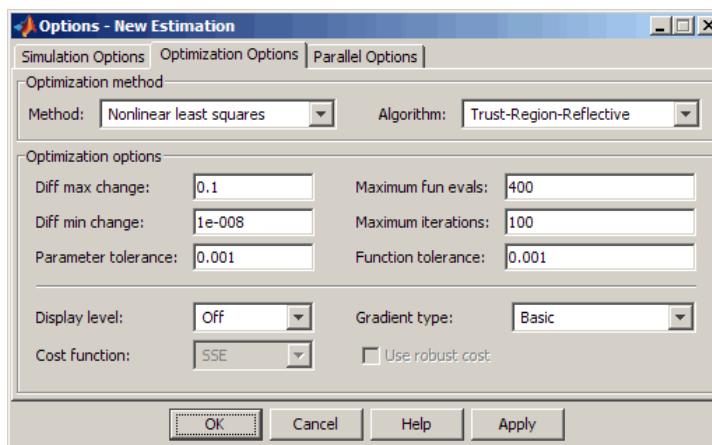
“How to Specify Estimation Options in the GUI” on page 2-30

### 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.

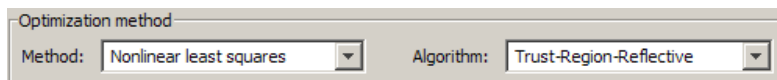


The following sections describe the estimation method settings and cost function:

- “Supported Estimation Methods” on page 2-27
- “Selecting Optimization Termination Options” on page 2-29
- “Selecting Additional Optimization Options” on page 2-29
- “Specifying Goodness of Fit Criteria (Cost Function)” on page 2-30

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

## Selecting Optimization Termination Options

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

The screenshot shows a dialog box titled "Optimization options" with the following fields:

Diff max change:	0.1	Maximum fun evals:	400
Diff min change:	1e-008	Maximum iterations:	100
Parameter tolerance:	1e-006	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 the bottom of the dialog box with the following options:

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.

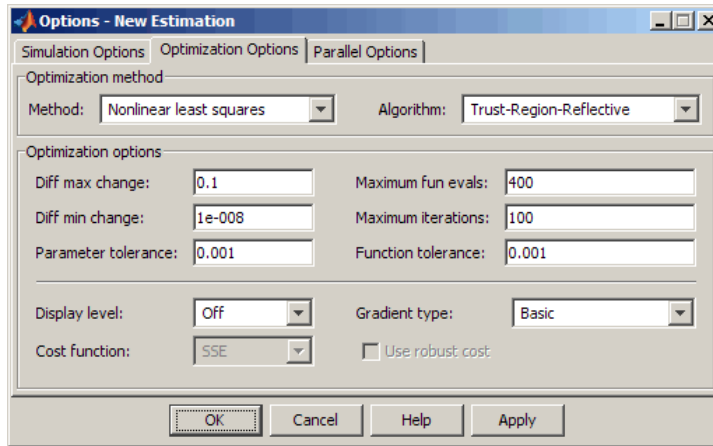
### How to Specify Estimation Options in the GUI

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.



## Simulation Options

### In this section...

“Accessing Simulation Options” on page 2-32

“Selecting Simulation Time” on page 2-33

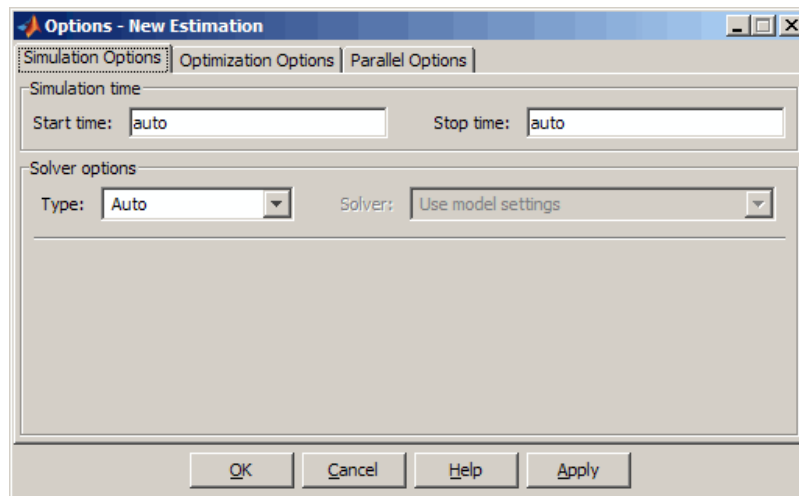
“Selecting Solvers” on page 2-34

## 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 2-33
- “Selecting Solvers” on page 2-34

## Selecting Simulation Time

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

The image shows a software interface for setting simulation time. It has a title bar that says "Simulation time". Below the title bar, there are two input fields. The first is labeled "Start time:" and contains the text "auto". The second is labeled "Stop time:" and also contains the text "auto".

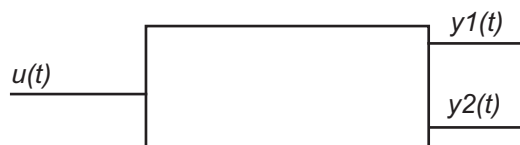
By default, **Start time** and **Stop time** are automatically computed based on the start and stop times specified in the Simulink model.

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  $t_{min}$  to  $t_{max}$ . The values  $t_{min}$  and  $t_{max}$  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:

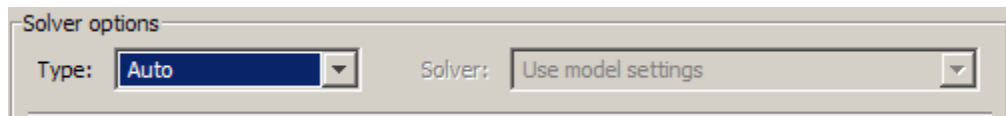
- **Start time** — Typically, the start time in the Simulink model is set to 0. For a nonzero start time, the simulated data corresponding to time points before  $t_1^1$  for  $y1(t)$  and  $t_1^2$  for  $y2(t)$  are discarded.
- **Stop time** — If the stop time  $t_{stop} \geq t_{max}$ , the simulated 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 2-35.
- **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 2-36.

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.

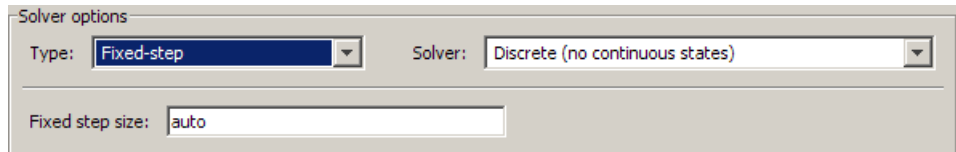
By default, the software automatically chooses the values for these options. To specify your own values, enter them in the appropriate fields. For more information, see “Solver Pane” in the Simulink documentation.

### 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)

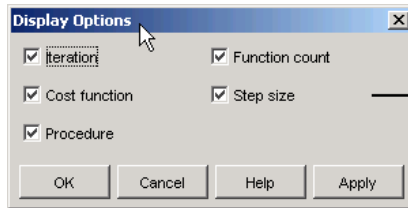
- ode4 (Runge-Kutta)
- ode3 (Bogacki-Shampine)
- ode2 (Heun)
- ode1 (Euler)



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.

# Progress 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.

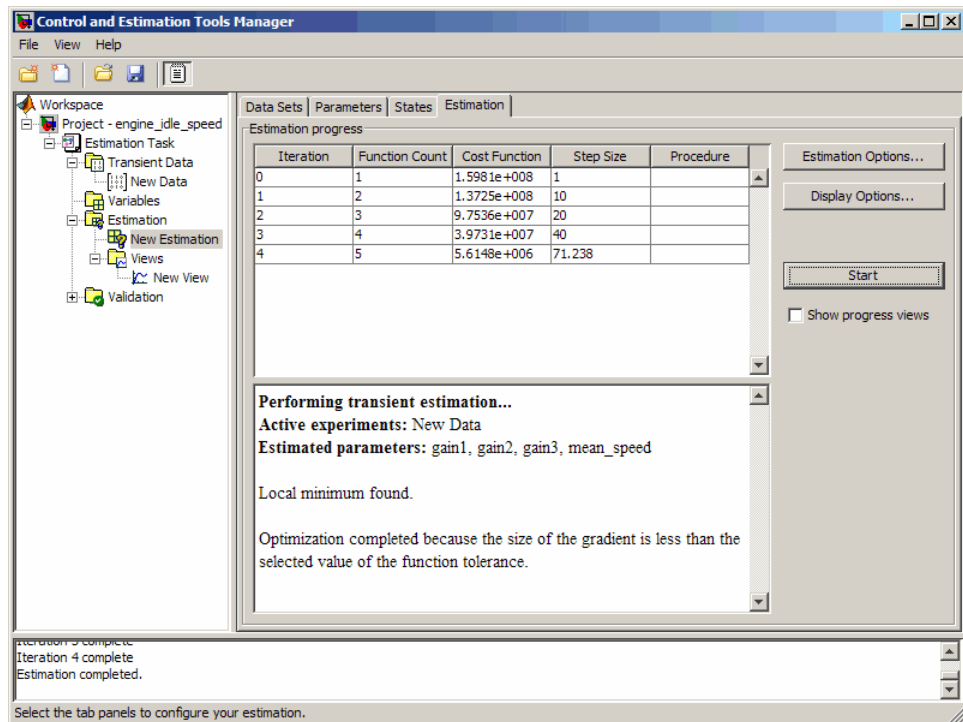
Clearing a check box implies that feature will not appear in the display table as the estimation progresses. To learn more about the display table, see “Iterative Display” in the Optimization Toolbox documentation.

## Estimate Parameters (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 “Specify Estimation Data (GUI)” on page 2-3 and “Specify Parameters to Estimate (GUI)” on page 2-7.

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

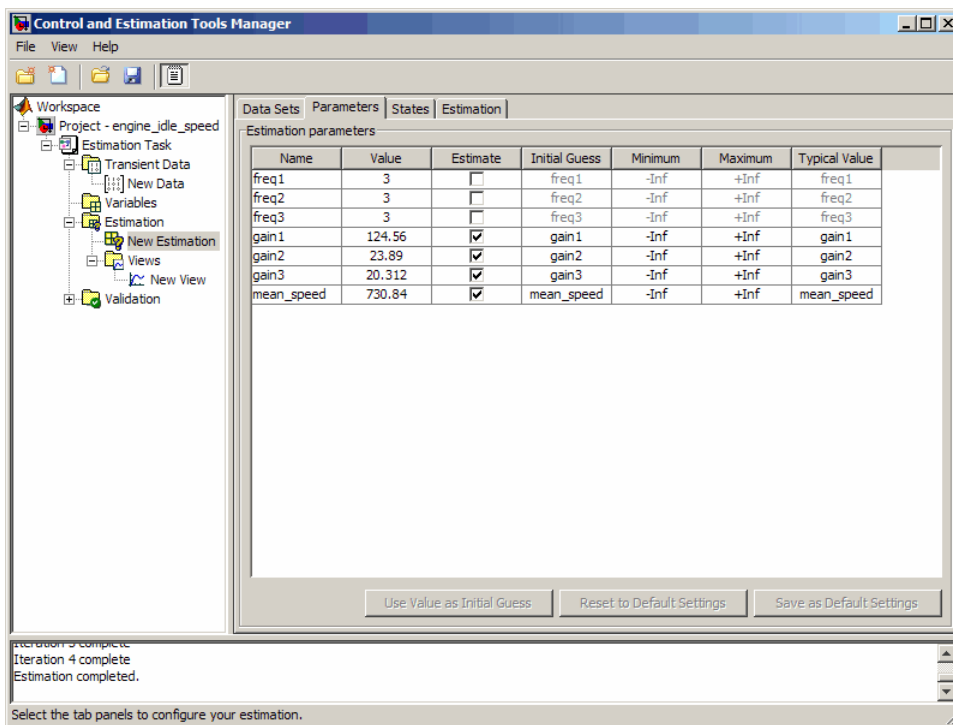
Click **Start** to begin the estimation process. At the end of the iterations, the window should resemble the following:



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.

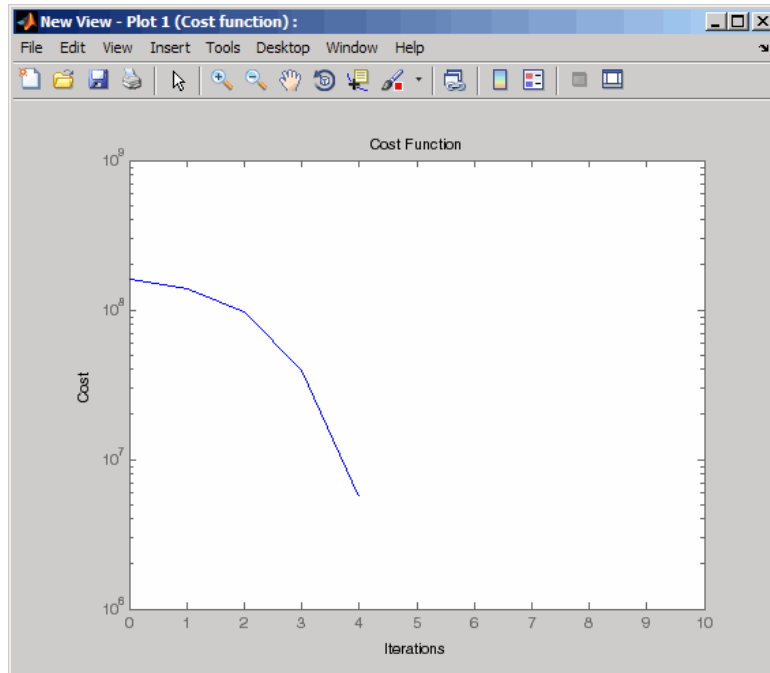
The **Estimation** pane displays each iteration of the optimization methods. To see the final values for the parameters, click the **Parameters** tab.



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.

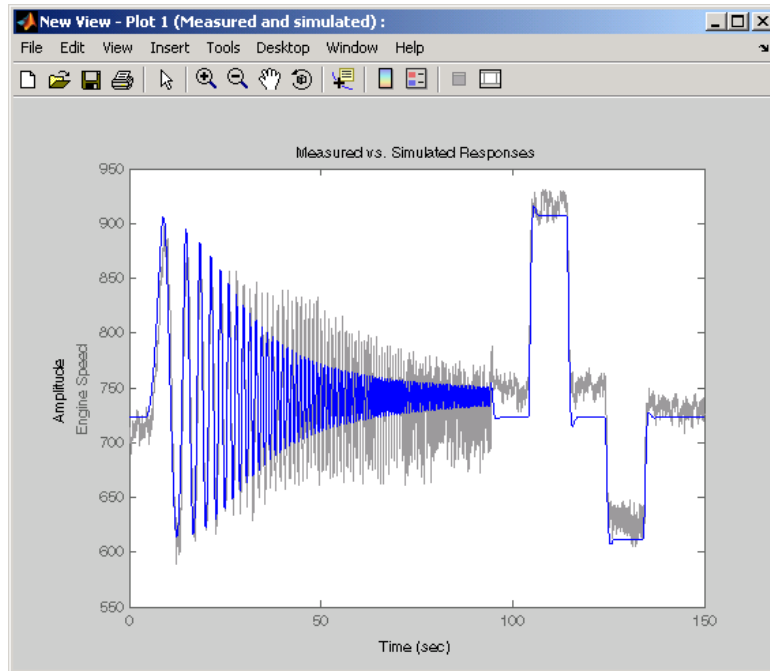
After the estimation process completes, the cost function minimization plot appears as shown in the following figure.





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.



## Validate Parameters (GUI)

### In this section...

“Model Validation” on page 2-43

“Load and Import Validation Data (GUI)” on page 2-44

“Compare Measured and Simulated Responses (GUI)” on page 2-46

“Compare Residuals (GUI)” on page 2-50

### Model Validation

After you complete estimating the parameters, as described in “Estimate Parameters (GUI)” on page 2-39, 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.

### Load and Import Validation Data (GUI)

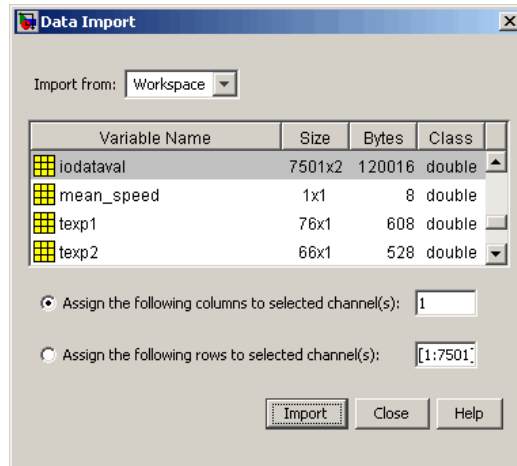
To validate the estimated parameters computed in “Estimate Parameters (GUI)” on page 2-39, you must first import the data into the Control and Estimation Tools Manager GUI.

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 “Import Data (GUI)” on page 1-4 for information on importing data, but the quickest way is to follow these steps:

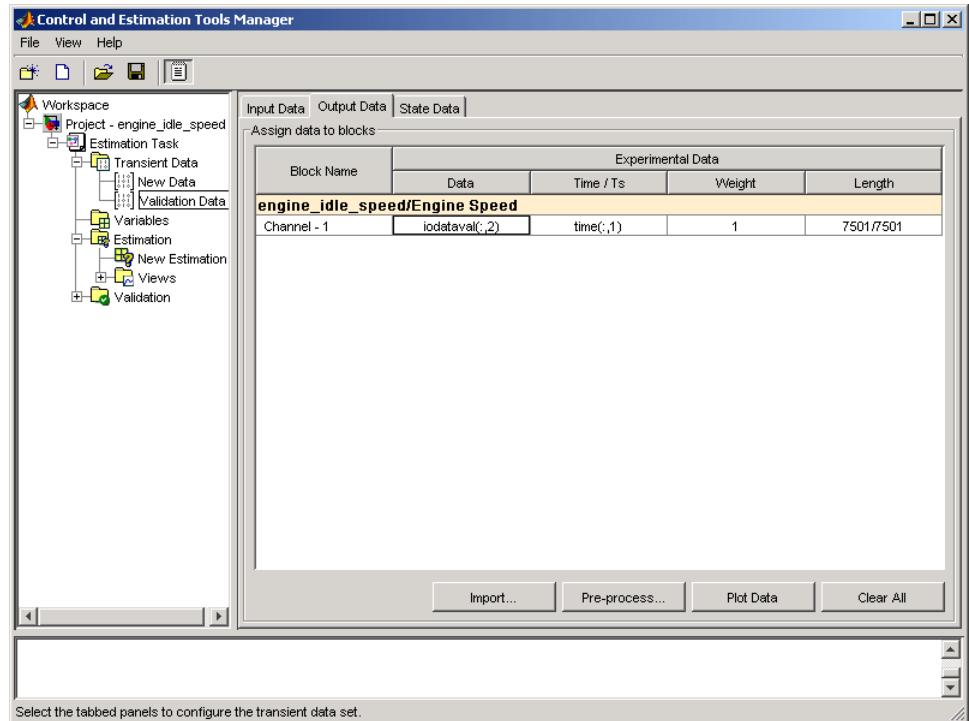
- 1** Right-click the **Transient Data** node in the **Workspace** tree in the Control and Estimation Tools Manager and select **New**.
- 2** Select **New Data (2)** from the **Transient Data** pane.
- 3** Right-click the **New Data (2)** node in the **Workspace** tree and select **Rename**. Change the 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

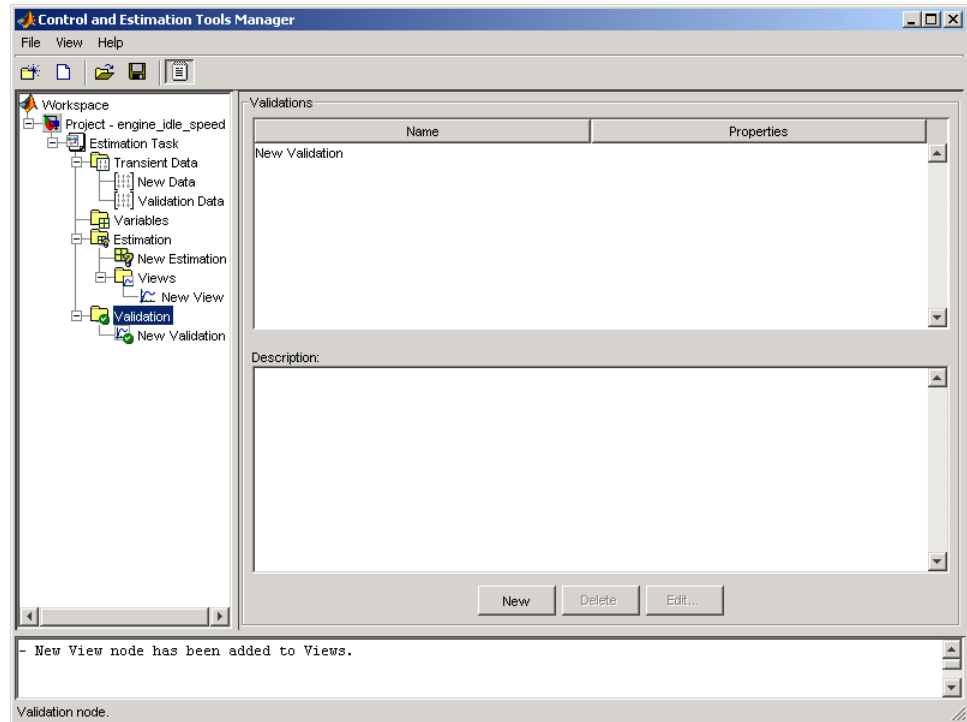
selecting it from the list in the Import Data dialog box and entering 2 in the **Assign columns** field. Click **Import** to import the output data.

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



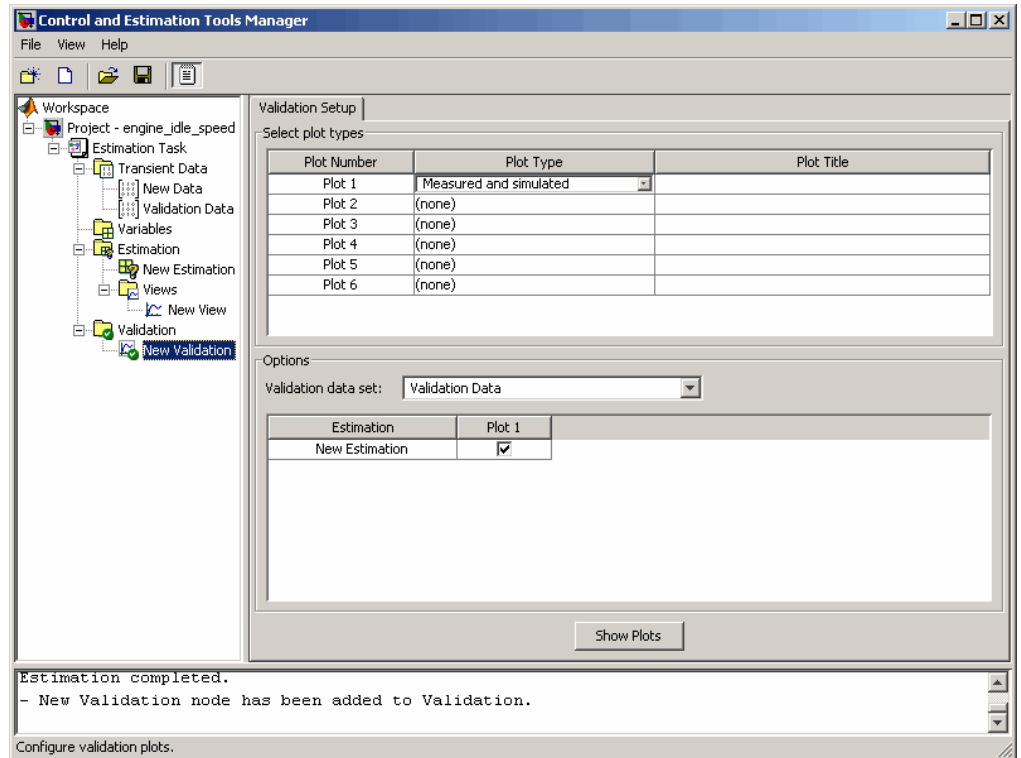
### Compare Measured and Simulated Responses (GUI)

After you import the validation data, as described in “Load and Import Validation Data (GUI)” on page 2-44, right-click the **Validation** node and select **New**. This creates a **New Validation** node in the Control and Estimation Tools Manager.



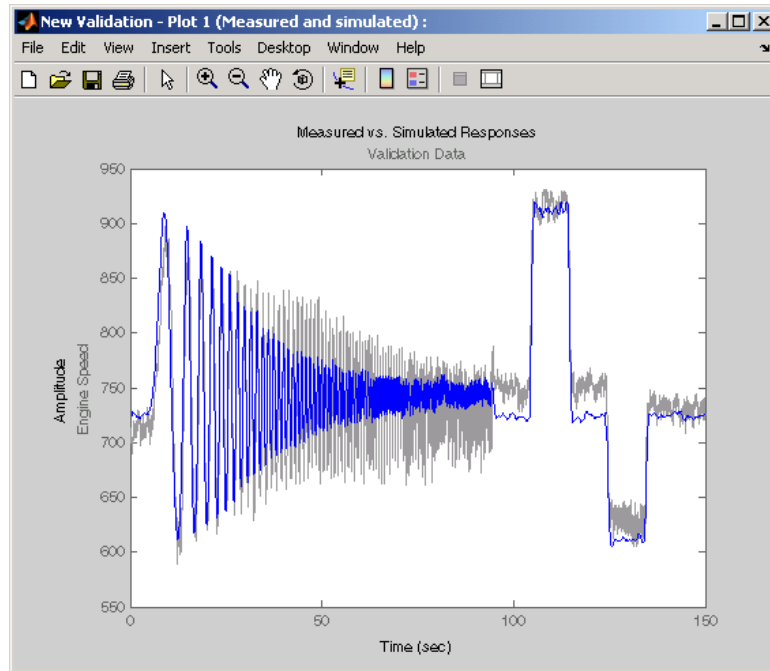
To perform the validation:

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



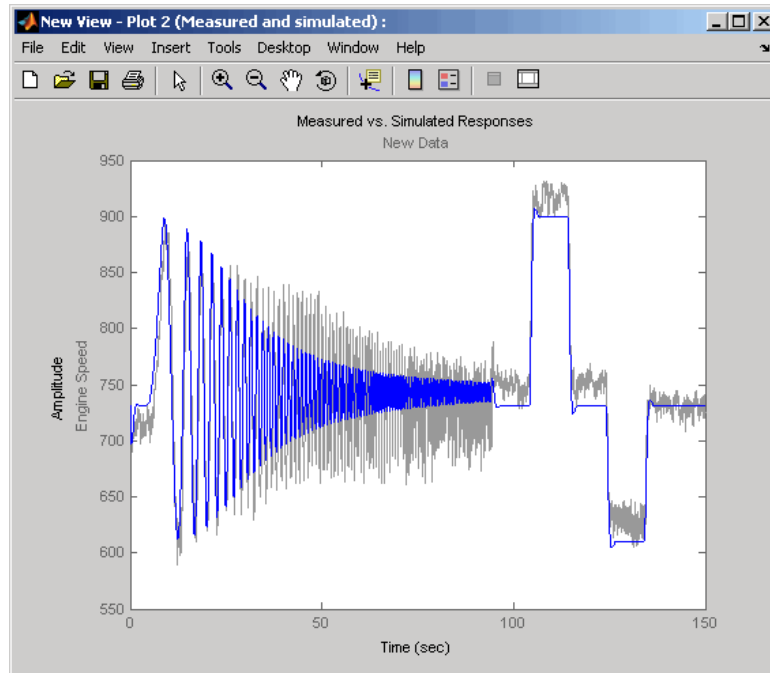
- 2 Click the **Plot Type** cell for Plot 1 and select Measured and simulated from the drop-down menu.
- 3 In the **Options** area, select Validation Data in the **Validation data set** drop-down list.
- 4 Click **Show Plots** to open a plot figure window as shown next.





### 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 “Progress Plots” on page 2-21.



**Measured and Simulated Data Views Plot**

---

**Tip** Examine the residuals compare the difference between the simulated response and measured data, as described in “Compare Residuals (GUI)” on page 2-50.

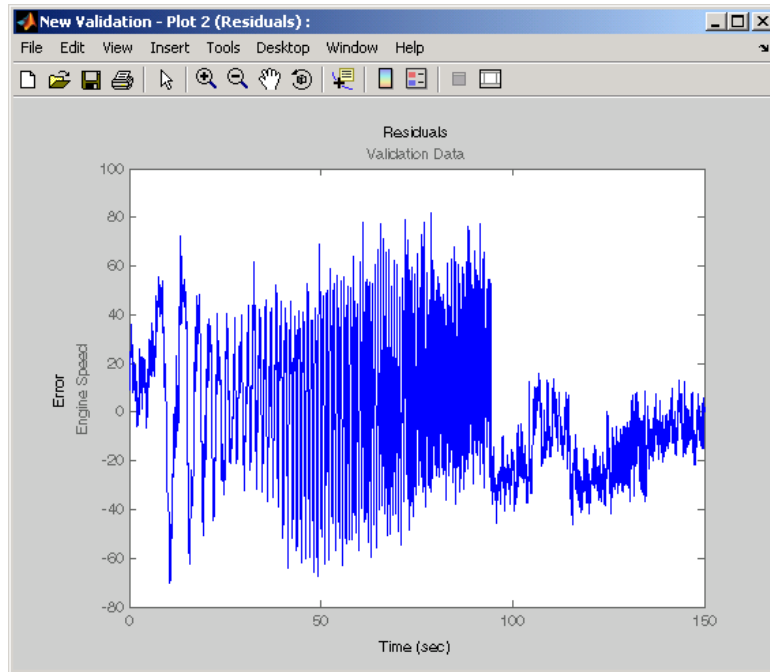
---

### **Compare Residuals (GUI)**

The residuals plot shows the difference between the simulated response and measured data. To indicate a good fit between the simulated output and measured data, the residuals:

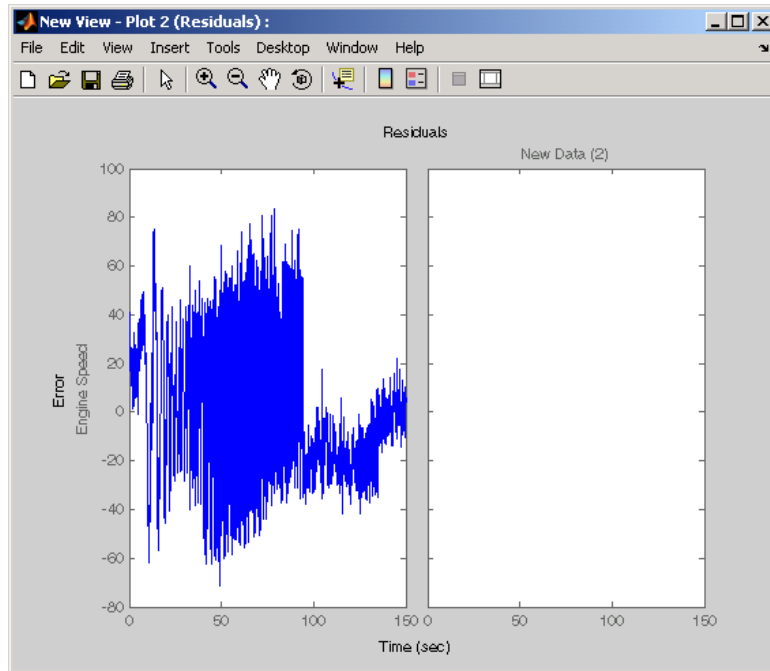
- Should lie within a small percent of the maximum output variation
- Should not display any systematic patterns

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 2-53

“Limitations” on page 2-53

“Setting the Accelerator Mode for Parameter Estimation” on page 2-53

### 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.

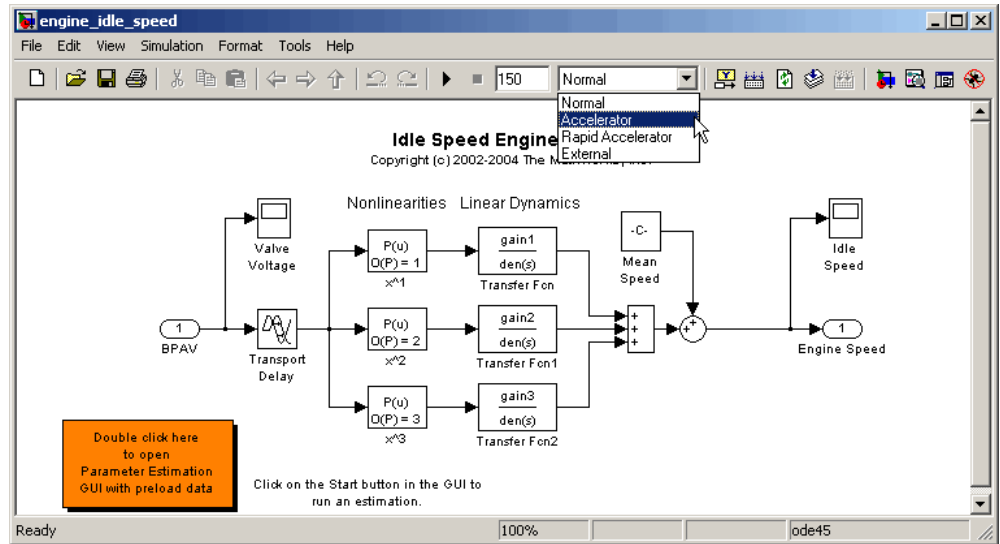
### Limitations

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.

## Speedup Using Parallel Computing

**In this section...**

“When to Use Parallel Computing for Parameter Estimation” on page 2-55

“How Parallel Computing Speeds Up Estimation” on page 2-56

### When to Use Parallel Computing for Parameter Estimation

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 Estimation” on page 2-56.

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

- “How to Use Parallel Computing (GUI)” on page 2-59
- “How to Use Parallel Computing (Code)” on page 2-132

## How Parallel Computing Speeds Up 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 2-56
- “Parallel Computing with the Pattern search Method” on page 2-57

### 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:

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



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.

## How to Use Parallel Computing (GUI)

### In this section...

“Configure Your System for Parallel Computing” on page 2-59

“Specify Model Dependencies” on page 2-60

“Estimate Model Parameters” on page 2-61

“Troubleshooting” on page 2-65

### Configure 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 2-59
- “Configuring Parallel Computing on Multiprocessor Networks” on page 2-59

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 “Define Cluster Profiles” 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 “Apply Cluster Profiles 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.

### **Specify 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 “Estimate Model Parameters” on page 2-61 and “How to Use Parallel Computing (Code)” on page 2-132.
- 2 Add the file dependencies, as described in “Configuring Parallel Computing on Multiprocessor Networks” on page 2-59.

---

**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.

---

## **Estimate Model Parameters**

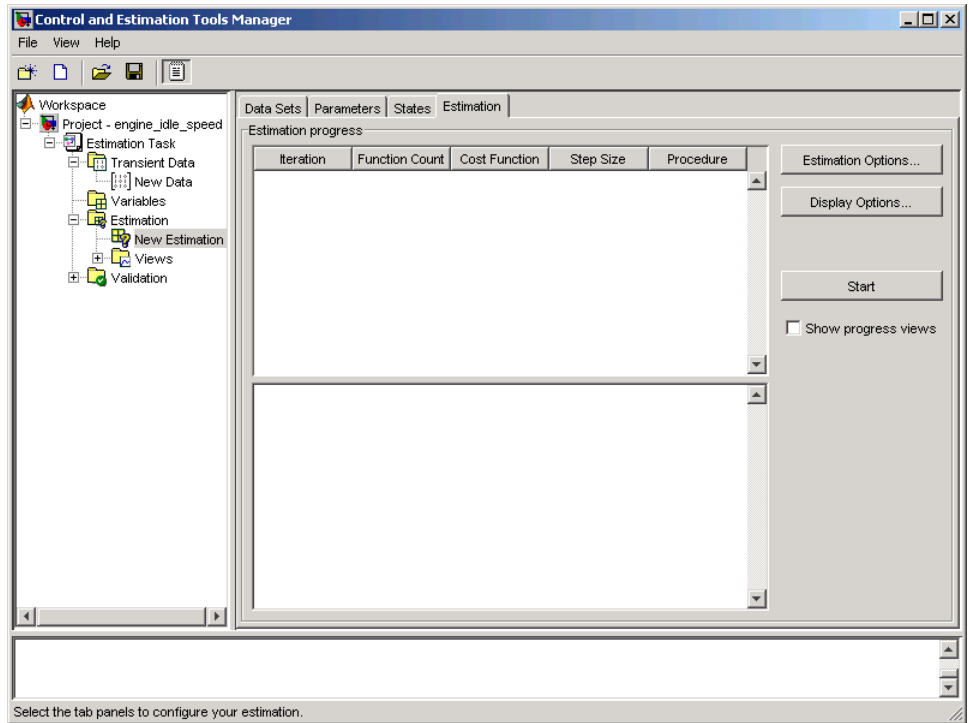
After you configure your system for parallel computing, as described in “Configure Your System for Parallel Computing” on page 2-59, you can use the GUI to estimate the model parameters.

---

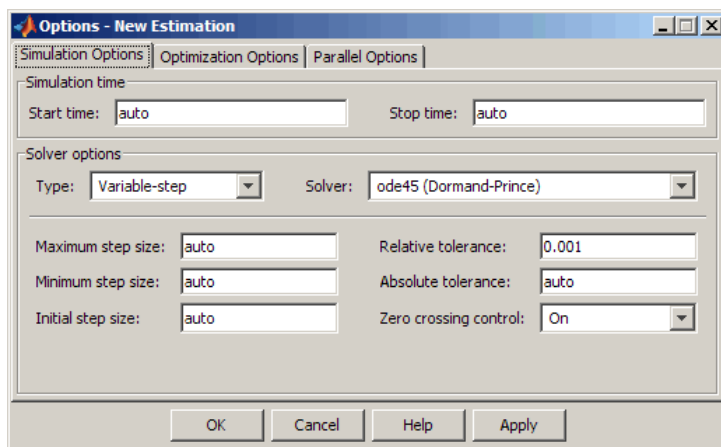
**Tip** If you want to use functions to estimate parameters using parallel computing, see “How to Use Parallel Computing (Code)” on page 2-132.

---

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



This action opens the Options - New Estimation dialog box.



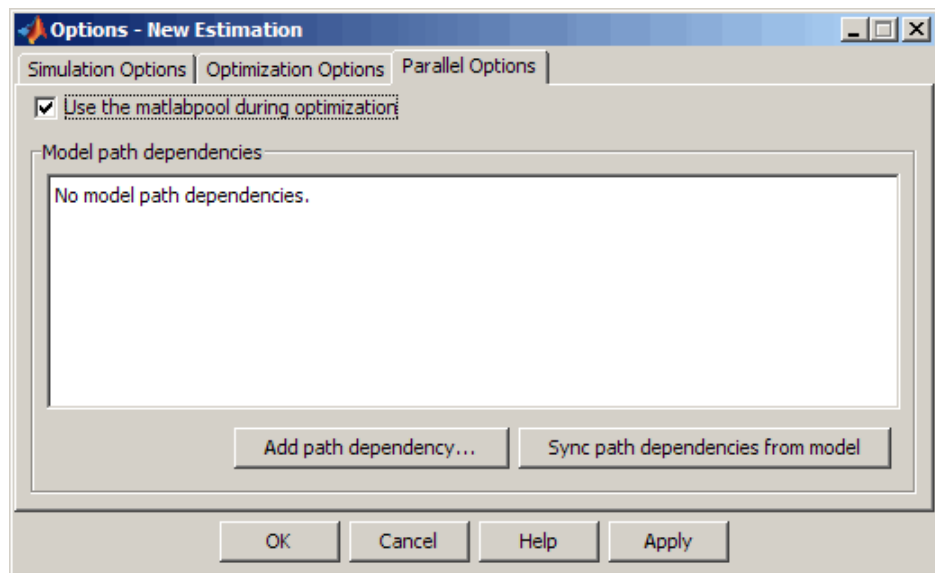
- 4 In the **Parallel Options** tab, select the **Use the matlabpool during optimization** option.

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

---

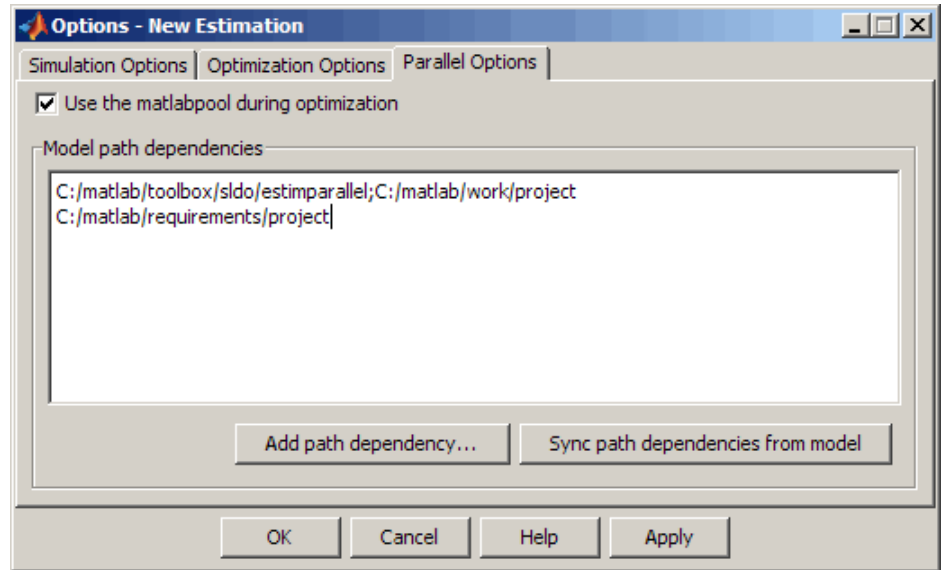
**Note** As described in “Specify Model Dependencies” on page 2-60, the automatic path dependencies check may not detect all the path dependencies in your model.

---



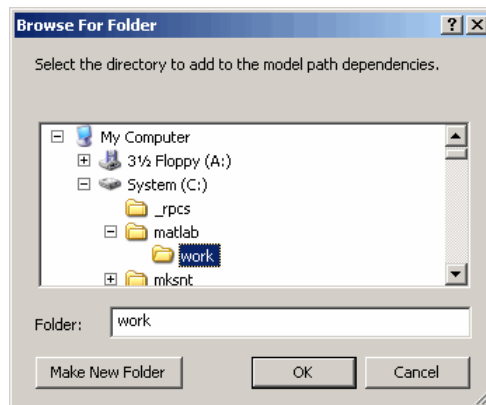
- 5 (Optional) Add the path dependencies that the automatic check does not detect.
  - a Specify the paths in the **Model path dependencies** list box.

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 2-65.

## Troubleshooting

- “Why are the estimation results with and without using parallel computing different?” on page 2-65
- “Why do I not see the estimation speedup I expected using parallel computing?” on page 2-66
- “Why does the estimation using parallel computing not make any progress?” on page 2-67
- “Why do I receive an error "Cannot save model tpe5468c55\_910c\_4275\_94ef\_305e2eeeeeef4"?” on page 2-67
- “Why does the estimation using parallel computing not stop when I click **Stop**?” on page 2-67

### 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 2-57.

### **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 2-57.

### **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 “Specify Model Dependencies” on page 2-60.

### **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 2-29.

### **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.

## Estimate Initial States

In this section...
“How to Estimate Initial States (GUI)” on page 2-68
“Estimating Initial Conditions for Blocks with External Initial Conditions” on page 2-71
“Estimate Initial States of a Mass-Spring-Damper System (GUI)” on page 2-71

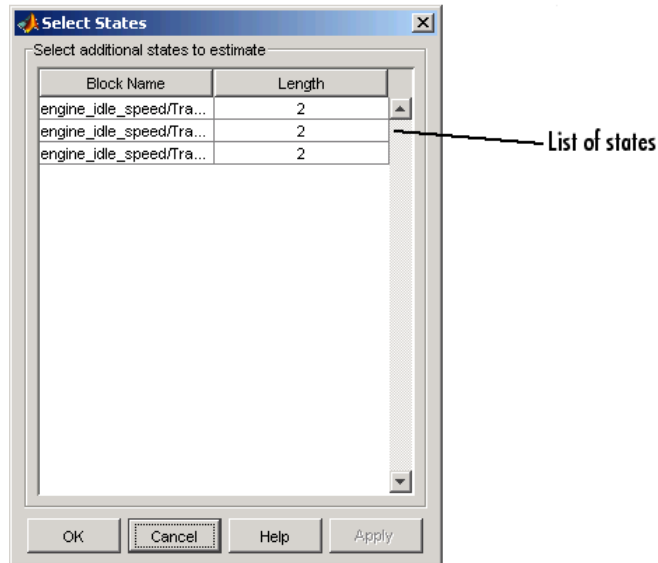
### How to Estimate Initial States (GUI)

You can estimate initial states of Simulink models containing any blocks, including Simscape™, SimMechanics™, SimPowerSystems™ and SimHydraulics® blocks. Typically, you estimate only those states whose values cannot be measured.

Before you estimate initial states, you must have already imported the estimation data and specified the parameters to estimate. See “Specify Estimation Data (GUI)” on page 2-3 and “Specify Parameters to Estimate (GUI)” on page 2-7.

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.



The dialog box shows all states in the model, including:

- States in Simscape, SimMechanics, SimPowerSystems and SimHydraulics blocks.

Block Name	Length
slido_rc_circuit.C1.vc_0	1
slido_rc_circuit.V1_5V_DC.v_1	1

- States with unique names, such as Velocity in an integrator block.

Block Name	Length
msd_system_1/Position	1
Velocity	1

- States in model references.

Block Name	Length
spe_modref_1/Referenced Model spe_modref_model_name/Position	1

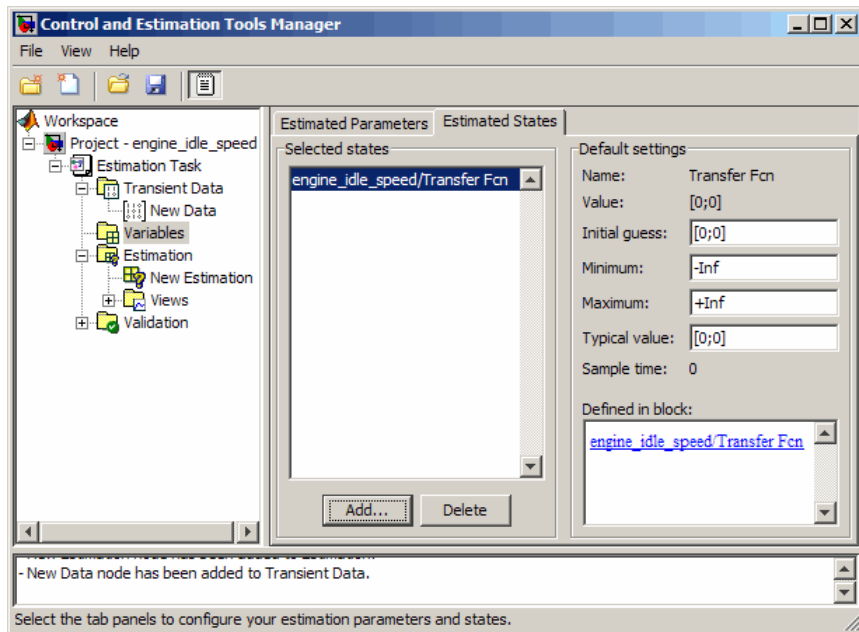
- States in blocks with vector inputs. For example, an Integrator block with vector inputs of size 3 and three named states position, velocity and acceleration:

Block Name	Length
acceleration	1
position	1
velocity	1

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

- To select adjacent states, hold the **Shift** key and click the states.
- To select nonadjacent states, hold the **Ctrl** key and click the states.

The states selected for estimation now appear in the **Estimated States** tab.



---

**Tip**

- The **Defined in block** area of the **Estimated States** tab shows the path to the block associated with the state. Click the link to highlight the block in the model.
  - For long state names, move the mouse cursor over the selected state name in the **Selected states** area or **Name** in the **Default settings** area to display the full name in a tooltip.
- 

For an example of estimating initial states using the GUI, see “Estimate Initial States of a Mass-Spring-Damper System (GUI)” on page 2-71.

## **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, you cannot estimate the initial conditions (ICs) of the integrator using Simulink Design Optimization software. Estimation is not possible 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.

## **Estimate Initial States of a Mass-Spring-Damper System (GUI)**

- “Loading the Example” on page 2-72
- “Model Parameters” on page 2-73
- “Setting Up the Estimation Project” on page 2-74
- “Importing Transient Data and Selecting Parameters for Estimation” on page 2-75

- “Selecting Parameters and Initial Conditions for Estimation” on page 2-77
- “Creating the Estimation Task” on page 2-78
- “Running the Estimation and Viewing Results” on page 2-80

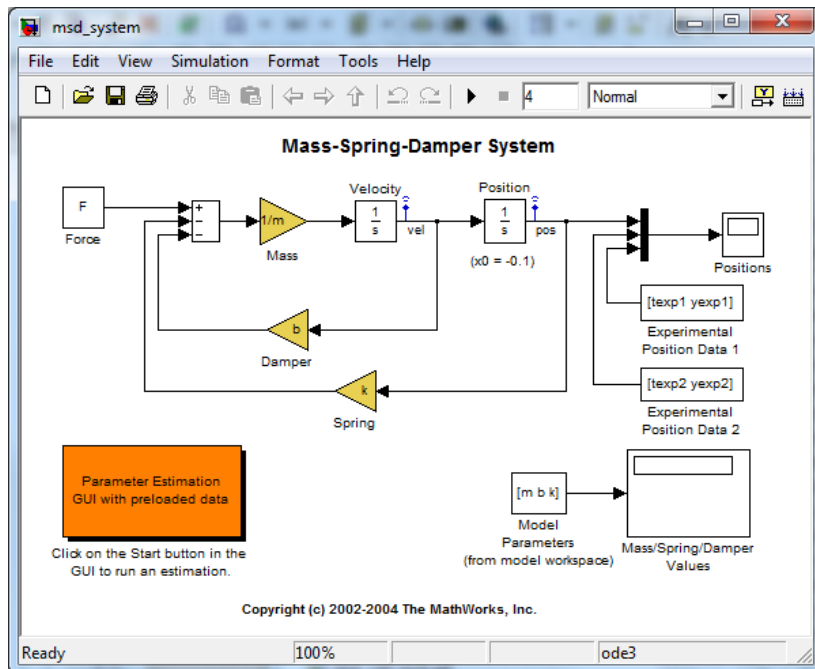
### Loading the Example

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**

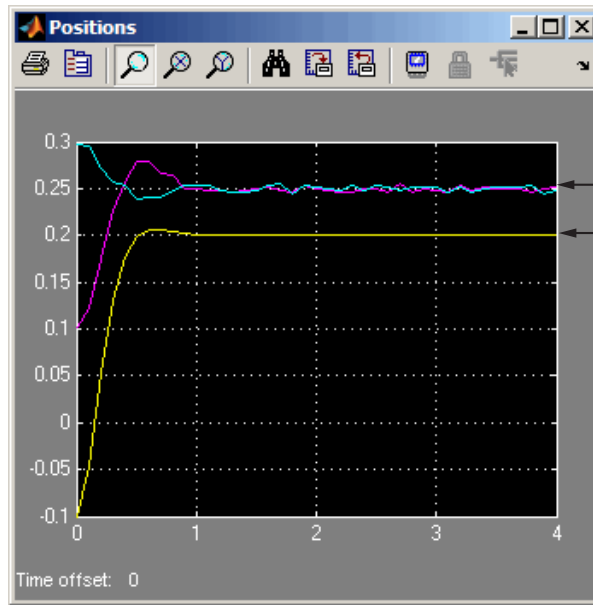
The output of the Simulink model, `msd_system`, is the position of the mass in a mass-spring-damper system. The model is subject to a constant force  $F$ , and an initial condition,  $x_0$ , for the mass displacement.  $x_0$  is the initial condition of the integrator block, Position.

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

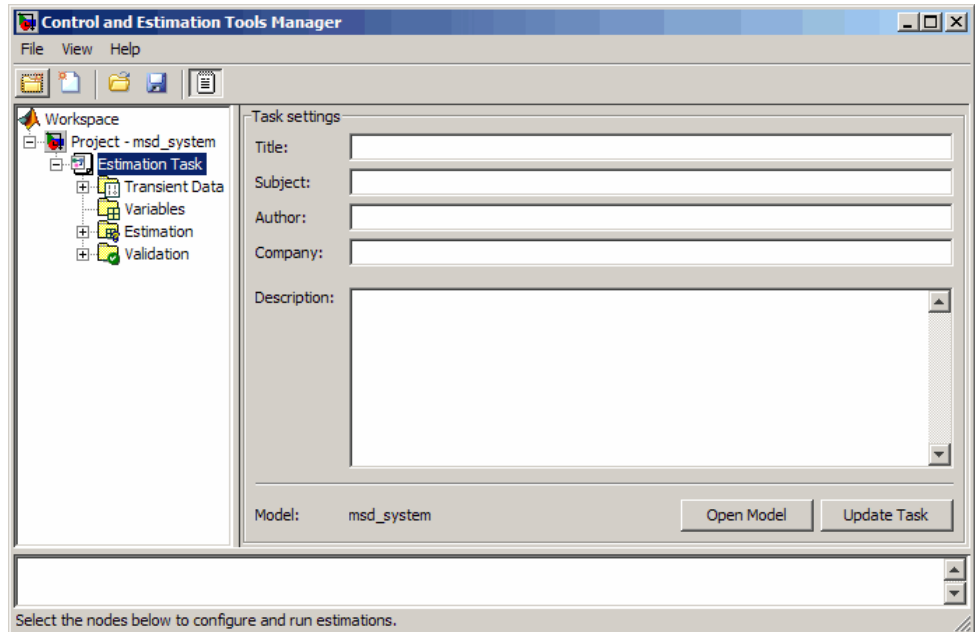


Magenta and cyan lines are experimental data sets with different initial conditions.

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

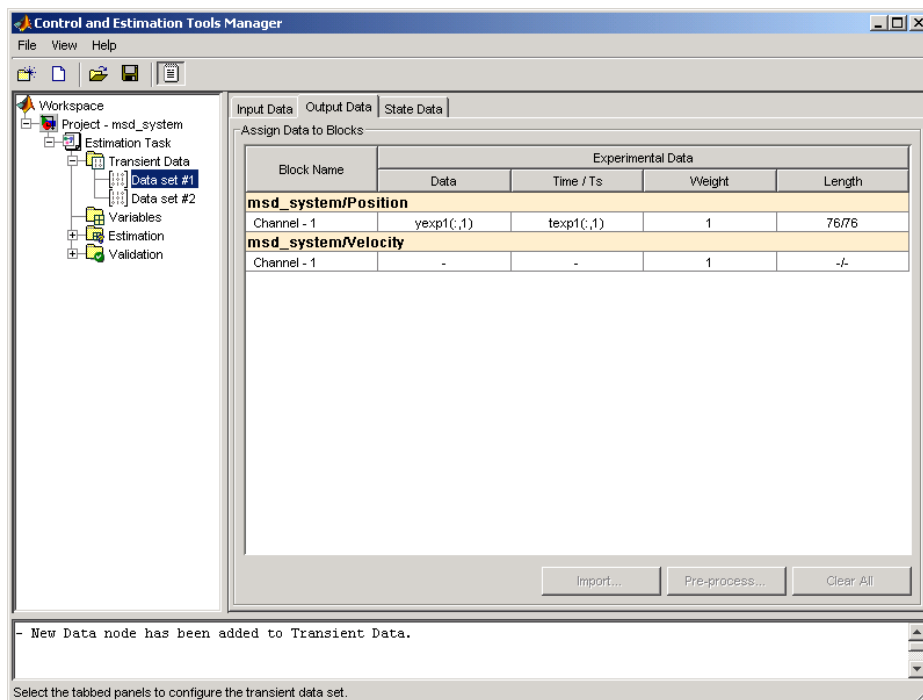
To import data for initial state estimation:

- 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** tab of the **New Data** node, select the **Data** column of `msd_system/Position`, 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 **msd\_system/Position**. 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.



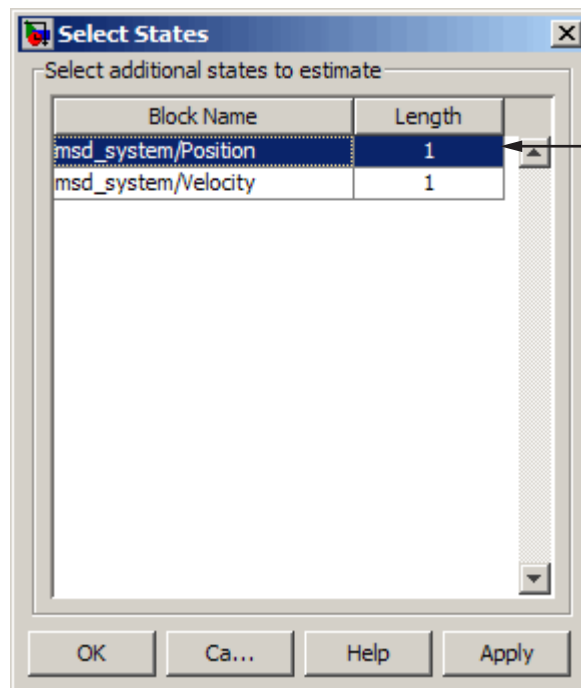
For more information on importing data, see “Import Data (GUI)” on page 1-4.

## 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**.

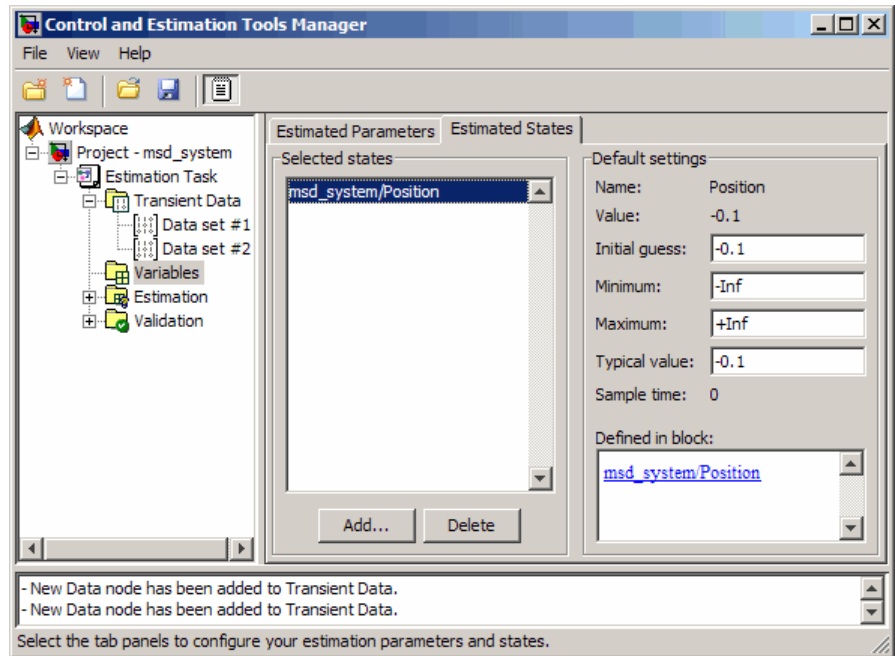
The Select States dialog box opens, which displays all the available states of the `msd_system` model. Select `msd_system/Position`, and click **OK**.



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.



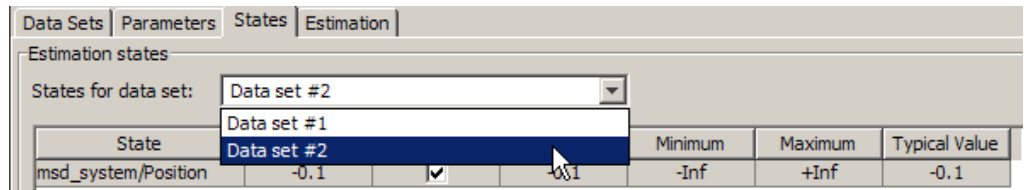
For more information on selecting parameters to estimate, see “Specify Parameters to Estimate (GUI)” on page 2-7.

### Creating the Estimation Task

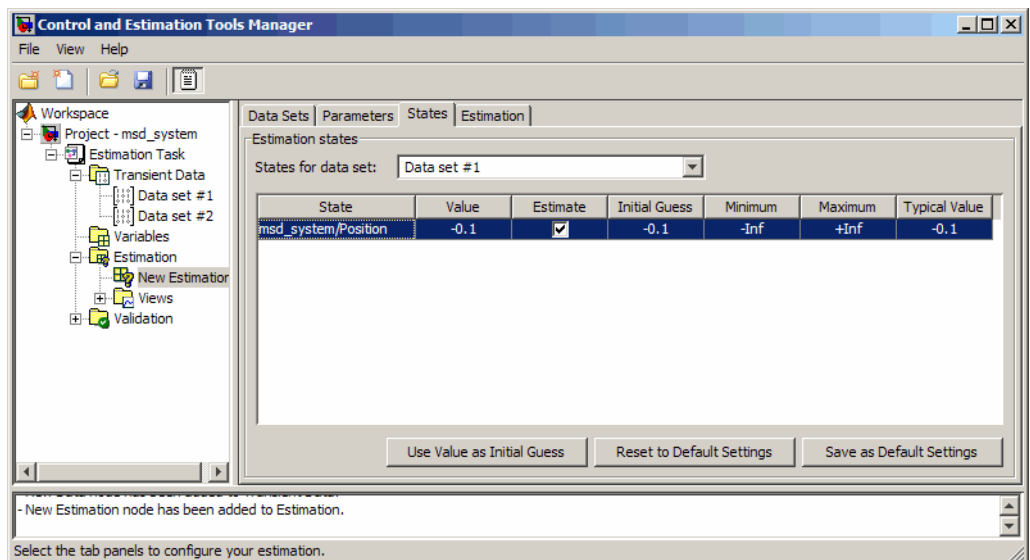
To create an estimation task in the Control and Estimation Tools Manager GUI, select the **Estimation** node in the **Workspace** tree, and click **New**. This action creates a **New Estimation** node.

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

- Data Set #1 and Data Set #2 in the **Data Sets** pane.
- **Estimate** for b, k, and m in the **Parameters** pane.
- **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.



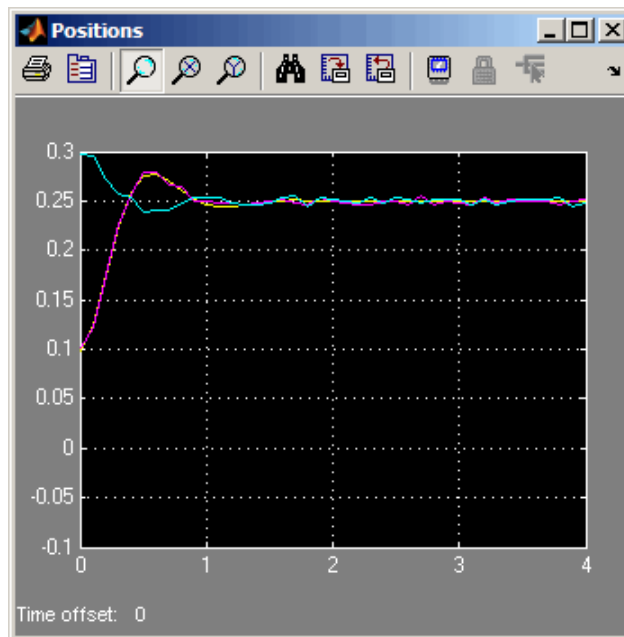
Although the initial positions for the two data sets differ, specify the initial state guesses for both data sets as  $-0.1$ . The Control and Estimation Tools Manager GUI now resembles the following figure.



### 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.



Data Sets   Parameters   States   Estimation						
Estimation states						
States for data set: <span style="border: 1px solid black; padding: 2px;">Data set #1</span>						
State	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Value
msd_system/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.

Data Sets   Parameters   States   Estimation						
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.

## Estimation Projects

In this section...
“Structure of an Estimation Project” on page 2-82
“Managing Multiple Projects and Tasks” on page 2-83
“Adding, Deleting and Renaming an Estimation Project” on page 2-84
“Saving Control and Estimation Tools Manager Projects” on page 2-85
“Loading Control and Estimation Tools Manager Projects” on page 2-86

### 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

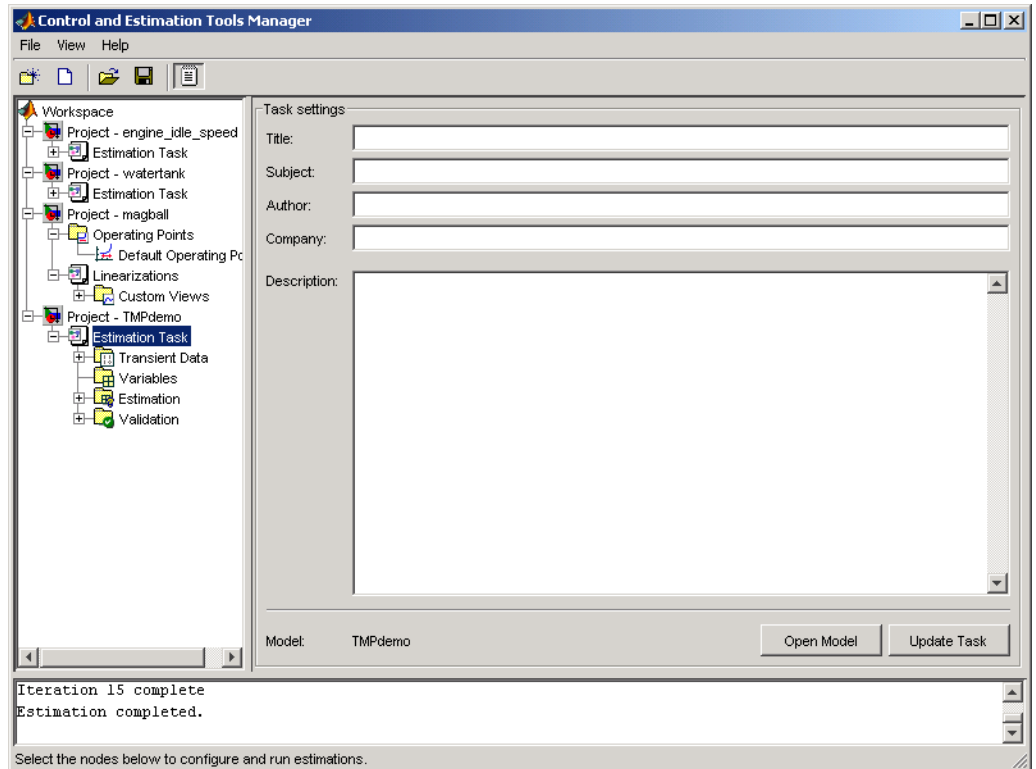
The default project name is the same as the Simulink model name. The project name is shown in the **Workspace** tree of the Control and Estimation Tools Manager.

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 2-85
- “Loading Control and Estimation Tools Manager Projects” on page 2-86

### **Adding, Deleting and Renaming an Estimation Project**

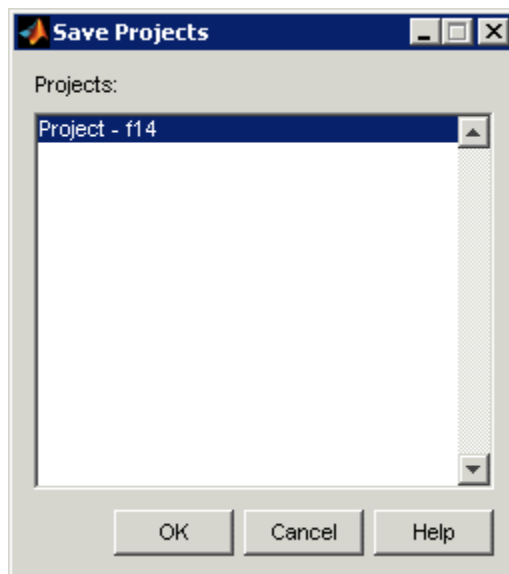
To add, delete, or rename the project or task:

- 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.

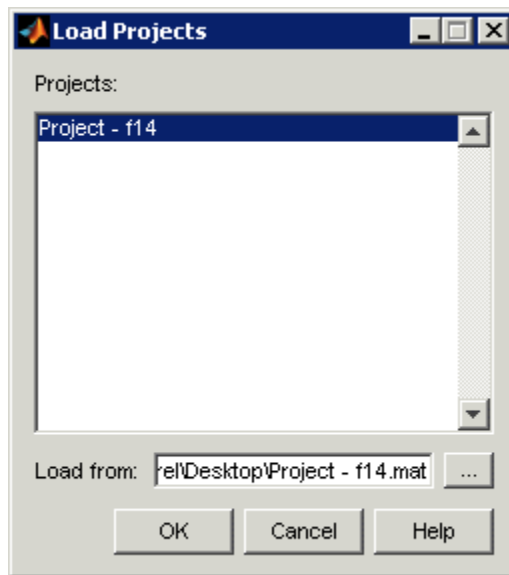


To save multiple projects within one file:

- 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.

## Estimate Parameters (Code)

### In this section...

- “How to Estimate Parameters (Code)” on page 2-87
- “F14 Parameters and Initial State Estimation (Code)” on page 2-89
- “RC Circuit Parameters and Initial State Estimation (Code)” on page 2-100
- “Parameter Estimation Objects” on page 2-105
- “Creating Transient Experiment Data Objects” on page 2-106
- “Creating Estimation Objects” on page 2-109
- “Parameter Objects” on page 2-117
- “State Objects” on page 2-120
- “Transient Data Objects” on page 2-123
- “State Data Objects” on page 2-129
- “How to Use Parallel Computing (Code)” on page 2-132

## How to Estimate Parameters (Code)

In addition to the Control and Estimation Tools Manager GUI, you can also use Simulink Design Optimization commands to perform parameter and state estimation. You can perform the same tasks as the Control and Estimation Tools Manager using these commands but have the advantages of command-line execution.

When you perform a state or parameter estimation using the GUI, the software creates MATLAB objects for *all* the states and parameters in 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.

The command-line approach is also 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 in MATLAB, see *Object-Oriented Programming*.

To perform estimation using the command-line interface:

- 1 Open a Simulink model.

---

**Note** The Simulink model must contain a fixed signal input block, an Inport block, Outport block, or logged signals to enable assigning data to the signals. For more information, see “Model Requirements for Importing Data” on page 1-2.

---

- 2 Define experiments containing empirical data sets, operating conditions and initial states of your model.
- 3 Create an estimation object.
- 4 Specify parameters to estimate.
- 5 (Optional) Specify states to estimate.
- 6 Perform the estimation.
- 7 Review the results and refine the parameters iteratively.
- 8 Validate estimation results.

---

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

---

For more information, see:

- “F14 Parameters and Initial State Estimation (Code)” on page 2-89
- “RC Circuit Parameters and Initial State Estimation (Code)” on page 2-100
- “Parameter Estimation Objects” on page 2-105



## F14 Parameters and Initial State Estimation (Code)

- “Objectives” on page 2-89
- “Data Description” on page 2-89
- “Opening the F14 Aircraft Model” on page 2-90
- “Baseline Simulation” on page 2-91
- “Specifying Experimental Data” on page 2-93
- “Specifying Parameters and States to Estimate” on page 2-94
- “Running the Estimation” on page 2-96
- “Viewing Estimated Parameter and State Values” on page 2-97
- “Rerunning the Estimation Using Estimated Parameter Values” on page 2-99
- “Saving and Loading the Estimation Results” on page 2-99

### Objectives

This example shows how to estimate model parameters and states using Simulink Design Optimization objects and commands. To learn more about the parameter estimation objects and their properties and methods, see “Parameter Estimation Objects” on page 2-105.

In this example, you estimate the following parameters and state of a F14 aircraft.

Parameters	State
<ul style="list-style-type: none"> <li>• Actuator time constant <math>T_a</math></li> <li>• Vertical velocity <math>Z_d</math></li> <li>• Pitch rate gains <math>M_d</math></li> </ul>	f14/Actuator Model

### Data Description

The `f14_estim.mat` file contains the following variables:

- `time` — Time vector.

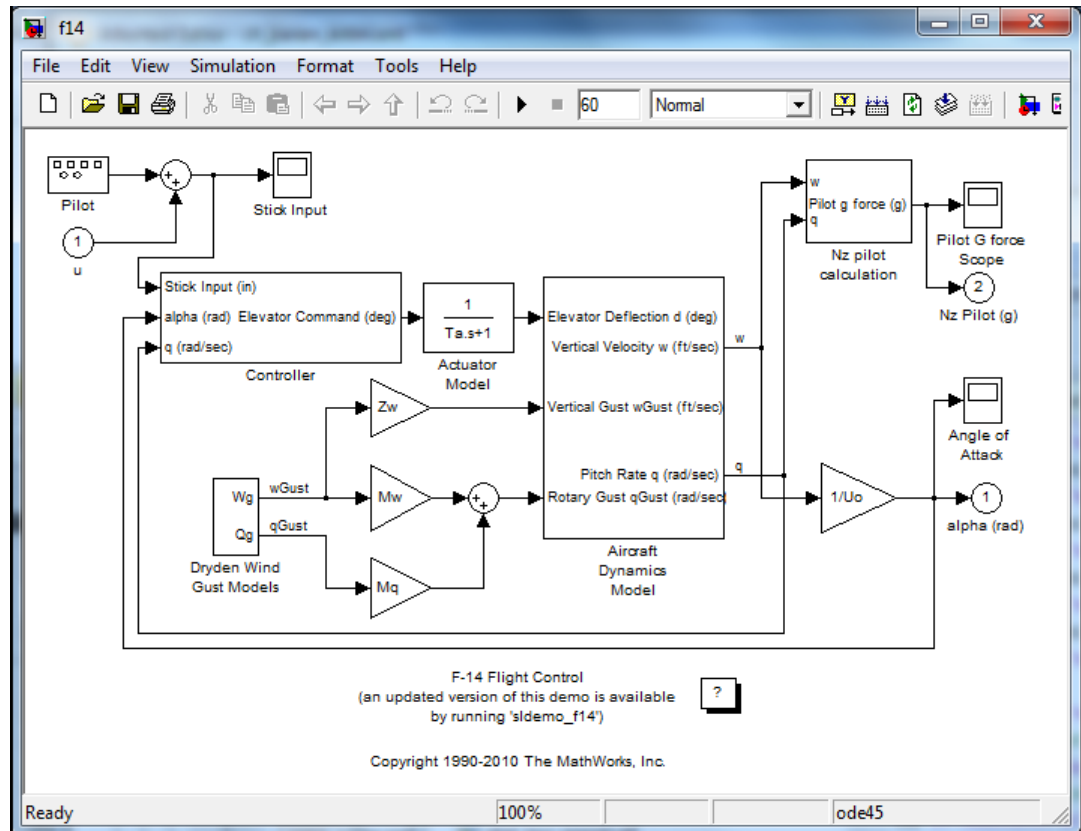
- `iodata` — Single-input multi-output data measured for:
  - A square wave input applied to the stick.
  - Pilot G force output
  - Angle of attack output

### **Opening the F14 Aircraft Model**

Open the `f14` Simulink model by typing the model name at the MATLAB prompt:

```
f14
```

The model opens, as shown in the following figure.



## F14 Fighter Jet Model

The command also loads the estimation data `iodata` into the MATLAB workspace.

## Baseline Simulation

Before performing an estimation, run a baseline simulation to compare the simulated response with initial parameter values and the measured data.

```
%% Load experimental data.
load f14_estim
```

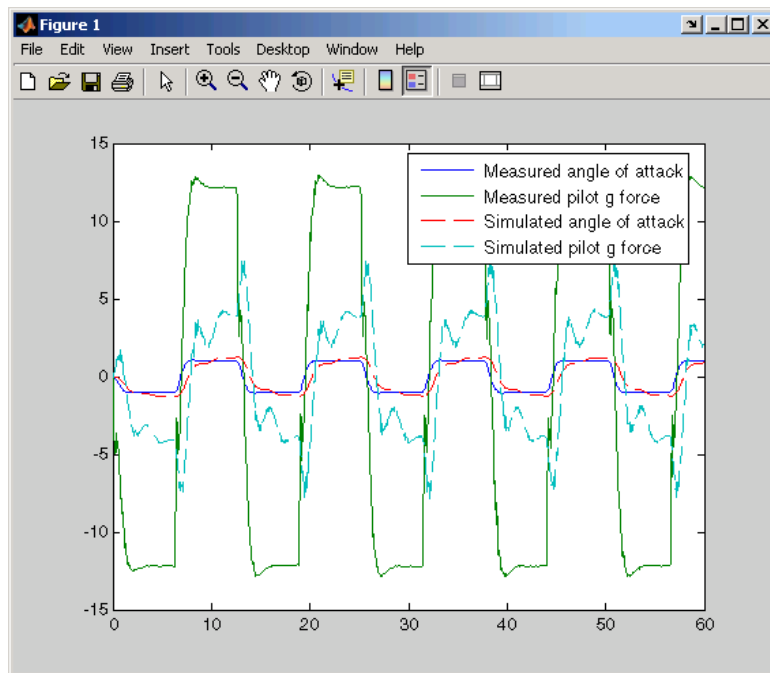
```
%% Initialize unknown parameters.
```

```
% 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');
```

See the `sim` reference page for information on how to simulate models from the MATLAB prompt.

The measured and simulated outputs are a poor match, as shown in the following figure.



Next, you estimate values of the model parameters  $T_a$ ,  $Z_d$ , and  $M_d$  to obtain a better match between the simulated and measured responses.

## Specifying Experimental Data

To specify experimental data for parameter estimation:

- 1 Create a `TransientExperiment` object using the constructor syntax:

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

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

- 2 Assign the experimental I/O data in `iodata` to the `Transient Experiment` object.

```
%% Assign input data.
```

```
set(exp_data.InputData(1), 'Data', iodata(:,1), 'Time', time);
```

```
% Assign output data.
```

```
set(exp_data.OutputData(1), 'Data', iodata(:,2), 'Time', ...
```

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

---

**Note** For multi-input, multi-output models:

- Assign separate data object to each input and output port. The data object can be a vector or matrix that corresponds to that channel.
  - Use multiple inport or outport blocks to represent multiple channels.
- 

### Specifying Parameters and States to Estimate

To specify parameters and states to estimate:

- 1 Create an Estimation object.

```
est = ParameterEstimator.Estimation('f14');
```

This command creates objects for all parameters and states in the model. To view the properties of `est`, type:

```
get(est)
```

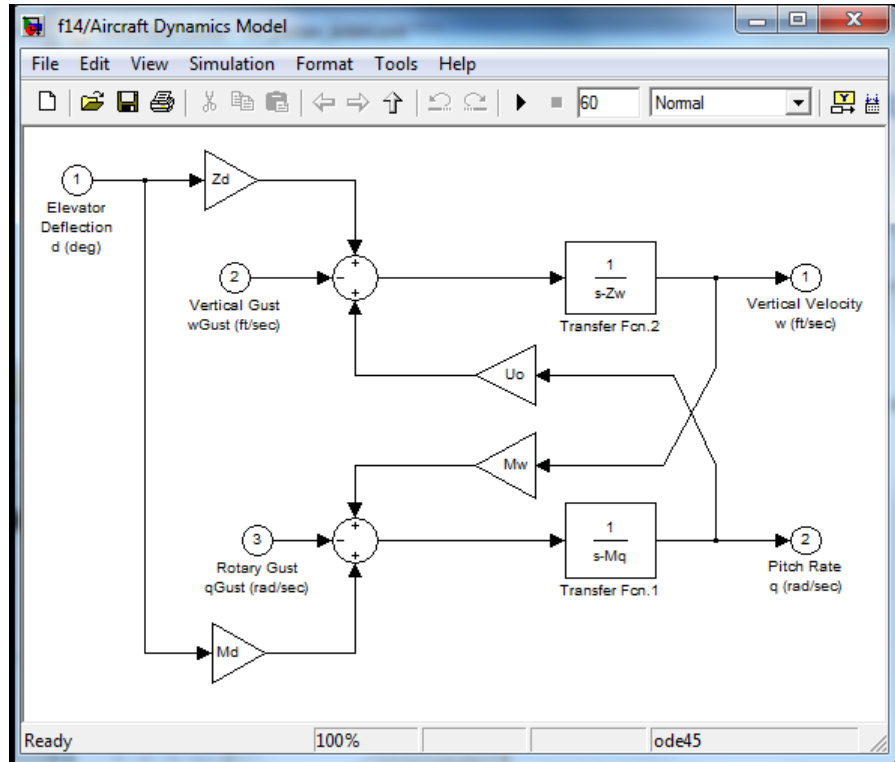
To view the parameter objects, type:

```
est.Parameters
```

To view the state objects, type:

```
est.States
```

- 2 Specify the model parameters  $T_a$ ,  $Z_d$  and  $M_d$  to estimate. These parameters are located in the F14 aircraft dynamics subsystem.



```

% Select Md.
est.Parameters(5).Estimated = true;
% Specify minimum and maximum values.
set(est.Parameters(5), 'Minimum', -10, 'Maximum', 0);

% Select Ta.
est.Parameters(9).Estimated = true;
% Specify minimum and maximum values.
set(est.Parameters(9), 'Minimum', 0.01, 'Maximum', 1);

% Select Zd.
est.Parameters(16).Estimated = true;
% Specify minimum and maximum values.
set(est.Parameters(16), 'Minimum', -100, 'Maximum', 0);

```

**3** Specify the f14//Actuator Model state to estimate.

```
% Select the state.
est.States(1).Estimated = true;
% Specify the minimum value of the state.
set(est.States(1), 'Minimum', -1, 'Maximum', +1);
```

### Running the Estimation

To run the estimation:

**1** Specify the experimental data for the estimation:

```
est.Experiments = exp_data;
```

**2** Estimate the model parameters and initial state:

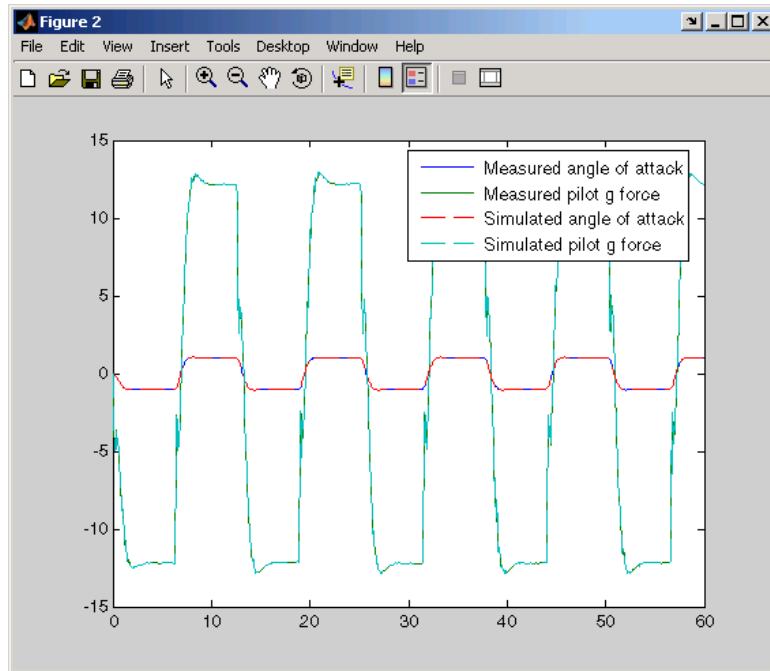
```
% Display the estimation iterations.
est.OptimOptions.Display = 'iter';
% Estimate parameters and state.
est.estimate
```

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

```
%% Plot measured data and final simulation responses.
[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');
```

The measured and simulated outputs now closely match, as shown in the following figure.





## Viewing Estimated Parameter and State Values

- 1 To view the estimated parameter values for comparison with the default values, extract the values from the Estimation object. For example:

```
% Extract Md.
MdValue = est.Parameters(5).Value

% Extract Ta.
TaValue = est.Parameters(9).Value

% Extract Zd.
ZdValue = est.Parameters(16).Value
```

These command returns the following results:

```
MdValue =
```

```
-6.8869
```

```
TaValue =
```

```
0.0501
```

```
ZdValue =
```

```
-64.0356
```

**2** To view the estimated initial state value, type:

```
% Extract initial state value.  
stateValue = est.State(1)
```

This command returns the following result:

```
(1) State data for f14/Actuator Model:  
    The block has 1 continuous state(s).  
        State value : 6.977e-006  
        Initial guess : 0  
        Estimated : true
```

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
```

---

**Note** You can use `find` to identify scalar, vector, or matrix parameters. The dimensions of the Estimated value must match the dimensions of the parameters you are trying to find. For example, find only scalar estimated parameters by typing:

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

To find only vector estimated parameters with dimensions 1-by-2, type:

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

---

## Rerunning the Estimation Using Estimated Parameter Values

If the measured and simulated responses are not a good match after estimation, you can rerun the estimation using the estimated parameter values as initial parameter values.

```
% Extract all estimated parameter values and assign them
% as initial guesses.
for k=1:length(est.Parameters)
    if (est.Parameters(k).Estimated == true)
        est.Parameters(k).InitialGuess = est.Parameters(k).Value;
    end
end

% Restart the estimation using results from previous estimation.
est.estimate;
```

## Saving and Loading the Estimation Results

You can save the estimation results as a MAT-file by typing:

```
save filename est
```

You can specify any value for *filename* — including a folder path — provided the filename is supported by your operating system.

To load the file, type:

```
load filename
```

which loads `est` into the MATLAB workspace.

### **RC Circuit Parameters and Initial State Estimation (Code)**

In this example, you estimate the capacitance value `C1` of an RC circuit modeled in Simulink software using Simscape blocks.

The `sldo_rc_circuit.mat` file contains experimental I/O data and includes the following variables:

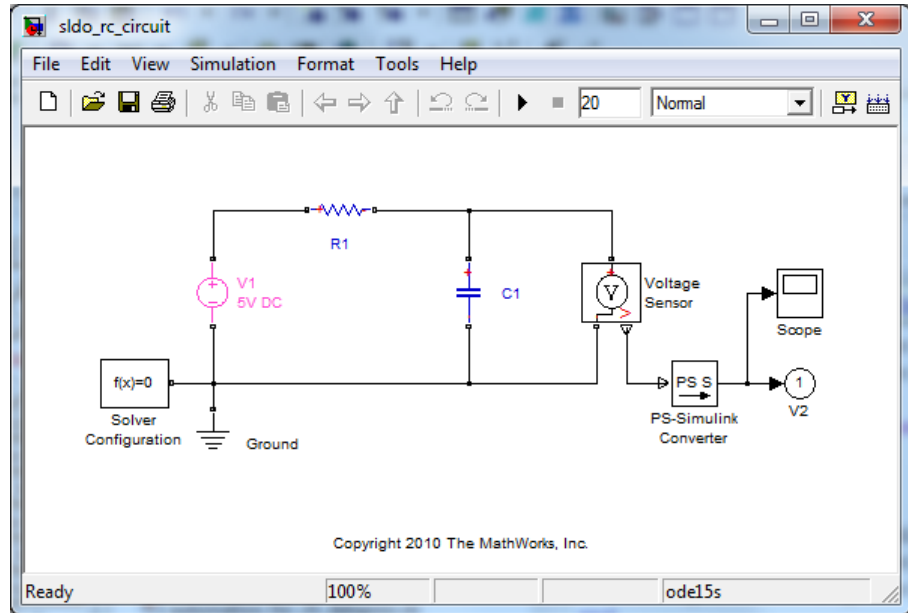
- `data` — Voltage across the capacitor when an input of 5V is applied to the RC circuit
- `time` — Time vector

To estimate the model parameter (`C1`) and state (initial capacitor voltage):

**1** Open the Simulink model.

```
model = 'sldo_rc_circuit';  
open_system(model)
```

The following model opens.

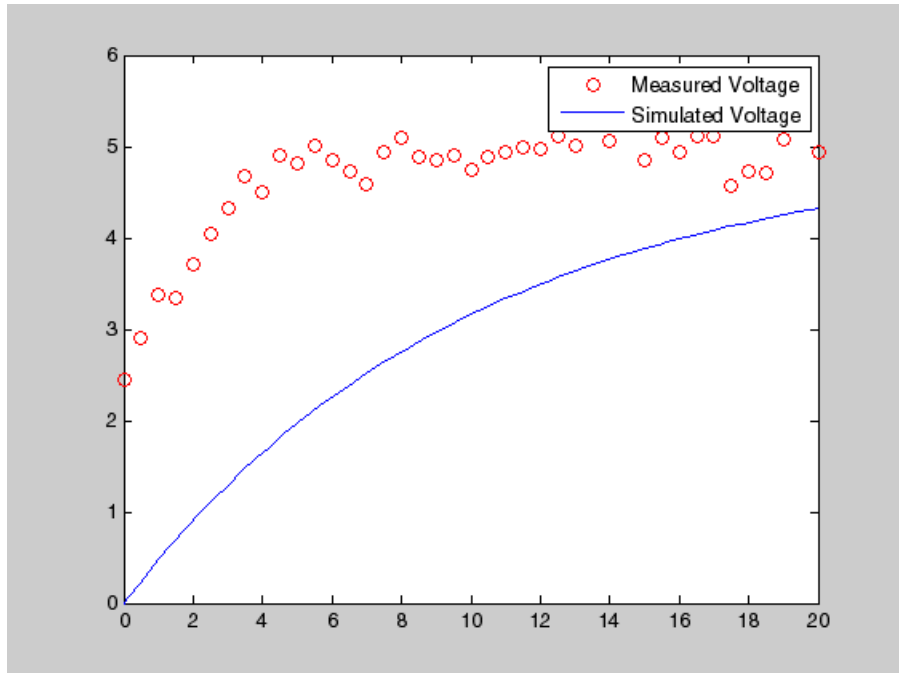


- 2 Perform a baseline simulation to compare the simulated model response with initial parameter values and measured data.

```
% Load experimental data.
load sldo_rc_circuit.mat
```

```
% Simulate model and compare response with experimental data.
SimOut = sim(model, 'ReturnWorkspaceOutputs', 'on');
plot(time,data,'ro', SimOut.find('tout'),SimOut.find('yout'),'b')
legend('Measured Voltage', 'Simulated Voltage')
```

The measured and simulated voltages do not match, as shown in the following figure.



**3** Create a Transient Experiment object, and assign the I/O data.

```
% Create data container for the experimental data.  
experiment = ParameterEstimator.TransientExperiment(model);  
experiment.Description = 'Response to 5V DC voltage';
```

```
% Specify measured output data for output voltage across  
% the capacitor.  
experiment.OutputData(1).Data = data;
```

```
% Specify the time vector.  
experiment.OutputData(1).Time = time;
```

**4** Create an Estimation object for the model.

```
est = ParameterEstimator.Estimation(model);
```

This command also creates Parameter and State objects for estimation.

- 5** Specify the experimental data to use for this estimation.

```
est.Experiments = experiment;
```

- 6** Estimate the model parameter C1.

- a** Specify the parameter to estimate.

```
% Specify the parameter to estimate.
est.Parameters(1).Estimated = true;
% Specify initial guess C1 = 470uF.
est.Parameters(1).InitialGuess = 470e-6;
```

- b** Estimate the parameter.

```
est.OptimOptions.Display = 'iter';
est.estimate
```

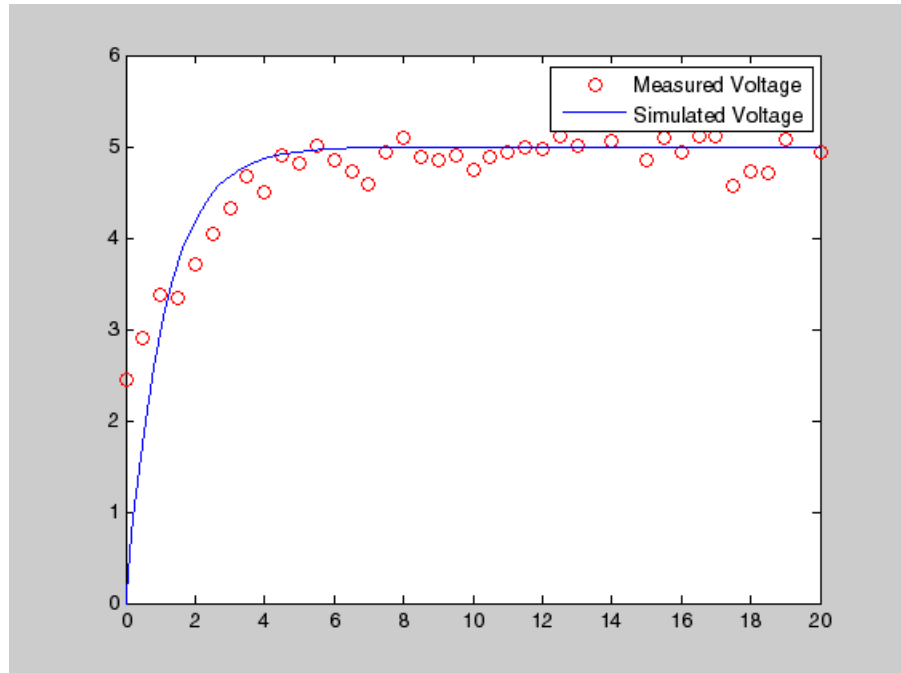
To view the estimated parameter value, type:

```
est.Parameters(1).Value
```

- 7** Simulate the model with the estimated parameters to see how well it matches the measured data.

```
SimOut = sim(model, 'ReturnWorkspaceOutputs', 'on');
plot(time,data,'ro', SimOut.find('tout'),SimOut.find('yout'),'b')
legend('Measured Voltage', 'Simulated Voltage')
```

The measured and simulated voltages match, as shown in the following figure. Because the software uses the initial capacitor voltage specified in the model during estimation, the initial voltage does not match the measured data.



Next, you estimate the initial capacitor voltage together with the capacitor value.

**8** Estimate the capacitance value together with the initial capacitor voltage.

**a** Specify initial state (initial capacitor voltage) to estimate.

```
% Specify initial state to estimate.  
est.States(1).Estimated = true;  
% Initial guess for C1 voltage 1V  
est.States(1).InitialGuess = 1.0;
```

**b** Estimate initial state and model parameter.

```
est.estimate
```

**9** Simulate the model to verify how well the simulated response using the estimated initial state and parameter matches the measured data.

**a** Assign initial state value to Simulink state structure before simulation.

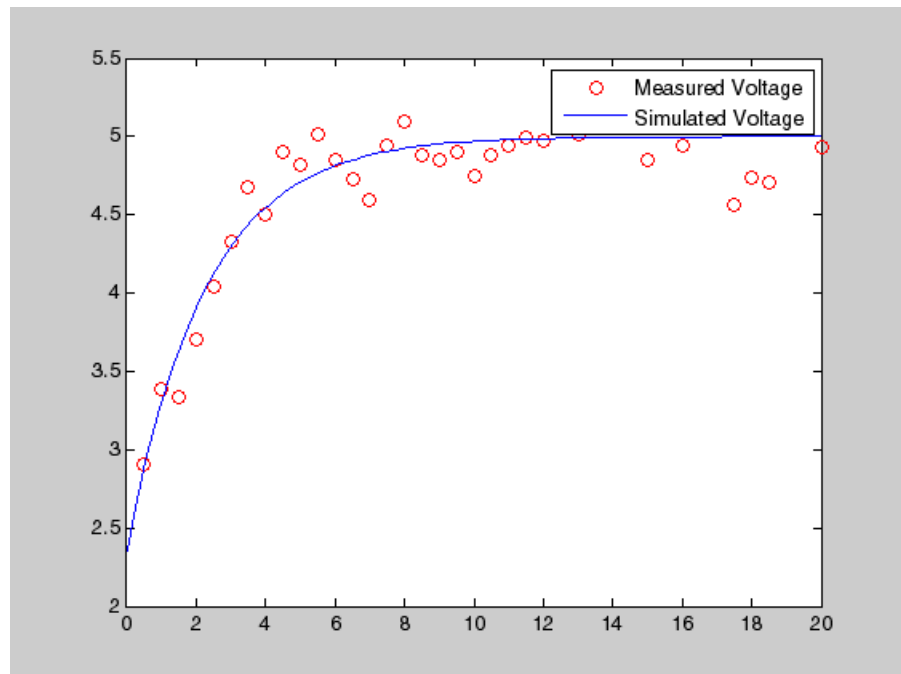


```
x0 = Simulink.BlockDiagram.getInitialState(model);
x0.signals(1).values = est.States(1).Value;
```

**b** Simulate the model.

```
SimOut = sim(model, 'LoadInitialState','on', 'InitialState','x0');
plot(time,data,'ro', SimOut.find('tout'),SimOut.find('yout'),'b')
```

The simulated voltage now accounts for the initial capacitor voltage, as shown in the following figure.



## Parameter Estimation Objects

The following sections describe how to create and modify parameter estimation objects:

- “Creating Transient Experiment Data Objects” on page 2-106
- “Creating Estimation Objects” on page 2-109

- “Parameter Objects” on page 2-117
- “State Objects” on page 2-120
- “Transient Data Objects” on page 2-123
- “State Data Objects” on page 2-129

First, a quick look at terminology:

- *Objects* are instantiations of *classes*.
- *Classes* define *properties* and *methods*. Classes can be grouped in a parent folder called the *package* folder.
- You use a *constructor* to create an object, and use the `set` method or dot notation to modify the properties of your objects.

For more information, see *Object-Oriented Programming* in the MATLAB documentation.

### **Creating Transient Experiment Data Objects**

- “What is a Transient Experiment Object?” on page 2-106
- “Constructor” on page 2-107
- “Properties” on page 2-107
- “Example: Creating a Transient Experiment Object” on page 2-107
- “Example: Creating an Experiment Object Using Transient Data and State Data Objects” on page 2-108
- “Modifying Properties” on page 2-109
- “Methods” on page 2-109

### **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. This object belongs to the `ParameterEstimator` package.

---

**Note** Creating a `TransientExperiment` object automatically creates `TransientData` and `StateData` objects.

---

## Constructor

The syntax to create a `TransientExperiment` object is:

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

where `model` specifies the name of the Simulink model.

## Properties

<code>Model</code>	Simulink model with which this experiment is associated.
<code>InputData</code> , <code>OutputData</code>	<p>TransientData 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:</p> <ul style="list-style-type: none"><li>• For input ports, assign zeros to these ports/channels during simulation.</li><li>• For output ports, don't use these ports/channels in the cost function.</li></ul>
<code>InitialStates</code>	StateData objects associated with dynamic blocks in the model.
<code>InitFcn</code>	Function to configure the model for this particular experiment.

## Example: Creating a Transient Experiment Object

To create an empty `TransientExperiment` for the `f14` model:

```
% Open the model.  
f14;  
% Create the transient experiment object.
```

```
exp1 = ParameterEstimator.TransientExperiment('f14')
```

The TransientExperiment object is shown as follows:

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

The Input Data and Output Data are TransientData object. The Initial States are StateData objects.

### **Example: Creating an Experiment Object Using Transient Data and State Data Objects**

To create a transient experiment from TransientData objects for I/Os and StateData objects for states:

```
% Open the model.
vdp;

% Create transient data object for the output data.
out1 = ParameterEstimator.TransientData('vdp/Out1');

% Create state data object for the initial state.
ic1 = ParameterEstimator.StateData('vdp/x1');
```

```
% Create an experiment using the previously-defined objects.
exp1 = ParameterEstimator.TransientExperiment...
(gcs, [], out1, ic1)
```

The TransientExperiment object is shown as follows:

Experimental transient data set for the model 'vdp':

```
Output Data
  (1) vdp/Out1

Input Data
  (none)

Initial States
  (1) vdp/x1
```

## Modifying Properties

The objects in `InputData`, `OutputData`, and `InitialStates` properties can be modified or removed as necessary. For example, to modify the `InputData` and `StateData` properties of the `TransientExperiment` object created in “Example: Creating a Transient Experiment Object” on page 2-107, type:

```
exp1.InputData(1).Data = [10 20 30]
exp1.InputData(1).Time = [1 2 3]
exp1.InitialStates(1).Data = 0.3
```

## Methods

update	Updates the object after the Simulink model has been modified. The objects in the <code>InputData</code> , <code>OutputData</code> , and <code>InitialStates</code> properties are also updated.
--------	--

## Creating Estimation Objects

- “What is an Estimation Object?” on page 2-110

- “Constructor” on page 2-110
- “Properties” on page 2-111
- “Example: Creating an Estimation Object” on page 2-115
- “Example: Estimating Parameters and States” on page 2-115
- “Modifying Properties” on page 2-116
- “Methods” on page 2-116

### **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. This object belongs to the `ParameterEstimator` package.

---

**Note** Creating an Estimation object automatically creates `Parameter` and `State` objects.

---

### **Constructor**

The syntax to create an estimation object is:

```
est = ParameterEstimator.Estimation('model');
```

Alternatively, you can create `Parameter` and `TransientExperiment` objects first and use them to construct an `Estimation` object using one of the following syntaxes:

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

`hParam` is an array of `Parameter` objects and `hExps` is an array of `TransientExperiment` objects.

## Properties

Model	Name of the Simulink model associated with this estimation.
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	Parameters objects to be used in estimations.
States	States 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 <code>simset</code> structure. This property is initialized to <code>simget(this.Model)</code> .
OptimOptions	Estimation options. Has the following fields:

Property	Description	Possible Settings
Method	Optimization solver. The following solvers are available: <ul style="list-style-type: none"> <li>• <code>lsqnonlin</code> — Optimization Toolbox function <code>lsqnonlin</code></li> <li>• <code>fmincon</code> — Optimization Toolbox function <code>fmincon</code></li> <li>• <code>patternsearch</code> — Global Optimization</li> </ul>	{'lsqnonlin'}   'fmincon'   'patternsearch'   'fminsearch'

	<p>Toolbox function <code>patternsearch</code></p> <ul style="list-style-type: none"> <li>• <code>fminsearch</code> — Optimization Toolbox function <code>fminsearch</code></li> </ul>	
Algorithm	The algorithm used by the optimization method.	<p>For <code>fmincon</code> optimization method:</p> <ul style="list-style-type: none"> <li>• <code>{'active-set'}</code></li> <li>• <code>'trust-region-reflective'</code></li> <li>• <code>'interior-point'</code></li> </ul>
Display	<p>The level of information that the optimization displays:</p> <ul style="list-style-type: none"> <li>• <code>off</code> — No output</li> <li>• <code>iter</code> — Output at each iteration</li> <li>• <code>final</code> — Final output only</li> <li>• <code>notify</code> — Output only if the function does not converge</li> </ul>	<code>'off'   {'iter'}   'final'   'notify'</code>
GradientType	<p>Method used to calculate gradients. Use one of the following finite difference methods for gradient calculation:</p> <ul style="list-style-type: none"> <li>• <code>basic</code> — Default method for computing the gradients</li> <li>• <code>refined</code> — Offers a more robust and less noisy gradient calculation method than <code>'basic'</code></li> </ul>	<code>{'basic'}   'refined'</code>



	The refined method is sometimes more expensive, and does not work with certain models such as SimPowerSystems models.	
MaxIter	Maximum number of iterations allowed	Positive integer
TolFun	Termination tolerance on the function value	Positive scalar
TolX	Termination tolerance on the parameter values	Positive scalar
UseParallel	<p>Parallel computing option for the following optimization methods:</p> <ul style="list-style-type: none"> <li>• lsqnonlin</li> <li>• fmincon</li> <li>• patternsearch</li> </ul> <hr/> <p><b>Note</b> Parallel Computing Toolbox software must be installed to enable parallel computing for the optimization methods.</p> <hr/> <p>When set to 'always', the methods compute the following in parallel:</p> <ul style="list-style-type: none"> <li>• lsqnonlin and fmincon — Compute</li> </ul>	'always'   {'never'}

	<p>finite difference gradients</p> <ul style="list-style-type: none"> <li>• <b>patternsearch</b> — Performs population evaluation</li> </ul> <p>Disable the option by setting to 'never'.</p>	
<b>ParallelPathDependencies</b>	Options to store model path dependencies when using parallel computing	Cell array of strings
<b>SearchMethod</b>	Search options for use with the <b>patternsearch</b> method	See “Search Options” in the Global Optimization Toolbox documentation.
<b>CostType</b>	Cost function that the optimization solver attempts to minimize.	'SAE'   {'SSE'}
<b>DiffMaxChange</b>	Maximum change in variables for finite-difference gradients	Positive scalar
<b>DiffMinChange</b>	Minimum change in variables for finite-difference gradients	Positive scalar
<b>MaxFunEvals</b>	Maximum number of function evaluations allowed	Positive integer
<b>RobustCost</b>	Use a robust cost function instead of the default least-squares cost. Useful if the experimental data has many outliers, or if your data is noisy.	'on'   {'off'}

**EstimInfo** This property is used to store estimation-related information at each iteration of the optimizer, and is initialized as

```
this.EstimInfo = struct( 'Cost', [],...
                        'Covariance', [],...
                        'FCount', [],...
                        'FirstOrd', [],...
                        'Gradient', [],...
                        'Iteration', [],...
                        'Procedure', [],...
                        'StepSize', [],...
                        'Values', [] );
```

### Example: Creating an Estimation Object

To create an estimation object for the f14 model:

```
% Open the model, if it is not already open.
f14;
% Create estimation object.
est1 = ParameterEstimator.Estimation(gcs)
```

This command returns the following result:

Estimated variables for the model 'f14':

Estimated Parameters

Using Experiments  
(none)

### Example: Estimating Parameters and States

Estimate the parameters Ta and Kf and states of the f14 model:

```
% Open the model, if it is not already open.
f14;
% Define experiments.
```

```
exp1 = ParameterEstimator.TransientExperiment(gcs);  
% Create estimation object and assign experimental data.  
est1 = ParameterEstimator.Estimation(gcs, [], exp1);  
% Specify parameters to estimate.  
est1.Parameters(2).Estimated = true;  
est.Parameters(9).Estimated = true;  
% Specify states to estimate.  
est1.States(1).Estimated = true;  
% Estimate parameters and states.  
est1.estimate;
```

### 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
```

### Methods

compare	Compare an experiment and a simulation.
simulate	Simulate the model with current parameters and states.
estimate	Run an estimation.
update	Update the estimation object after the Simulink model has been modified.

## Parameter Objects

- “What is a Parameter Object?” on page 2-117
- “Constructor” on page 2-117
- “Properties” on page 2-118
- “Example: Creating a Parameter Object Automatically” on page 2-119
- “Modifying Properties” on page 2-119
- “Methods” on page 2-120

### What is a Parameter Object?

The Parameter object refers to the parameters of the Simulink model. This object also stores information such as whether the parameter is to be estimated, initial values, current values, and ranges. This object belongs to the `ParameterEstimator` package.

One Parameter object corresponds to each parameter in the Simulink model. These objects represent parameters represented as scalars, vectors, and multidimensional arrays.

### Constructor

Creating an Estimation object automatically creates Parameter objects corresponding to each model parameter.

If your model has a large number of parameters, constructing a Parameter object manually for each parameter that you want to estimate makes handling of the objects more manageable. For an example, see the F14 Parameter Estimation at the Command Line demo.

To manually construct a Parameter object, type one of the following syntaxes:

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

## Properties

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"><li>• 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.</li><li>• 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.</li></ul> <p>This property is set to false by default, meaning that the parameter value is not estimated.</p>
InitialGuess	<p>Separate properties are required to hold the initial and current values of the parameters. 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.</p> <p>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"><li>• If any element in <code>Estimated</code> is set to true, then the corresponding element of <code>InitialGuess</code></li></ul>

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.

<code>Minimum, Maximum</code>	Parameter range.
<code>TypicalValue</code>	The typical values of the parameters. This property is used in estimations for scaling purposes. The default value is 1.

### Example: Creating a Parameter Object Automatically

To automatically create parameter objects for the f14 model:

```
% Open the model.
f14;
% Create an estimation object.
est = ParameterEstimator.Estimation('f14')
```

To view the parameter objects:

```
est.Parameters
```

To view the value of a parameter:

```
est.Parameters(1).Value
```

### Modifying Properties

After a parameter object is created, you can modify its properties. For example:

```
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`.

### Methods

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

### State Objects

- “What is a State Object?” on page 2-120
- “Constructor” on page 2-120
- “Properties” on page 2-121
- “Example: Creating a State Object Automatically” on page 2-122
- “Modifying Properties” on page 2-123
- “Methods” on page 2-123

### What is a State Object?

The `State` object is similar to the `Parameter` object, and refers to the states of the Simulink model. This object belongs to the `ParameterEstimator` package.

One `State` object corresponds to each block with states in the model.

### Constructor

Creating an `Estimation` object automatically creates `State` objects.

If your model has a large number of states, constructing a `State` object manually for each state that you want to estimate makes handling of the objects more manageable. For an example, see the F14 Parameter Estimation at the Command Line demo.

To manually construct a `State` object, type one of the following syntaxes:



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

## Properties

Block	Name of the Simulink block whose states are defined by this object.
StateName	Name of a state. This property shows a state name if your model has: <ul style="list-style-type: none"><li>• Blocks, such as integrators, containing states with unique names</li><li>• Blocks from Simscape, SimMechanics, SimPowerSystems and SimHydraulics software</li></ul>
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.
Estimated	A Boolean array of the same size as that of Value. Depending on the value of the elements of the Estimated property, the behavior of the corresponding elements of Value is as follows: <ul style="list-style-type: none"><li>• The elements of Value are estimated if the corresponding elements in Estimate are set to true. The result is stored in the Value property.</li><li>• The elements of Value are not estimated if the corresponding elements in Estimated are set to false. However, these elements are used to reset the corresponding states during estimations.</li></ul> This property is set to false by default, meaning that the state value is not estimated.

<code>InitialGuess</code>	<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.</p> <p>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"><li>• 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.</li><li>• 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.</li></ul>
<code>Minimum, Maximum</code>	State vector range.
<code>TypicalValue</code>	The typical values of the states. This property is used in estimations for scaling purposes. The default value is 1.
<code>Ts</code>	Sampling time of discrete blocks. Set to zero for continuous blocks. This property is read-only and is currently used for information only.

### Example: Creating a State Object Automatically

To create a state objects for all blocks with states in the `f14` model:

```
% Open the model.  
f14;  
% Create an estimation object.  
est = ParameterEstimator.Estimation('f14')
```

To view the state objects:

```
est.States
```

To view the value of a state:

```
est.States(1).Value
```

## Modifying Properties

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

```
est.States(1).Estimated = true; % Estimate this state
```

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

## Methods

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

## Transient Data Objects

- “What is a Transient Data Object?” on page 2-124
- “Constructor” on page 2-124
- “Properties” on page 2-125
- “Example: Creating Transient Data Objects from Transient Experiment Object” on page 2-127
- “Example: Using Transient Data Objects for Creating an Experiment Object” on page 2-127
- “Modifying Properties” on page 2-128
- “Methods” on page 2-128

## 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. This object belongs to the `ParameterEstimator` package.

Transient data objects are associated with three types of Simulink blocks:

- Inport blocks
- Outport blocks
- Blocks that have logged signals

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

Creating an `TransientExperiment` object automatically creates `TransientData` objects corresponding to Inport blocks, Outport blocks or logged signals in the model.

Alternatively, you can also create a `TransientData` object using the following constructor syntaxes and assign them to the `TransientExperiment` object:

```
% Inport Outport block.
io_ports = ParameterEstimator.TransientData('block');

% Block with logged signal.
logged_signals = ParameterEstimator.TransientData('block',portnumber);

% Additional input arguments.
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

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 <code>Block</code> property, which may be <code>Inport</code> , <code>Outport</code> , or <code>Signal</code> .
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 <code>CompiledPortDimensions</code> 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 <code>Dimensions</code> is either a scalar or a 1-by-2 array.
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"><li>• Scalar or vector-valued data. The data is of the form <math>Ns \ m</math>, where <math>Ns</math> is the number of data samples, and <math>m</math> is the number of channels in the signal.</li><li>• Multidimensional data (matrix and higher dimensions). The data is of the form <math>m1 \ . \ . \ mn \ Ns</math>, where <math>Ns</math> is the number of data samples, and <math>mi</math> is the number of channels in the <math>i</math>th dimension of the signal.</li><li>• For missing or unspecified data, NaNs are used.</li></ul>

`Ts`,  
`Tstart`,  
`Tstop`

For uniformly sampled data, `Ts` is the sample time and `Tstart` is the start time of the signal. The stop time `Tstop` and the time vector `Time` are given by

$$Tstop = Tstart + Ts * (Ns - 1)$$

$$Time = Tstart : Ts : Tstop$$

For nonuniform time data, `Ts` is set to NaN, and the start and stop times are calculated from the time vector.

`Time`

The time data in column vector format. The length of `Time` must be consistent with the number of samples in `Data`.

For a nonuniformly spaced `Time` vector, its length should match the length of `Data`.

Otherwise, `Time` is automatically adjusted based on the length of `Data`.

Modifying `Ts` resets `Time` internally. In this case, `Time` is a virtual property whose value is computed from `Ts` and `Tstart` when you request it. The rules for setting time related properties are

- Modifying `Time` sets

`Ts = NaN`

`Tstart = Time(1)`

- 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.

### Example: Creating Transient Data Objects from Transient Experiment Object

To automatically create transient data objects from a transient experiment object:

```
% Open the model.
f14;
% Create the transient experiment object.
exp1 = ParameterEstimator.TransientExperiment('f14')
```

To view the transient data objects:

```
class(exp1.InputData)
exp1.InputData
```

These commands return the following results:

```
ans =
```

```
ParameterEstimator.TransientData
```

```
(1) Transient data for Inport block f14/u:
```

```
Sampling time: 1 sec.
```

```
Data set has 0 samples and 1 channels.
```

### Example: Using Transient Data Objects for Creating an Experiment Object

To create an experiment object using transient data objects:

```
% Open the model.
vdp;

% Create transient data object for the output data.
out1 = ParameterEstimator.TransientData('vdp/Out1');

% Create an experiment using the previously-defined objects.
exp1 = ParameterEstimator.TransientExperiment...
(gcs, [], out1);
```

### Modifying Properties

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.

### Methods

<code>select</code>	Extracts a portion of data. The result is returned in a new transient data object. For example:  <pre>in2 = select(in1, 'Sample', 10:100); % 91 samples in3 = select(in1, 'Range', [1 4]); % Samples for 1&lt;t&lt;4 % ... or an alternative in3 = select(in1, 'Sample', find(in1.Time &gt; 1 &amp; in1.Time &lt;</pre>
---------------------	---

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
```



<code>hiliteBlock</code>	Highlights the block associated with this object in the Simulink diagram.
<code>update</code>	Updates the transient data object from the corresponding Simulink model block.

## State Data Objects

- “What is a State Data Object?” on page 2-129
- “Constructor” on page 2-130
- “Properties” on page 2-130
- “Example: Creating State Data Objects from Transient Experiment Object” on page 2-131
- “Example: Using State Data Objects for Creating an Experiment Object” on page 2-131
- “Modifying Properties” on page 2-132
- “Methods” on page 2-132

### What is a State Data Object?

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

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

Creating an `Transient Experiment` object automatically creates `StateData` objects.

To manually create a `StateData` object, type one of the following syntaxes:

```
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

<code>Block</code>	Name of the Simulink block whose states are defined by this object.
<code>StateName</code>	Name of a state. This property shows a state name if your model has: <ul style="list-style-type: none"><li>• Blocks, such as integrators, containing states with unique names</li><li>• Blocks from Simscape, SimMechanics, SimPowerSystems and SimHydraulics software</li></ul>
<code>Dimensions</code>	Scalar value to store the number of states of the relevant block.
<code>Data</code>	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: <ul style="list-style-type: none"><li>• If <code>Data</code> is not empty, use it when forming the state vector.</li><li>• 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</li></ul>

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` Sampling time of discrete blocks. Set to 0 for continuous blocks. This property is read only and is currently used for information only.

### Example: Creating State Data Objects from Transient Experiment Object

To create transient data objects automatically by creating a transient experiment object:

```
% Open the model.
f14;
% Create the transient experiment object.
exp1 = ParameterEstimator.TransientExperiment('f14')
```

To view the state data objects:

```
exp1.InitialStates
```

### Example: Using State Data Objects for Creating an Experiment Object

To create an experiment object using transient data objects:

```
% Open the model.
vdp;

% Create state data object for the initial state.
ic1 = ParameterEstimator.StateData('vdp/x1');

% Create an experiment using the previously-defined objects.
exp1 = ParameterEstimator.TransientExperiment...
```

```
(gcs, [], ic1);
```

## 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.

## Methods

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

## How to Use Parallel Computing (Code)

After you configure your system for parallel computing, as described in “Configure Your System for Parallel Computing” on page 2-59, you can estimate the model parameters using the command-line functions. To learn more about parameter estimation using parallel computing, see “Speedup Using Parallel Computing” on page 2-55.

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 “How to Estimate Parameters (Code)” on page 2-87.

- 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 “Specify Model Dependencies” on page 2-60, 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 2-65.



# Response Optimization

---

- “Response Optimization Workflow” on page 3-3
- “How the Optimization Algorithm Formulates Minimization Problems” on page 3-4
- “Specify Signals to Log (GUI)” on page 3-14
- “Specifying Time Domain Design Requirements (GUI)” on page 3-16
- “Specifying Frequency Domain Design Requirements (GUI)” on page 3-32
- “Design Variables (GUI)” on page 3-54
- “General Options” on page 3-60
- “Optimization Options” on page 3-64
- “Creating Linearization I/O Sets (GUI)” on page 3-69
- “Linearization Options” on page 3-71
- “Plots” on page 3-74
- “Compare Requirements and Design Variables Using Spider Plot” on page 3-81
- “Run the Optimization (GUI)” on page 3-85
- “Design Optimization to Meet Time- and Frequency-Domain Requirements” on page 3-88
- “Design Optimization to Meet a Custom Objective Using the GUI” on page 3-103
- “Design Optimization to Meet a Custom Objective at the Command Line” on page 3-123
- “Design Optimization to Meet Custom Signal Requirements (GUI)” on page 3-133

- “Design Optimization to Meet Frequency-Domain Requirements” on page 3-138
- “Optimizing Parameters for Robustness” on page 3-157
- “Accelerating Model Simulations During Optimization” on page 3-171
- “Speedup Using Parallel Computing” on page 3-173
- “How to Use Parallel Computing” on page 3-178
- “Refining and Troubleshooting Optimization Results” on page 3-186
- “Managing Design Optimization Tool Session” on page 3-196



## Response Optimization Workflow

The process for optimizing model parameters to meet design requirements consists of the following tasks:

- 1 Specify design requirements
- 2 Specify design variables
- 3 (Optional) Specify optimization options
- 4 Optimize the design

### Related Examples

- “Optimize Controller Parameters to Meet Step Response Requirements (GUI)”
- “Optimize Controller Parameters to Track Reference Signal (GUI)”
- “Optimize Controller Parameters to Meet Step Response Requirements (Code)”

## How the Optimization Algorithm Formulates Minimization Problems

When you optimize parameters of a Simulink model to meet design requirements, Simulink Design Optimization software automatically converts the 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 optimization methods to adjust tuned parameters to better meet the objectives.

This topic describes how the software formulates the constrained optimization problem used by the optimization algorithms. For each optimization algorithm, the software formulates one of the following types of minimization problems:

- Feasibility
- Tracking
- Mixed feasibility and tracking

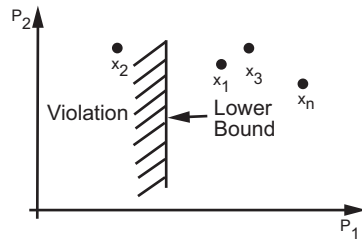
For more information on how each optimization algorithm formulates these problems, see:

- “Gradient Descent Method Problem Formulations” on page 3-8
- “Simplex Search Method Problem Formulations” on page 3-10
- “Pattern Search Method Problem Formulations” on page 3-11
- “Gradient Computations” on page 3-12

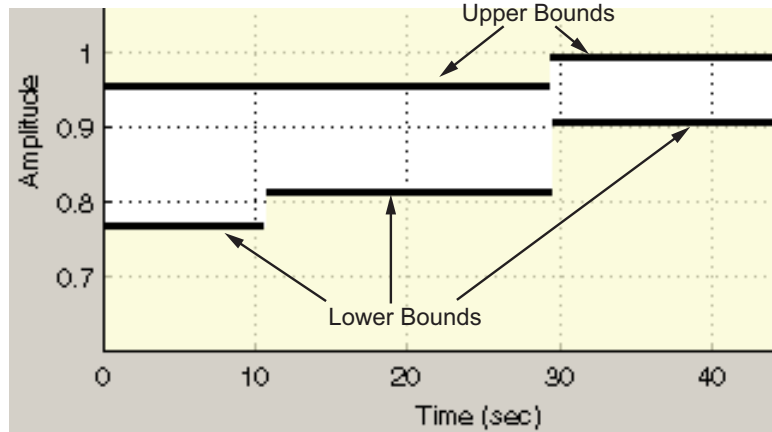
### Feasibility Problem and Constraint Formulation

*Feasibility* means that the optimization algorithm finds parameter values that satisfy all constraints to within specified tolerances but does not minimize any objective or cost function in doing so.

In the following figure,  $x_1$ ,  $x_3$ , and  $x_n$  represent a combination of parameter values  $P_1$  and  $P_2$  and are feasible solutions because they do not violate the lower bound constraint.



In a Simulink model, you constrain a signal by specifying lower and upper bounds in a Check block or a requirement object, as shown in the following figure.



These constraints are *piecewise linear bounds*. A piecewise linear bound  $y_{bnd}$  with  $n$  edges can be represented as:

$$y_{bnd}(t) = \begin{cases} y_1(t) & t_1 \leq t \leq t_2 \\ y_2(t) & t_2 \leq t \leq t_3 \\ \vdots & \vdots \\ y_n(t) & t_n \leq t \leq t_{n+1} \end{cases},$$

The software computes the signed distance between the simulated response and the edge. The signed distance for lower bounds is:

$$c = \begin{bmatrix} \max_{t_1 \leq t \leq t_2} y_{bnd} - y_{sim} \\ \max_{t_2 \leq t \leq t_3} y_{bnd} - y_{sim} \\ \max_{t_n \leq t \leq t_{n+1}} y_{bnd} - y_{sim} \end{bmatrix},$$

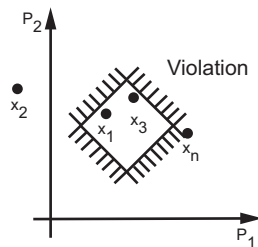
where  $y_{sim}$  is the simulated response and is a function of the parameters being optimized.

The signed distance for upper bounds is:

$$c = \begin{bmatrix} \max_{t_1 \leq t \leq t_2} y_{sim} - y_{bnd} \\ \max_{t_2 \leq t \leq t_3} y_{sim} - y_{bnd} \\ \max_{t_n \leq t \leq t_{n+1}} y_{sim} - y_{bnd} \end{bmatrix}.$$

At the command line, `opt_fcn` supplies `c` directly from the `Cleq` field of `vals`.

If *all* the constraints are met ( $c \leq 0$ ) for some combination of parameter values, then that solution is said to be feasible. In the following figure,  $x_1$  and  $x_3$  are feasible solutions.



When your model has multiple requirements or vector signals feeding a requirement, the constraint vector is extended with the constraint violations for each signal and bound:

$$C = [c_1; c_2; \dots; c_n].$$

## Tracking Problem

In addition to lower and upper bounds, you can specify a reference signal in a Check Against Reference block or `sdo.requirements.SignalTracking` object, which the Simulink model output can track. The tracking objective is a sum-squared-error tracking objective.

You specify the reference signal as a sequence of time-amplitude pairs:

$$y_{ref}(t_{ref}), t_{ref} \in \{T_{ref0}, T_{ref1}, \dots, T_{refN}\}.$$

The software computes the simulated response as a sequence of time-amplitude pairs:

$$y_{sim}(t_{sim}), t_{sim} \in \{T_{sim0}, T_{sim1}, \dots, T_{simN}\},$$

where some values of  $t_{sim}$  may match the values of  $t_{ref}$ .

A new time base,  $t_{new}$ , is formed from the union of the elements of  $t_{ref}$  and  $t_{sim}$ . Elements that are not within the minimum-maximum range of both  $t_{ref}$  and  $t_{sim}$  are omitted:

$$t_{new} = \{t : t_{sim} \cap t_{ref}\}$$

Using linear interpolation, the software computes the values of  $y_{ref}$  and  $y_{sim}$  at the time points in  $t_{new}$  and then computes the scaled error:

$$e(t_{new}) = \frac{(y_{sim}(t_{new}) - y_{ref}(t_{new}))}{\max_{t_{new}} |y_{ref}|}.$$

Finally, the software computes the weighted, integral square error:

$$f = \int w(t) e(t)^2 dt.$$

**Note** The weight  $w(t)$  is 1 by default. You can specify a different value of weight only at the command line.

When your model has requirements or vector signals feeding a requirement, the tracking objective equals the sum of the individual tracking integral errors for each signal:

$$F = \sum f_i.$$

### Gradient Descent Method Problem Formulations

The Gradient Descent method uses the Optimization Toolbox function `fmincon` to optimize model parameters to meet design requirements.

Problem Type	Problem Formulation
<p><b>Feasibility Problem</b></p>	<p>The software formulates the constraint <math>C(x)</math> as described in “Feasibility Problem and Constraint Formulation” on page 3-4.</p> <ul style="list-style-type: none"> <li>If you do not select the maximally feasible solution option (i.e., the optimization terminates as soon as a feasible solution is found), the software uses the following problem formulation:           <math display="block">\begin{aligned} \min_{[x,\gamma]} &amp; 0 \\ \text{s.t. } &amp; C(x) \leq (1-w)\gamma \\ &amp; \underline{x} \leq x \leq \bar{x} \\ &amp; 0 \leq \gamma \end{aligned}</math> <p>The constraint weight vector <math>w</math> is dimensionally commensurate with <math>C(x)</math> and defaults to a vector of ones. When all the constraint weights are one, the scalar slack variable <math>\gamma</math> plays no role in the calculation. Otherwise, the slack variable permits a feasible solution with <math>C(x) \leq (1-w)\gamma</math> rather than <math>C(x) \leq 0</math>.</p> </li> <li>If you select the maximally feasible solution option (i.e., the optimization continues after an initial feasible solution is found), the software uses the following problem formulation:</li> </ul>

Problem Type	Problem Formulation
	$\min_{[x, \gamma]} \gamma$ $s.t. \quad C(x) \leq \gamma$ $\underline{x} \leq x \leq \bar{x}$ <p>In this formulation, <math>x</math> is a feasible solution if <math>\gamma \leq 0</math>.</p>
<b>Tracking Problem</b>	<p>The software formulates the tracking objective <math>F(x)</math> as described in “Tracking Problem” on page 3-7 and minimizes the tracking objective:</p> $\min_x F(x)$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$

Problem Type	Problem Formulation
<b>Mixed Feasibility and Tracking Problem</b>	<p>The software minimizes following problem formulation:</p> $\begin{aligned} \min_x \quad & F(x) \\ \text{s.t.} \quad & C(x) \leq 0 \\ & \underline{x} \leq x \leq \bar{x} \end{aligned}$ <hr/> <p><b>Note</b> When tracking a reference signal, the software ignores the maximally feasible solution option.</p> <hr/>

### Simplex Search Method Problem Formulations

The Simplex Search method uses the Optimization Toolbox function `fminsearch` and `fminbnd` to optimize model parameters to meet design requirements. `fminbnd` is used if one scalar parameter is being optimized, otherwise `fminsearch` is used. You cannot use parameter bounds  $\underline{x} \leq x \leq \bar{x}$  with `fminsearch`.

Problem Type	Problem Formulation
<b>Feasibility Problem</b>	<p>The software formulates the constraint <math>C(x)</math> as described in “Feasibility Problem and Constraint Formulation” on page 3-4 and then minimizes the maximum constraint violation:</p> $\min_x \max(C(x))$
<b>Tracking Problem</b>	<p>The software formulates the tracking objective <math>F(x)</math> as described in “Tracking Problem” on page 3-7 and then minimizes the tracking objective:</p>



Problem Type	Problem Formulation
	$\min_x F(x)$
<b>Mixed Feasibility and Tracking Problem</b>	<p>The software formulates the problem in two steps:</p> <ol style="list-style-type: none"> <li>1 Finds a feasible solution.</li> </ol> $\min_x \max(C(x))$ <ol style="list-style-type: none"> <li>2 Minimizes the tracking objective. The software uses the results from step 1 as initial guesses and maintains feasibility by introducing a discontinuous barrier in the optimization objective.</li> </ol> $\min_x \Gamma(x)$ <p>where</p> $\Gamma(x) = \begin{cases} \inf & \text{if } \max(C(x)) > 0 \\ F(x) & \text{otherwise} \end{cases}$

## Pattern Search Method Problem Formulations

The Pattern Search method uses the Global Optimization Toolbox function `patternsearch` to optimize model parameters to meet design requirements.

Problem Type	Problem Formulation
<b>Feasibility Problem</b>	<p>The software formulates the constraint <math>C(x)</math> as described in “Feasibility Problem and Constraint Formulation” on page 3-4 and then minimizes the maximum constraint violation:</p> $\min_x \max(C(x))$ <p>s.t. <math>\underline{x} \leq x \leq \bar{x}</math></p>
<b>Tracking Problem</b>	<p>The software formulates the tracking objective <math>F(x)</math> as described in “Tracking Problem” on page 3-7 and then minimizes the tracking objective:</p>

Problem Type	Problem Formulation
	$\min_x F(x)$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$
<p><b>Mixed Feasibility and Tracking Problem</b></p>	<p>The software formulates the problem in two steps:</p> <p><b>1</b> Finds a feasible solution.</p> $\min_x \max(C(x))$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$ <p><b>2</b> Minimizes the tracking objective. The software uses the results from step 1 as initial guesses and maintains feasibility by introducing a discontinuous barrier in the optimization objective.</p> $\min_x \Gamma(x)$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$ <p>where</p> $\Gamma(x) = \begin{cases} \inf & \text{if } \max(C(x)) > 0 \\ F(x) & \text{otherwise} \end{cases}$

### Gradient Computations

For the Gradient descent (fmincon) optimization solver, given an objective function `opt_fcn` and scalar parameter `x`, the gradients are computed using numerical perturbation:

$$dx = \sqrt[3]{\text{eps}} \times \max(x_{\text{typical}} - x)$$

$$dL = \max(x - dx, x_{\text{min}})$$

$$dR = \min(x + dx, x_{\text{max}})$$

$$F_L = \text{opt\_fcn}(dL)$$

$$F_R = \text{opt\_fcn}(dR)$$

$$\frac{dF}{dx} = \frac{(F_L - F_R)}{(dL - dR)}$$

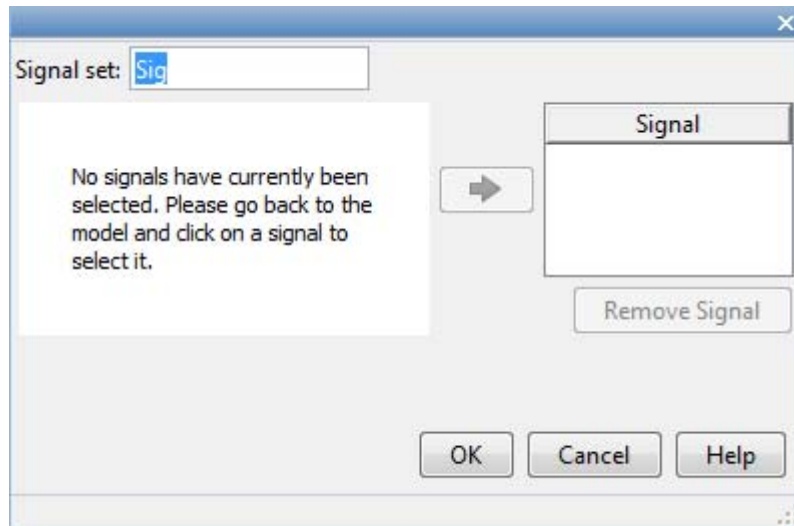
$dx$  is relatively large to accommodate simulation solver tolerances.

If you want to compute the gradients in any other way, you can do so in the cost function you write for performing design optimization programmatically. See `sdo.optimize` and `GradFcn` of `sdo.OptimizeOptions` for more information.


## Specify Signals to Log (GUI)

Design requirements require logged model signals. During optimization, the model is simulated using the current value of the design variables and the logged signal is used to evaluate the design requirements.

- 1 In the **Design Optimization tool**, select **Signal** in the **New** drop-down list. A window opens where you select a signal to log.
- 2 In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- 3 Select the signal and click  to add it to the signal set.
- 4 In **Signal set** box, enter a name for the selected signal set.

Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

**See Also**

- “Optimize Controller Parameters to Track Reference Signal (GUI)”
- `sdo.SimulationTest`

## Specifying Time Domain Design Requirements (GUI)

### In this section...

“Specifying Piecewise-Linear Lower and Upper Bounds” on page 4-4

“Specifying Step Response Characteristics” on page 4-35



“Track Reference Signals” on page 4-13

“Specifying Custom Requirements” on page 4-38

“Editing Design Requirements” on page 4-17


### Specifying Piecewise-Linear Lower and Upper Bounds

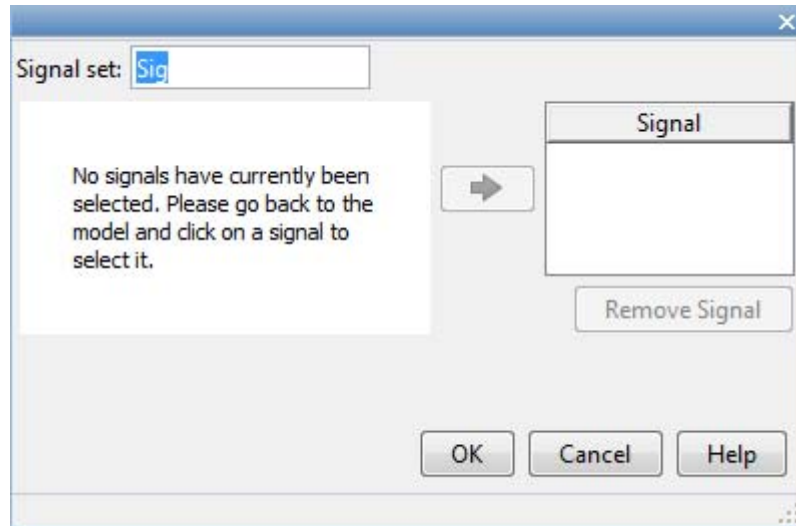
To specify upper and lower bounds on a signal:

- 1** In the Design Optimization tool, select **Signal Bound** in the **New** drop-down list. A window opens where you specify upper or lower bounds on a signal.
- 2** Specify a requirement name in the **Name** box.
- 3** Select the requirement type using the **Type** list.
- 4** Specify the edge start and end times and corresponding amplitude in the **Time (s)** and **Amplitude** columns.
- 5** Click  to specify additional bound edges.  
  
Select a row and click  to delete a bound edge.
- 6** In the **Select Signals to Bound** area, select a logged signal to apply the requirement to.


If you have already selected signals, as described in “Specify Signals to Log (GUI)” on page 3-14, they appear in the list. Select the corresponding check-box.

If you haven't selected a signal to log:

- a Click . A window opens where you specify the logged signal.
- b In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.

Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- 7 Click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the requirement also appears in the Design Optimization tool window.

- 8 (Optional) In the graphical display, you can:
  - “Moving Constraints” on page 4-6

- “Split and Join Constraints” on page 4-8

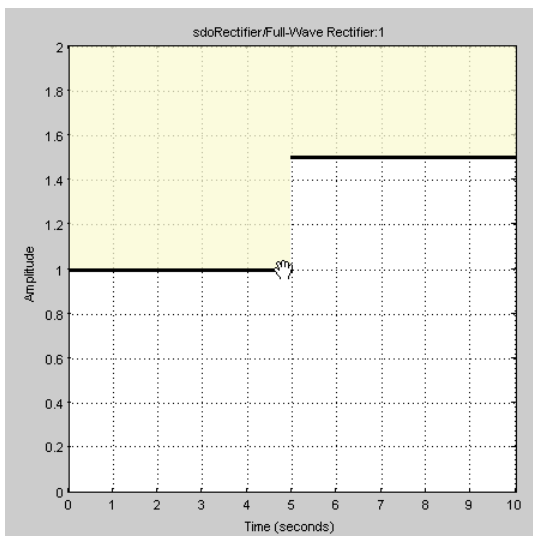
Alternatively, you can add a Check Custom Bounds block to your model to specify piecewise-linear bounds.

## Moving Constraints

- “Moving Constraints Graphically” on page 4-6
- “Positioning Constraints Exactly” on page 4-7

Constraint-bound edges define time-domain constraints you would like to place on a particular signal in your model. You can position these edges, which appear as a yellow shaded region bordered by a black line, graphically or exactly.

**Moving Constraints Graphically.** Use the mouse to click and drag edges in the amplitude versus time plot, as shown in the following figure.



- To move a constraint edge boundary or to change the slope of a constraint edge, position the pointer over a constraint edge 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 edges on either side of the boundary might not maintain their slopes.

- To move an entire constraint edge up, down, left, or right, position the mouse pointer over the edge 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 edges on either side of the boundary might not maintain their slopes.

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

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

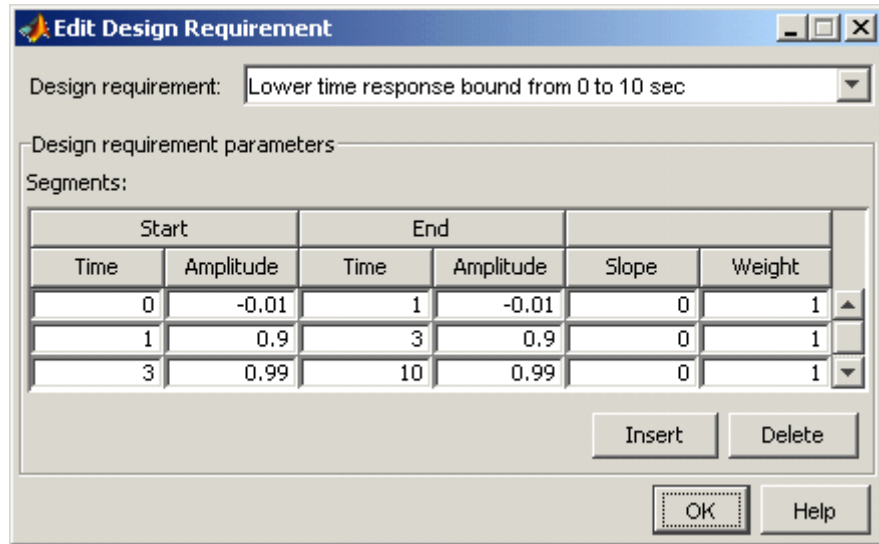
---

**Note** You can move a lower bound constraint edge above an upper bound constraint edge, or vice versa, but this produces an error when you attempt to run the optimization.

---

**Positioning Constraints Exactly.** To position a constraint edge exactly:

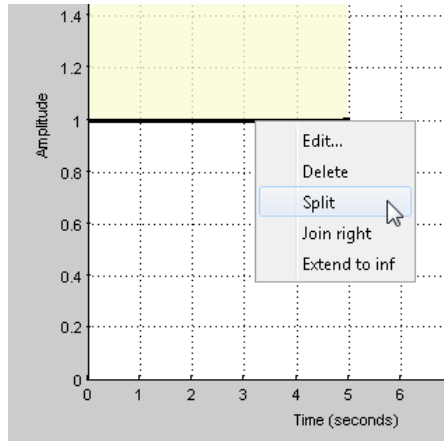
- 1 Position the pointer over the edge you want to move and right-click. Select **Edit** to open the Edit Design Requirement dialog box.



- 2 Specify the position of each constraint edge in the **Time** and **Amplitude** columns.

### Split and Join 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 segment too the left or right respectively.

## Specifying Step Response Characteristics

To specify step response characteristics:

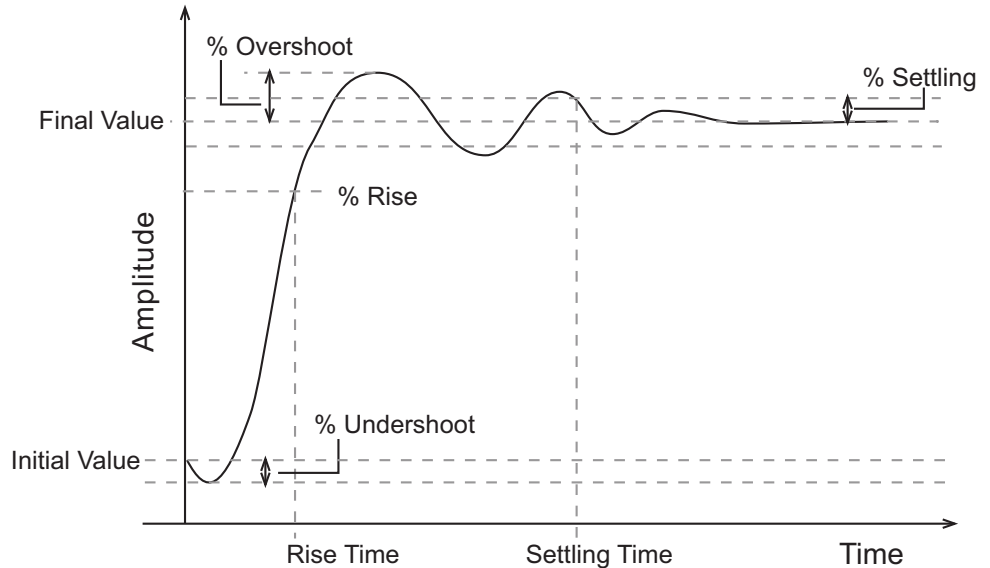
- 1 You can apply this requirement to either a signal or a linearization of your model.

In the Design Optimization Tool, click **New**. To apply this requirement to a signal, select the **Step Response Envelope** entry in the **New Time Domain Requirement** section of the **New** list. To apply this requirement to a linearization of your model, select the **Step Response Envelope** entry in the **New Frequency Domain Requirement** section of the **New** list. The latter option requires Simulink Control Design software.

A window opens where you specify the step response requirements on a signal, or system.

- 2 Specify a requirement name in the **Name** box.

**3** Specify the step response characteristics:



- **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
- **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.

#### 4 Specify the signals or systems to be bound.


You can apply this requirement to a model signal or to a linearization of your Simulink model (requires Simulink Control Design software).

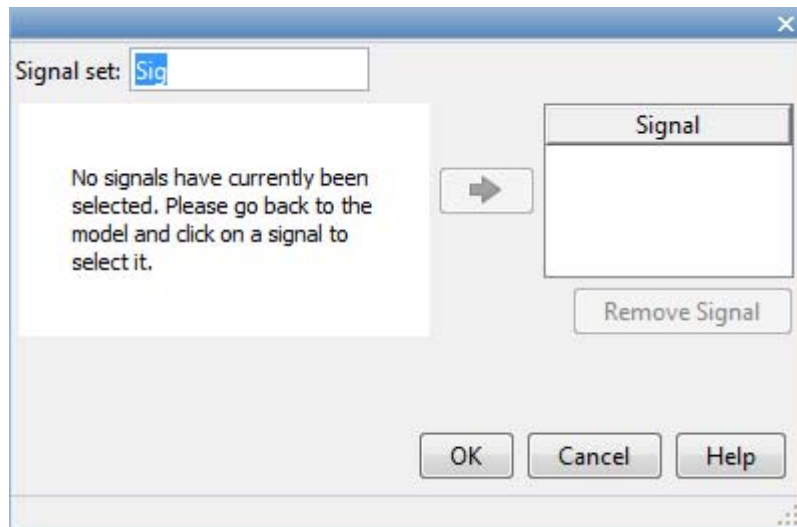
- Apply this requirement to a model signal:

In the **Select Signals to Bound** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, it appears in the list. Select the corresponding check-box.

If you haven't selected a signal to log:

- Click . A window opens where you specify the logged signal.
- In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.


Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- Apply this requirement to a linear system.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5 Click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the requirement also appears in the Design Optimization tool window.

Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.

## See Also

“Optimize Controller Parameters to Meet Step Response Requirements (GUI)”

## Track Reference Signals

Use reference tracking to force a model signal to match a desired signal.

To track a reference signal:


- 1** In the Design Optimization tool, select **Signal Tracking** in the **New** drop-down list. A window opens where you specify the reference signal to track.
- 2** Specify a requirement name in the **Name** box.
- 3** Define the reference signal by entering vectors, or variables from the workspace, in the **Time vector** and **Amplitude** fields.

Click **Update reference signal data** to use the new amplitude and time vector as the reference signal.

- 4** Specify how the optimization solver minimizes the error between the reference and model signals using the **Tracking Method** list:
  - SSE — Reduces the sum of squared errors
  - SAE — Reduces the sum of absolute errors
- 5** In the **Specify Signal to Track Reference Signal** area, select a logged signal to apply the requirement to.


If you already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, they appear in the list. Select the corresponding check-box.

If you haven’t selected a signal to log:

- a** Click . A window opens where you specify the logged signal.
- b** In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.

Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- e Select the check-box corresponding to the signal and click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the signal bound also appears in the Design Optimization tool window.

---

**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 3-66.

---




Alternatively, you can use the Check Against Reference block to specify a reference signal to track.


### See Also

“Optimize Controller Parameters to Track Reference Signal (GUI)”

## Specifying Custom Requirements

To specify custom requirements such as minimizing system energy:

- 1 In the Design Optimization tool, select **Custom Requirement** in the **New** drop-down list. A window opens where you specify the custom requirement.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the requirement type using the **Type** list.
- 4 Specify the name of the function that contains the custom requirement in the **Function** box. The field must be specified as a function handle using `@`. The function must be on the MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is `myCustomRequirement`.

- 5 (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design software), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

- Apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, it appears in the list. Select the corresponding check-box.

If you haven’t selected a signal to log:

- a Click . A window opens where you specify the logged signal.
- b In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.


Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- Apply this requirement to a linear system.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

## 6 Click OK.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.

## See Also

- “Design Optimization to Meet a Custom Objective Using the GUI” on page 3-103
- “Design Optimization to Meet Custom Signal Requirements (GUI)” on page 3-133

## Editing Design Requirements

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.

#### 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.

**Edit Design Requirement Dialog Box Parameters (Continued)**

<b>Parameter</b>	<b>Found in</b>	<b>Description</b>
<b>Slope (1/s)</b>	Upper and lower time response bounds	Defines the slope, in 1/s, of a constraint segment. It is an alternative method of specifying the magnitude values. Entering a new <b>Slope</b> value changes any previously defined magnitude values.
<b>Final value</b>	Step response bounds	Defines the input level after the step occurs.
<b>Rise time</b>	Step response bounds	Defines a constraint segment for a particular rise time.
<b>% Rise</b>	Step response bounds	The percentage of the step's range used to describe the rise time.
<b>Settling time</b>	Step response bounds	Defines a constraint segment for a particular settling time.
<b>% Settling</b>	Step response bounds	The percentage of the final value that defines the settling region used to describe the settling time.
<b>% Overshoot</b>	Step response bounds	
<b>% Undershoot</b>	Step response bounds	Defines the constraint segments for a particular percent undershoot.

## Specifying Frequency Domain Design Requirements (GUI)

### In this section...

“Specifying Lower Bounds on Gain and Phase Margin” on page 4-20

“Specifying Piecewise-Linear Lower and Upper Bounds on Frequency Response” on page 4-22

“Specifying Bound on Closed-Loop Peak Gain” on page 4-24

“Specifying Lower Bound on Damping Ratio” on page 4-26

“Specifying Upper and Lower Bounds on Natural Frequency” on page 4-28

“Specifying Upper Bound on Approximate Settling Time” on page 4-30

“Specifying Piecewise-Linear Upper and Lower Bounds on Singular Values” on page 4-32

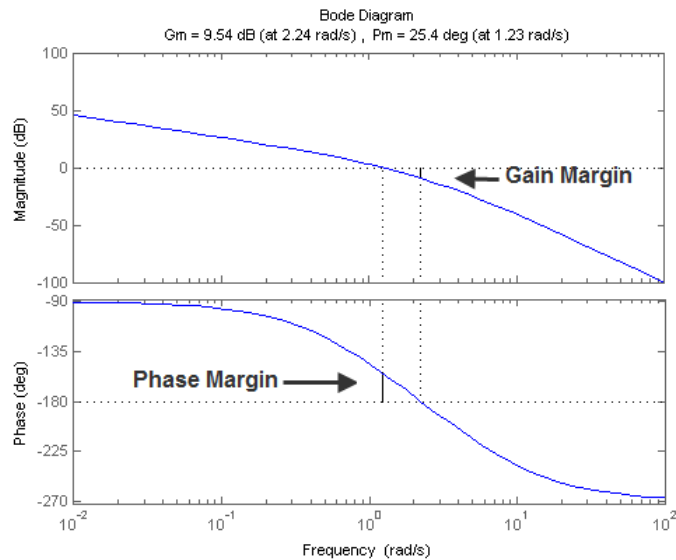
“Specifying Step Response Characteristics” on page 4-35

“Specifying Custom Requirements” on page 4-38

### Specifying Lower Bounds on Gain and Phase Margin

To specify lower bounds on the gain and phase margin of a linear system:

- 1** In the Design Optimization tool, select **Gain and Phase Margin** in the **New** list. A window opens where you specify lower bounds on the gain and phase margin of your linear system.
- 2** Specify a requirement name in **Name**.
- 3** Specify bounds on the gain margin or phase margin, or both.



- **Gain margin** — Amount of gain increase or decrease required to make the loop gain unity at the frequency where the phase angle is  $-180^\circ$ .
- **Phase margin** — Amount of phase increase or decrease required to make the phase angle  $-180^\circ$  when the loop gain is 1.0


To specify a lower bound on the gain margin or phase margin, or both, select the corresponding check box and enter the lower bound value.

- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

#### 5 Click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the requirement also appears in the Design Optimization tool window.

#### 6 (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Gain and Phase Margins block to specify bounds on the gain and phase margin. (Requires Simulink Control Design.)


## Specifying Piecewise-Linear Lower and Upper Bounds on Frequency Response

To specify upper or lower bounds on the magnitude of a system response:

- 1 In the Design Optimization tool, select **Bode Magnitude** in the **New** list. A window opens where you specify the lower or upper bounds on the magnitude of the system response.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the requirement type using the **Type** list.
- 4 Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns.



**5** Insert or delete bound edges.

Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

**6** In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a** Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

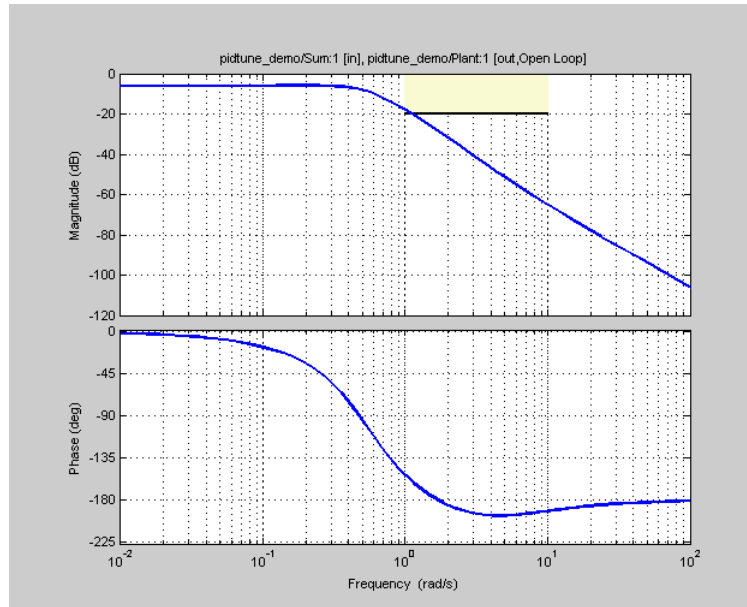
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

**7** Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window. A graphical display of the requirement also appears in the Design Optimization tool window.



**8** (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Bode Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

## Specifying Bound on Closed-Loop Peak Gain

To specify an upper bound on the closed-loop peak response of a system:


- 1** In the Design Optimization tool, select **Closed-Loop Peak Gain** in the **New** list. A window opens where you specify an upper bound on the closed-loop peak gain of the system.
- 2** Specify a requirement name in the **Name** box.

- 3** Specify the upper bound on the closed-loop peak gain in the **Closed-Loop peak gain** box.
- 4** In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a** Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

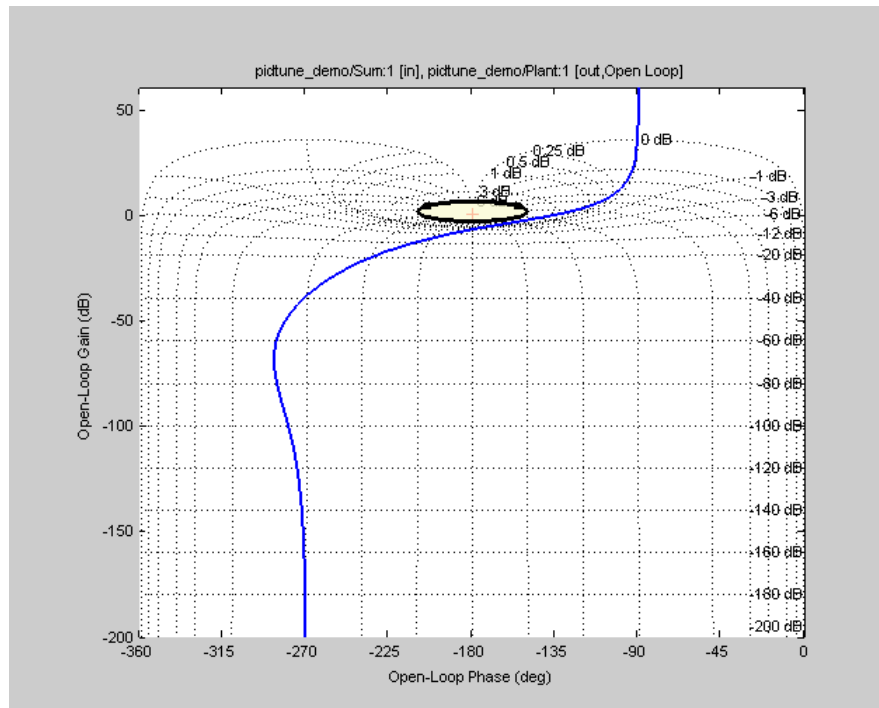
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5** Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window. A graphical display of the requirement also appears in the Design Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Nichols Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

## Specifying Lower Bound on Damping Ratio

To specify a lower bound on the damping ratio of the system:


- 1 In the Design Optimization tool, select **Damping Ratio** in the **New** list. A window opens where you specify an upper bound on the damping ratio of the system.

- 2** Specify a requirement name in the **Name** box.
- 3** Specify the lower bound on the damping ratio in the **Damping ratio** box.
- 4** In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a** Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

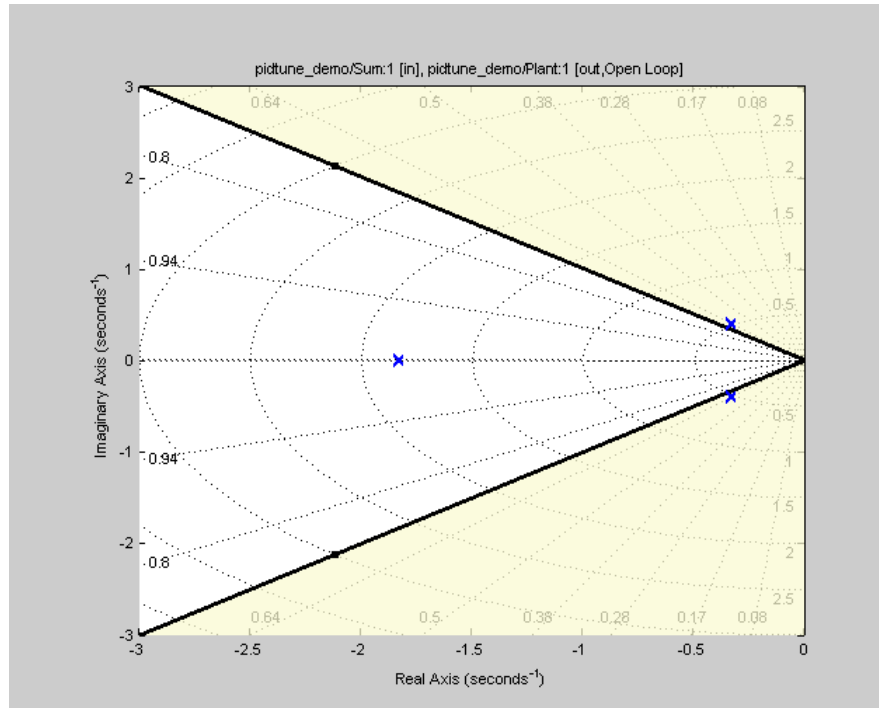
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5** Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the damping ratio. (Requires Simulink Control Design.)

## Specifying Upper and Lower Bounds on Natural Frequency

To specify a bound on the natural frequency of the system:


- 1 In the Design Optimization tool, select **Natural Frequency** in the **New** list. A window opens where you specify a bound on the natural frequency of the system.

- 2 Specify a requirement name in the **Name** box.
- 3 Specify a lower or upper bound on the natural frequency in the **Natural frequency** box.
- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

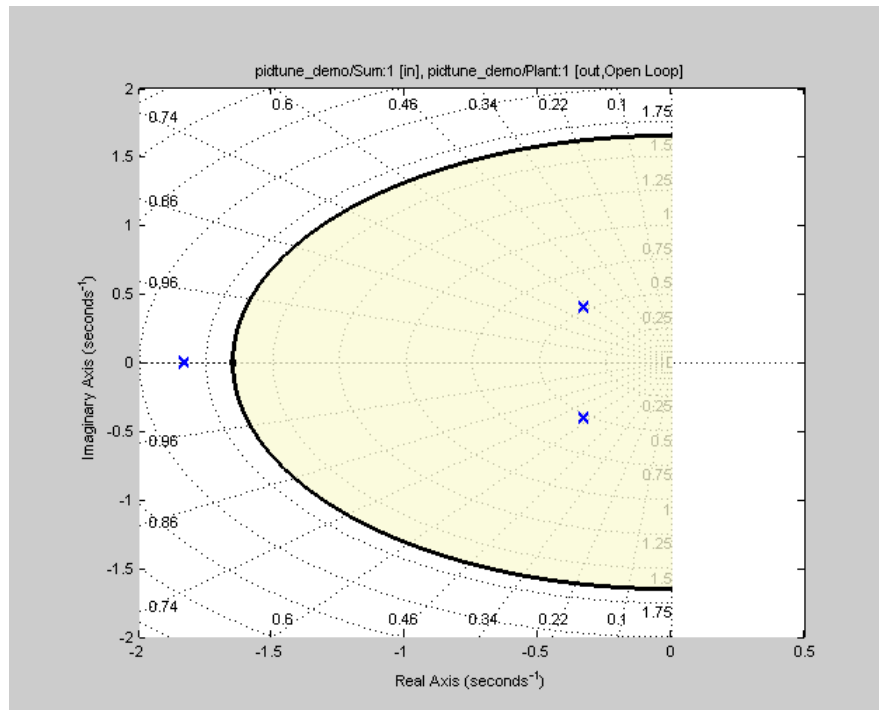
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5 Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.



**6** (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the natural frequency. (Requires Simulink Control Design.)

## Specifying Upper Bound on Approximate Settling Time

To specify an upper bound on the approximate settling time of the system:

- 1** In the Design Optimization tool, select **Settling Time** in the **New** list. A window opens where you specify an upper bound on the approximate settling time of the system.




- 2 Specify a requirement name in the **Name** box.
- 3 Specify the upper bound on the approximate settling time in the **Settling time** box.
- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

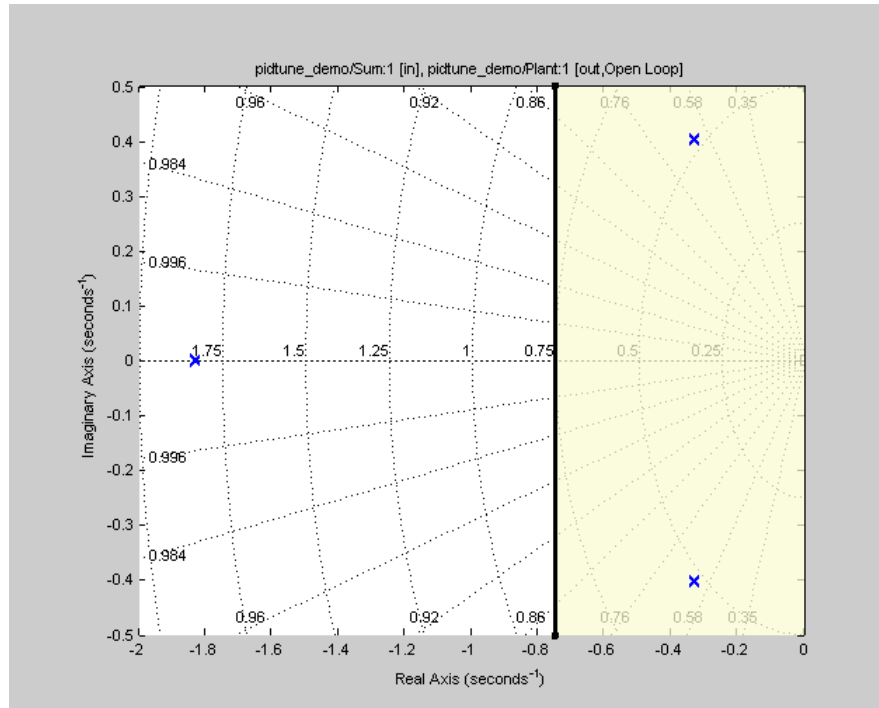
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5 Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.



6 (Optional) In the graphical display, you can:


- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Pole-Zero Characteristics block to specify the approximate settling time. (Requires Simulink Control Design.)

## Specifying Piecewise-Linear Upper and Lower Bounds on Singular Values

To specify piecewise-linear upper and lower bounds on the singular values of a system:

- 1 In the Design Optimization tool, select **Singular Values** in the **New** list. A window opens where you specify the lower or upper bounds on the singular values of the system.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the requirement type using the **Type** list.
- 4 Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns, respectively.
- 5 Insert or delete bound edges.

Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

- 6 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

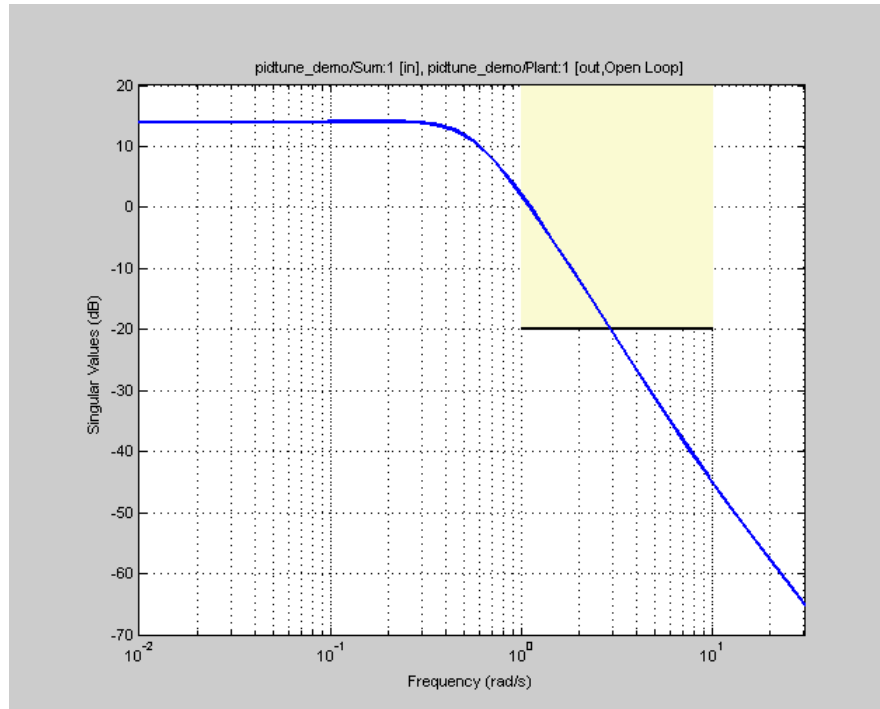
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

**7** Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.



**8** (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Singular Value Characteristics block to specify bounds on the singular value. (Requires Simulink Control Design.)

## Specifying Step Response Characteristics

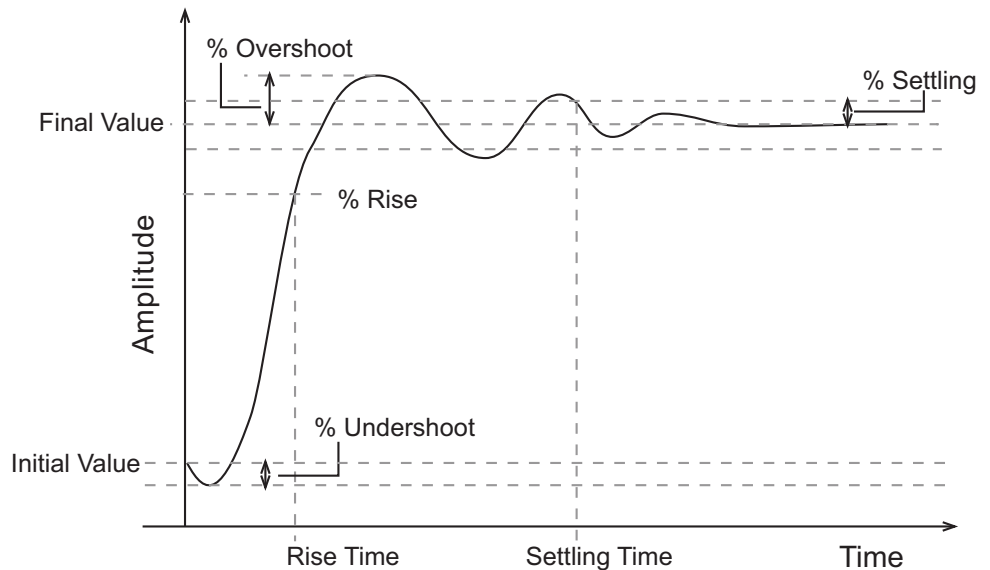
To specify step response characteristics:

- 1 You can apply this requirement to either a signal or a linearization of your model.

In the Design Optimization Tool, click **New**. To apply this requirement to a signal, select the **Step Response Envelope** entry in the **New Time Domain Requirement** section of the **New** list. To apply this requirement to a linearization of your model, select the **Step Response Envelope** entry in the **New Frequency Domain Requirement** section of the **New** list. The latter option requires Simulink Control Design software.

A window opens where you specify the step response requirements on a signal, or system.

- 2 Specify a requirement name in the **Name** box.
- 3 Specify the step response characteristics:



- **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
- **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.

#### 4 Specify the signals or systems to be bound.


You can apply this requirement to a model signal or to a linearization of your Simulink model (requires Simulink Control Design software).

- Apply this requirement to a model signal:

In the **Select Signals to Bound** area, select a logged signal to which you will apply the requirement.

If you have already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, it appears in the list. Select the corresponding check-box.


If you haven't selected a signal to log:

- Click . A window opens where you specify the logged signal.

- b In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.

Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.


- Apply this requirement to a linear system.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.

- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

#### **5** Click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the requirement also appears in the Design Optimization tool window.

Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.

#### **See Also**

“Optimize Controller Parameters to Meet Step Response Requirements (GUI)”


## **Specifying Custom Requirements**

To specify custom requirements such as minimizing system energy:

- 1** In the Design Optimization tool, select **Custom Requirement** in the **New** drop-down list. A window opens where you specify the custom requirement.
- 2** Specify a requirement name in the **Name** box.
- 3** Specify the requirement type using the **Type** list.
- 4** Specify the name of the function that contains the custom requirement in the **Function** box. The field must be specified as a function handle using



@. The function must be on the MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is `myCustomRequirement`.

**5** (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design software), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

- Apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, it appears in the list. Select the corresponding check-box.

If you haven’t selected a signal to log:

- a Click . A window opens where you specify the logged signal.
- b In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.

Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- Apply this requirement to a linear system.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click



to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

#### **6** Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.

#### **See Also**

- “Design Optimization to Meet a Custom Objective Using the GUI” on page 3-103
- “Design Optimization to Meet Custom Signal Requirements (GUI)” on page 3-133

## Design Variables (GUI)

### In this section...

“Specifying Design Variables” on page 3-54

“Specifying Independent Parameters” on page 3-55

“Specify Independent Parameters to Optimize” on page 3-56

### Specifying Design Variables

Before running the optimization, you must define the model parameters to optimize. These parameters form the *design variable set* for optimization. By tuning these parameters, Simulink Design Optimization software attempts to make the signals meet the requirements.

Simulink Design Optimization software optimizes the response signals of the model by varying the 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 **New** in the **Design Variable Set** drop-down list of the Design optimization tol.

The design variables can be scalar, vector, matrix or an expression that evaluates to one of these values.

- 1 Select **New** in the **Design Variables Set** drop-down list.

A window opens where you specify design variables. All model parameters are displayed in this window.

- 2 Select one or more parameter names and click



to add the selected parameters to a design variables set.

---

**Note** You can add the same parameter to multiple design variable sets.

---

### 3 (Optional) Specify design variable settings.

Setting	Description	Default
<b>Variable</b>	The name of the parameter.	Not an editable field
<b>Value</b>	Value of the model parameter. This value is used by the optimization method as the initial value and is modified during optimization.	Current value of the parameter in the model. If you edit this column, click <b>Update model variable values</b> to update the values in the model.
<b>Minimum</b>	The minimum value or lower bound for the parameter. You can edit this field to provide an alternate minimum value.	- Inf
<b>Maximum</b>	The maximum value or upper bound for the parameter. You can edit this field to provide an alternate maximum value.	Inf
<b>Scale</b>	During optimization, the design variables are scaled, or normalized, by dividing their current value by a scale value. You can edit this field to provide an alternate scaling factor.	1

The check-box indicates whether the parameter is selected as an design variable in the set. 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).

Expand **Variable Detail** to see the block in the model that contains this parameter.

**4** Click **OK** to create a design variable set.

## 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.

- 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 dialog box when you select design or uncertain variables. Add these parameters to a design variables set to tune them during response optimization.

---

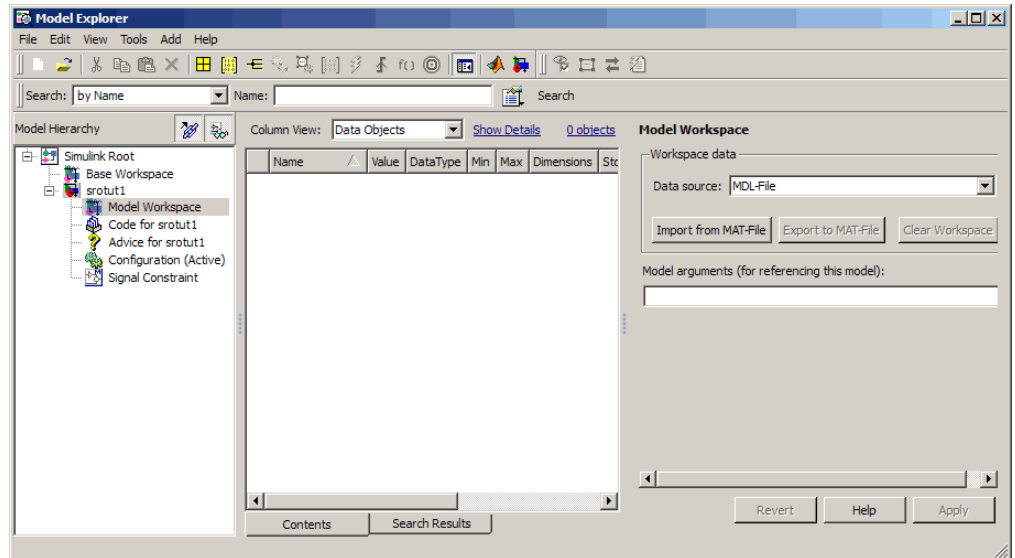
**Caution** Avoid adding independent parameters together with their corresponding dependent parameters to the lists of tuned and uncertain parameters. Otherwise, the optimization could give incorrect results. For example, when a parameter  $x$  depends on the parameters  $a$  and  $b$ , avoid adding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

---

### Specify Independent Parameters to Optimize

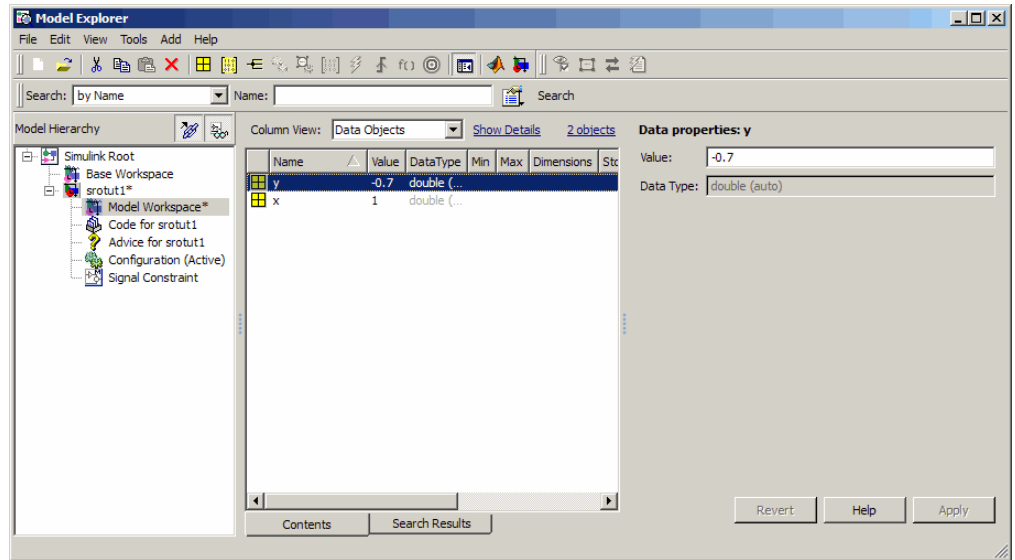
Assume that the parameter `Kint` in the model `srotut1` is related to the parameters  $x$  and  $y$  according to the relationship  $Kint=x+y$ . Also assume that the initial values of  $x$  and  $y$  are 1 and -0.7, respectively. To tune  $x$  and  $y$  instead of `Kint`, first define these parameters 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 design or uncertain parameter, `x` and `y` appear in the dialog box.

## General Options

### In this section...

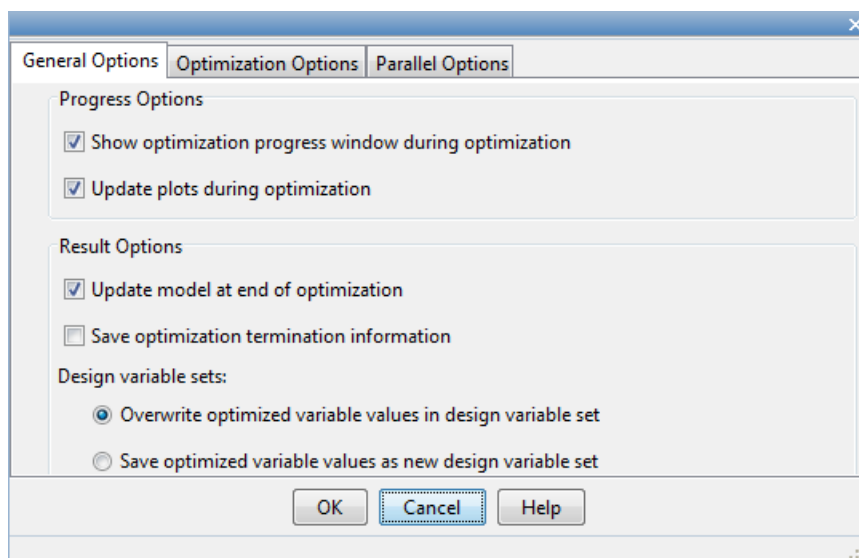
“Accessing General Options” on page 3-60

“Progress Options” on page 3-60

“Result Options” on page 3-61

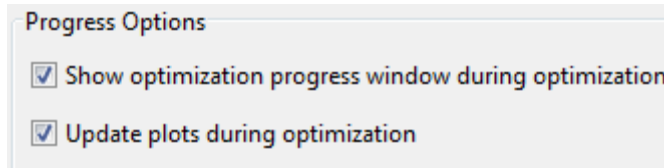
## Accessing General Options

You can set optimization progress and result options. To set these options, click **Options** in the Design Optimization tool. A window opens. Select the **General Options** tab.



## Progress Options

You can specify options related to optimization progress using the options in the **Progress Options** area.



- **Show optimization progress window during optimization**

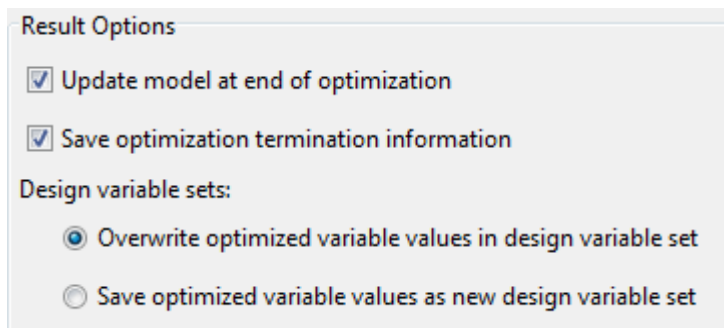
Opens an optimization progress window during optimization. The window displays information such as constraint violations and cost function if **Display level** is **Iteration**. The window is updated at the end of the optimization with the termination information such as whether the optimization converged.

- **Update plots during optimization**

Updates model response and design variable plots at each optimization iteration.

## Result Options

You can specify options for optimization results in the **Result Options** area.



- **Update model at end of optimization**

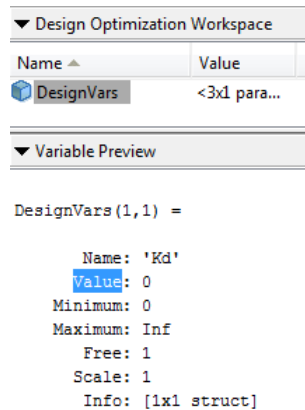
Updates optimized parameter values in the Simulink model after the optimization terminates.

- **Save optimization termination information**

Saves termination information returned by the optimization solver as a variable named `info` in the **Design Optimization Workspace**. `info` is a structure with one or more of the following fields:

- `F` — Optimized cost (objective) value.
- `Cleq` — Optimized nonlinear inequality constraint violations.  
The field appears if the optimization problem includes a nonlinear inequality constraint.  
The value is a  $m \times 1$  vector. Positive values indicate that the constraint has not been satisfied. Check `exitflag` to confirm that the optimization succeeded.
- `Ceq` — Optimized nonlinear equality constraint violations.  
The field appears if the optimization problem includes a nonlinear equality constraint.  
The value is a double  $r \times 1$  vector. Any nonzero values indicate that the constraint has not been satisfied. Check `exitflag` to confirm that the optimization succeeded.
- `Gradients` — Cost and constraint gradients at the optimized parameter values. See “How the Optimization Algorithm Formulates Minimization Problems” on page 3-4 on how the solver computes gradients.  
This field appears if the solver specified in the `Method` property of `sdo.OptimizeOptions` computes gradients.  
The value is a structure.
- `exitflag` — Integer identifying the reason the algorithm terminated. See `fmincon`, `patternsearch` and `fminsearch` for a list of the values and the corresponding termination reasons.
- `iterations` — Number of optimization iterations
- `SolverOutput` — A structure with solver-specific output information. The fields of this structure depends on the optimization solver specified in the `Method` property of `sdo.OptimizeOptions`. See `fmincon`, `patternsearch` and `fminsearch` for a list of solver outputs and their description.

- **Stats** — A structure that contains statistics collected during optimization, such as start and end times, number of function evaluations and restarts.
- **Design variable sets**
  - **Overwrite optimized variable values in design variable set**  
Overwrites the optimized model parameter values in the design variable set variable used in the optimization. You can see the updated values in the Value field.



The screenshot shows the 'Design Optimization Workspace' interface. It features a table with columns 'Name' and 'Value'. Below the table is a 'Variable Preview' section for the variable 'Kd'.

Name	Value
DesignVars	<3x1 para...

```

▼ Design Optimization Workspace
Name ▲      Value
DesignVars  <3x1 para...

▼ Variable Preview

DesignVars(1,1) =
    Name: 'Kd'
    Value: 0
    Minimum: 0
    Maximum: Inf
    Free: 1
    Scale: 1
    Info: [1x1 struct]

```

- **Save optimized variable values as new design variable set**  
Creates a new variable in the **Design Optimization Workspace** that contains the optimized parameter values. To update the model with the optimized parameter values, select the variable in the **Design Variables Set** drop-down list and click **Update model variable values**.

## Optimization Options

### In this section...

“Accessing Optimization Options” on page 3-64

“Selecting Optimization Methods” on page 3-65

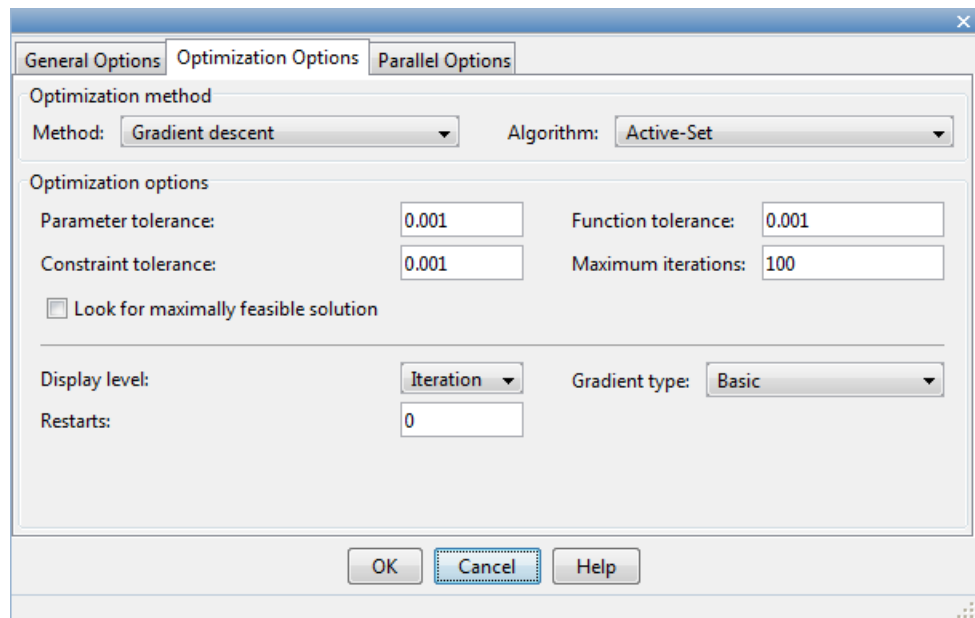
“Selecting Optimization Termination Options” on page 3-66

“Selecting Additional Optimization Options” on page 3-67

## Accessing Optimization Options

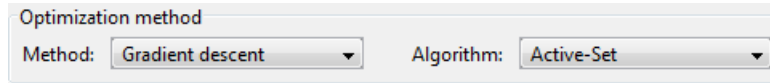
You can set several options for the optimization. These options include the optimization methods and the tolerances the methods use.

To set optimization options, click **Options** in the Design Optimization tool. A window opens. Select the **Optimization Options** tab.



## Selecting Optimization Methods

Both the **Method** and **Algorithm** options in the **Optimization method** area define the optimization method.



Optimization method

Method: Gradient descent Algorithm: Active-Set

The choices for the **Method** option are:

- **Gradient descent** (default) — Uses the Optimization Toolbox function `fmincon` to optimize the response signal subject to the constraints.

The **Algorithm** options for **Gradient descent** are:

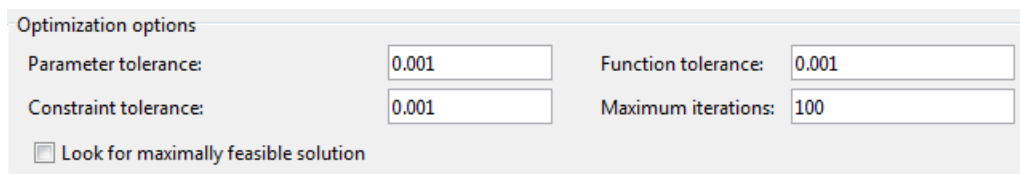
Algorithm Option	Learn More
Active-Set (default)	“fmincon Active Set Algorithm” in the Optimization Toolbox documentation.
Interior-Point	“fmincon Interior Point Algorithm” in the Optimization Toolbox documentation.
Trust-Region-Reflective	“fmincon Trust Region Reflective Algorithm” in the Optimization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

- **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.

For more information on the problem formulations for each optimization method, see “How the Optimization Algorithm Formulates Minimization Problems” on page 3-4.

## Selecting Optimization Termination Options

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



The screenshot shows a panel titled "Optimization options" with the following settings:

Parameter tolerance:	0.001	Function tolerance:	0.001
Constraint tolerance:	0.001	Maximum iterations:	100

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.

---

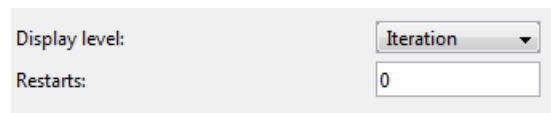
**Note** If selected, the software ignores this option when tracking a reference signal.

---

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 screenshot shows a light gray panel with two controls. The first control is labeled 'Display level:' and has a dropdown menu with 'Iteration' selected. The second control is labeled 'Restarts:' and has a text input field containing the number '0'.

- “Display Level” on page 3-67
- “Restarts” on page 3-68

## Display Level

The **Display level** option specifies the form of the output that appears in the Optimization Progress window. The options are:

- **Iterations** (default) — Displays information after each iteration
- **Off** — Turns off all output display
- **Notify** — Displays output only if the function does not converge
- **Final** — Displays only the final output

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

<b>Method</b>	<b>Function</b>	<b>Output Information</b>
Gradient descent	fmincon	fmincon section of “Function-Specific Headings” in the Optimization Toolbox documentation
Simplex search	fminsearch	fminsearch section of “Function-Specific 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.

## Creating Linearization I/O Sets (GUI)

This example shows how to create a linearization input/output set using the Design Optimization Tool.

You create a linearization input/output set using the Create Linearization I/O Set dialog box. This dialog box may be accessed in two ways:

- In the Design Optimization tool, select **Linearization I/Os** in the **New** drop-down list.
- In a requirement dialog box, in the **Select Systems to Bound** area, click

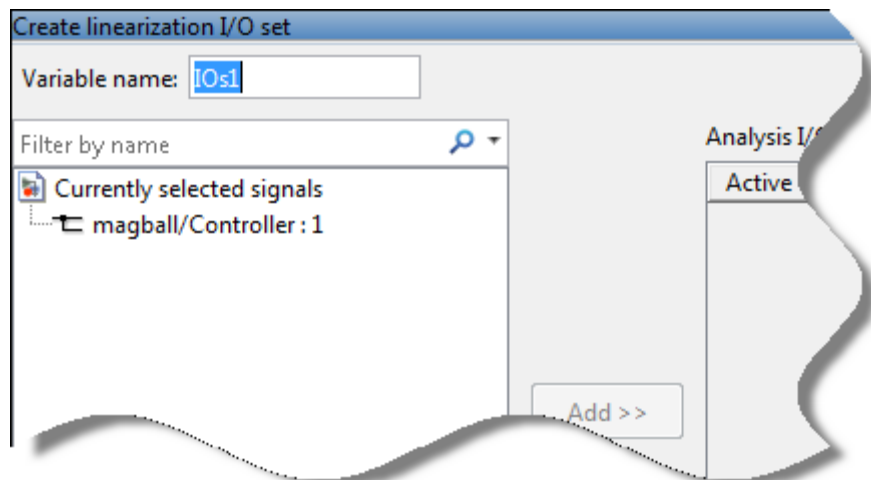


### Create Linearization I/O Set

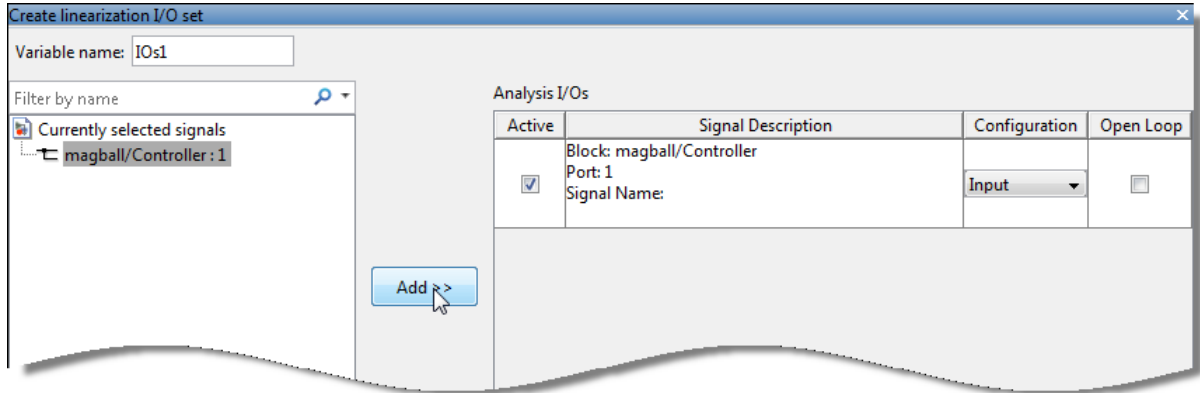
To create a new linearization I/O set:

- 1 In your Simulink model, select a signal that you want to define as a linearization input or output point.

The selected signal appears in the Create linearization I/O set dialog box under **Currently selected signals**.



- 2 In the Create linearization I/O set dialog box, click the signal name under Currently selected signals.
- 3 Click **Add**. The signal appears in the list of Analysis I/Os.



- 4 Select the linearization point type from the **Configuration** drop-down list. For example, if you want the selected signal to be a linearization output point, select **Output**.
- 5 If you want the signal to be a loop opening point, click the **Open Loop** check box.
- 6 Repeat steps 1–5 for any other signals you want to define as linearization I/O points.

---

**Tip** To highlight in the Simulink model the blocks that are included in the linearization specified by the current list of Analysis I/Os, click **Highlight**.

---

- 7 After you define all the signals for the I/O set, enter a name for the I/O set in the **Variable name** box.
- 8 Click **OK**.

## Linearization Options

### In this section...

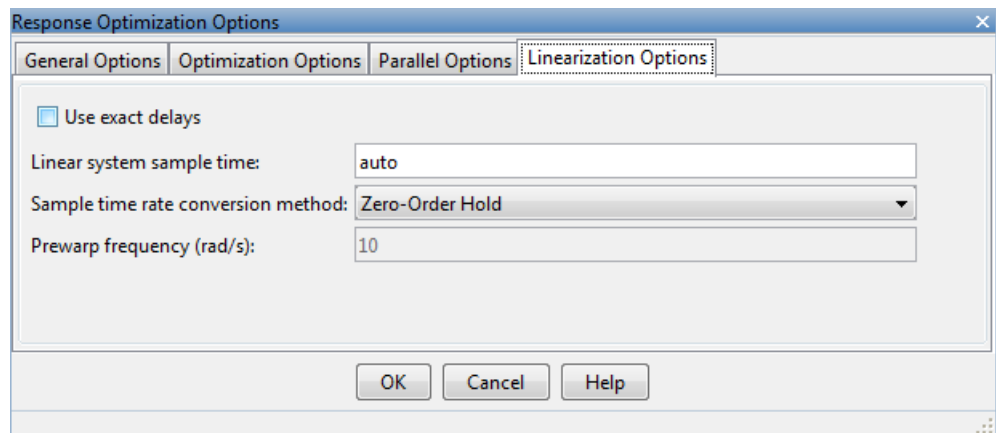
“Accessing Linearization Options ” on page 3-71

“Configuring Linearization Options” on page 3-71

### Accessing Linearization Options

If you have Simulink Control Design, you can set several options for linearization. These options include the linearization methods and the sample time of the linear systems.

To set linearization options, click **Options** in the Design Optimization tool. A window opens. Select the **Linearization Options** tab.



### Configuring Linearization Options

#### Models with Time Delays

Simulink Control Design lets you choose whether to linearize models using exact representation or Pade approximation of continuous time delays. How you treat time delays during linearization depends on your nonlinear model.

Simulink blocks that model time delays are:

- Transport Delay block
- Variable Time Delay block
- Variable Transport Delay block
- Delay block
- Unit Delay block

By default, linearization uses Pade approximation for representing time delays in your linear model.

Use Pade approximation to represent time delays when:

- Applying more advanced control design techniques to your linear plant, such as LQR or H-infinity control design.
- Minimizing the time to compute a linear model.

Specify to linearize with exact time delays for:

- Minimizing errors that result from approximating time delays
- PID tuning or loop-shaping control design methods in Simulink Control Design
- Discrete-time models (to avoid introducing additional states to the model)

The software treats discrete-time delays as internal delays in the linearized model. Such delays do not appear as additional states in the linearized model.

**Specify How Delays are Treated.** To specify whether the linearization should approximate delays or use them exactly, set the **Use exact delays** check box appropriately.

### Linearization Sampling Time

To specify the sampling time of the linearized model, use the **Linear system sample time** box. By default, the software chooses the slowest applicable sampling time. Use 0 to specify a continuous-time linear model.

## Linearization Rate Conversion Method

When you linearize models with multiple sample times, such as a discrete controller with a continuous plant, a rate conversion algorithm generates a single-rate linear model. The rate conversion algorithm affects linearization results.

Rate Conversion Method	When to Use
Zero-Order Hold	Use when you need exact discretization of continuous dynamics in the time-domain for staircase inputs.
Tustin	Use when you need good frequency-domain matching between a continuous-time system and the corresponding discretized system, or between an original system and the resampled system.
Tustin with Prewarping	Use when you need good frequency domain matching at a particular frequency between the continuous-time system and the corresponding discretized system, or between an original system and the resampled system.
Upsampling when possible (Zero-Order Hold, Tustin, and Tustin with Prewarping)	Upsample discrete states when possible to ensure gain and phase matching of upsampled dynamics. You can only upsample when the new sample time is an integer multiple of the sampling time of the original system. Otherwise, the software uses an alternate rate conversion method.

## Plots

In this section...
“Adding Plots in Design Optimization Tool” on page 3-74
“Plotting Current Response” on page 3-74
“Plotting Intermediate Steps” on page 3-74
“Modifying Plot Properties” on page 3-74
“Plot Types” on page 3-76

### Adding Plots in Design Optimization Tool

To create a new plot or to add to an existing plot in the Design Optimization Tool, choose the variable to plot in the **Data to Plot** list. Then select the plot type using the **Add Plot** list. The **Add Plot** list has entries for the supported plot types for the given plot variable.

### Plotting Current Response

To display the current response, click **Plot Current Response** in the **Response Optimization** tab or the **Plot** tab of the Design Optimization tool. The current response appears as a thick line.

### Plotting 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 in the plot and select **Responses > Show Iteration Responses**. The response at an intermediate step is based on parameter values at that intermediate point in the optimization.

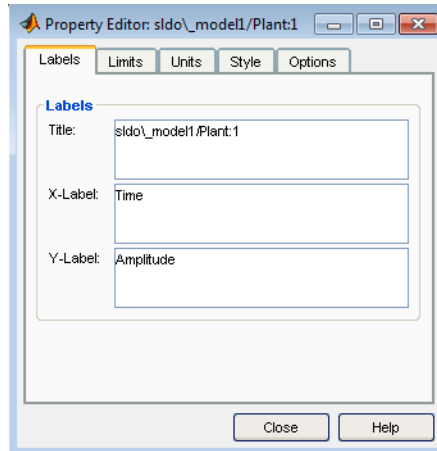
### Modifying Plot Properties

#### Modifying Properties of Response Plots

Right-click the white space in a plot and select **Axes Properties** to open the Property Editor dialog box.



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

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** — Numerical ranges of the  $x$ - and  $y$ - axes

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.

- **Units** — Units where applicable (e.g., rad/s to Hertz). If you cannot customize units, the Property Editor displays that no units are available for the selected plot.
- **Style** — Show a grid, adjust font properties, such as font size, bold and italics, and change the axes foreground color

- **Options** — Change options where applicable. These include peak response, settling time, phase and gain margins, etc. Plot options change with each plot response type. The Property Editor displays only the options that make sense for the selected response plot. For example, phase and gain margins are not available for step responses.

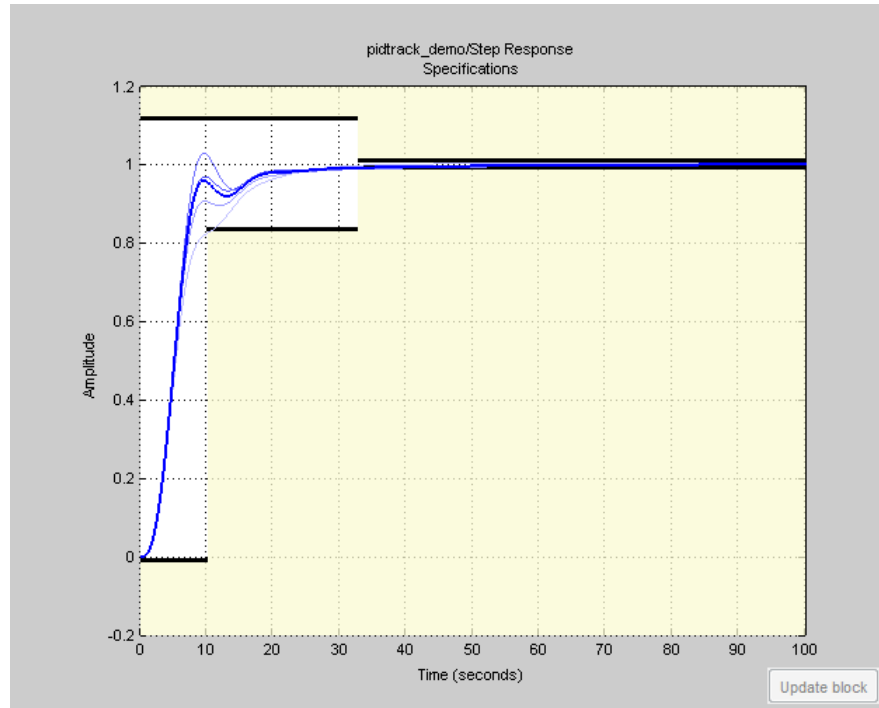
### Plot Types

- “Response Plots” on page 3-76
- “Spider Plots” on page 3-78
- “Iteration Plots” on page 3-79

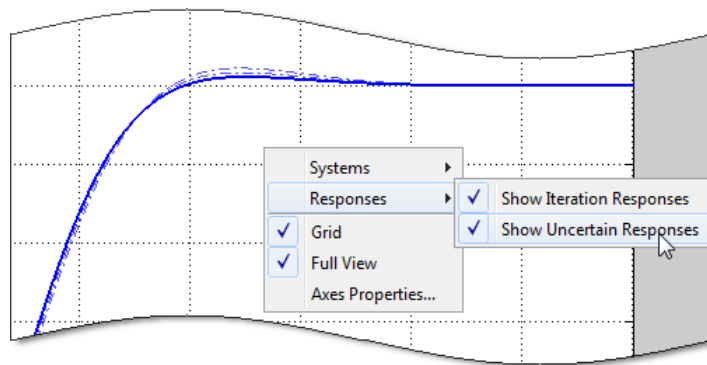
### Response Plots

You can view model signals and the requirements applied to the signal using a response plot. You can also plot the frequency response of a system (requires Simulink Control Design).

The response plot shows the system response as it varies during optimization. You can also view the uncertain system responses in the plot.



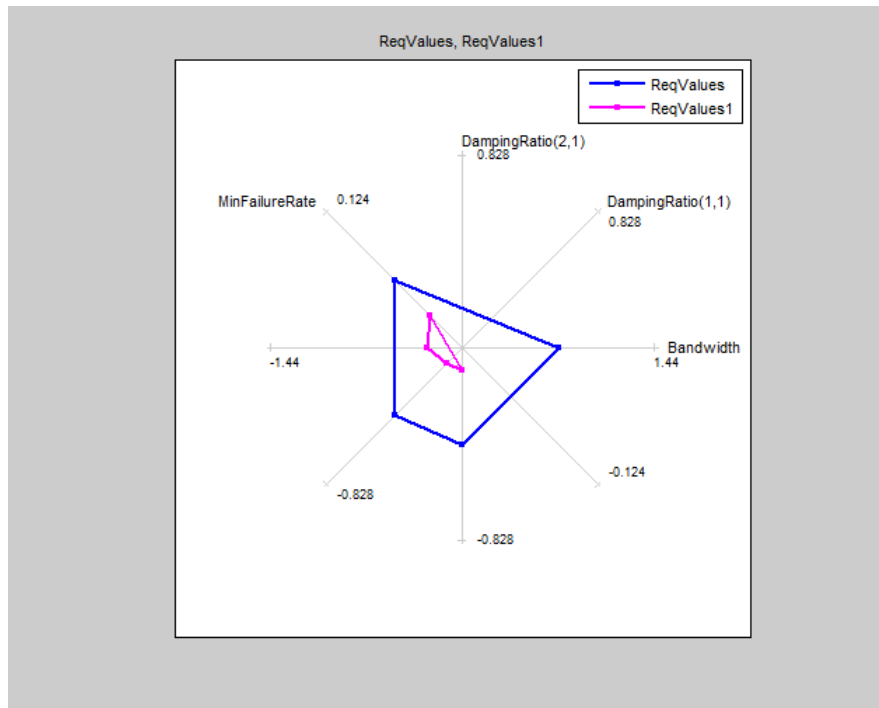
**Tip** To select the responses and systems displayed in a given plot, right-click on the plot and use the **Systems** and **Responses** menu.



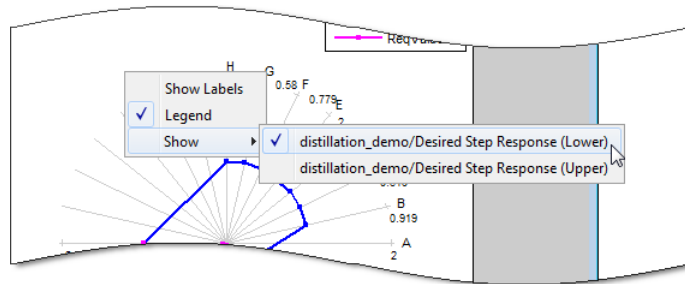
### Spider Plots

You can compare the values of design variable sets or evaluated requirements using a spider plot.

Spider plots depict multivariate data using an axis for each variable. The various axes are arranged clockwise and have a common intersecting point, as this example shows:



**Tip** To view only some of the requirement values in a given plot, right-click on the plot and select the requirements of interest in the **Show** list.

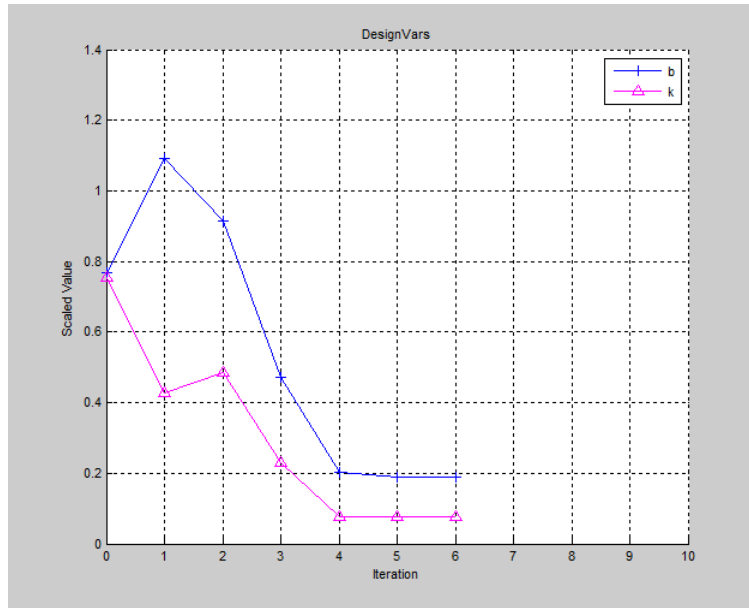


For information on using a spider plot to compare design variables sets or evaluated requirements, see “Compare Requirements and Design Variables Using Spider Plot” on page 3-81

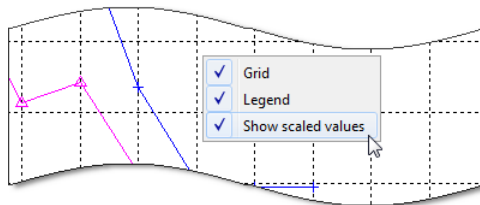
### Iteration Plots

You can plot the values of design variables and requirements as they vary during optimization using an iteration plot.

Iteration plots depict the value of the plot variable(s) for each iteration. The x-axis represents the iteration number, as this example shows:



**Tip** To view scaled values of the plot variable(s), right-click on the plot and select **Show scaled values**.



## Compare Requirements and Design Variables Using Spider Plot

This example shows how to use a spider plot to compare requirement evaluations before and after optimizing the response. You can use a similar procedure to compare the values of sets of design variables.

**Open the Simulink model and load the pre-configured Design Optimization Tool session.**

For this example, which uses a distillation column model, the step response requirements are pre-configured and loaded in the model workspace.

- Open the distillation model.

```
sys = 'distillation_demo';  
open_system(sys)
```

- Open the Design Optimization Tool.

In the Simulink model window, select **Tools > Response Optimization**.

Alternatively, click the Response Optimization GUI with preloaded data block in the model and skip the next step.

- Load the pre-configured Design Optimization Tool session.

Click the **Design Optimization** tab. In the **Open** drop-down list, select **Open from model workspace**. A window opens where you select the Design Optimization Tool session to load. Select `distillation_optim` and click **OK**.

The pre-configured step response requirements are loaded in the Design Optimization Tool.

**Evaluate the value of the requirement before optimization.**

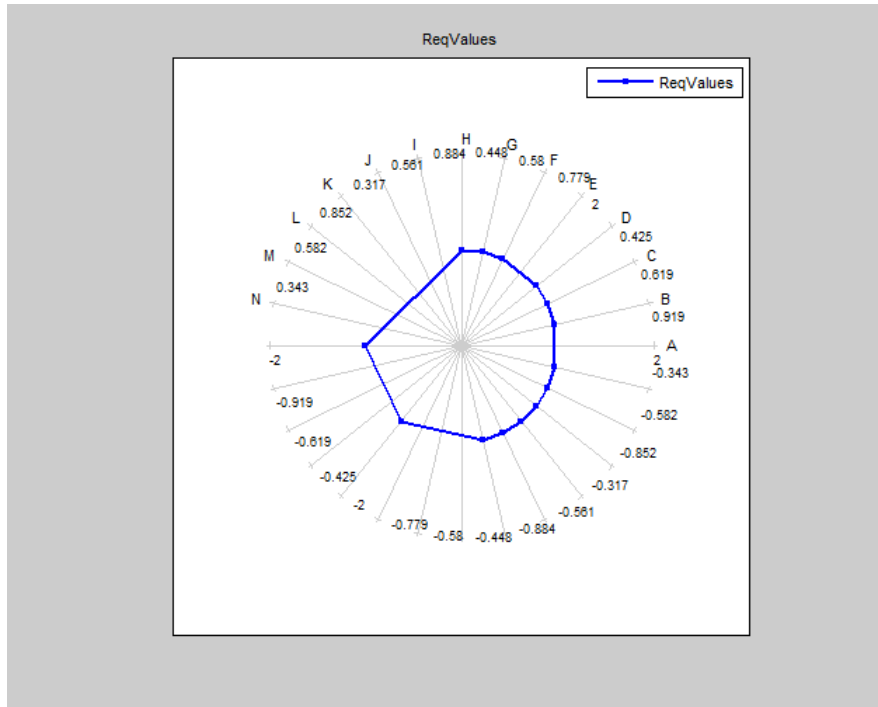
In the **Response Optimization** tab, click **Evaluate Requirements**.

A new variable, `ReqValues`, containing the evaluation of the requirements appears in the **Design Optimization Workspace**.

When optimizing the model response, you create a set of requirements that it must satisfy. If the requirements are violated, meaning that they evaluate to nonnegative values, the design variables must be optimized. After the optimization, you can compare the original requirement value with the requirement evaluated using the optimized design variable values.

**Plot the value of the requirement before optimization using a spider plot.**

- In the **Data to Plot** list, select ReqValues.
- In the **Add Plot** list, select Spider plot.



The plot has an axis for each edge-and-signal combination defined in the `distillation_demo/Desired Step Response` check block. Points on each axis represent the violation for that signal-edge combination and the plot connects these points to form a closed polygon representing the initial design.



Note that some points are negative, representing satisfied constraints, and some positive, representing violated constraints.

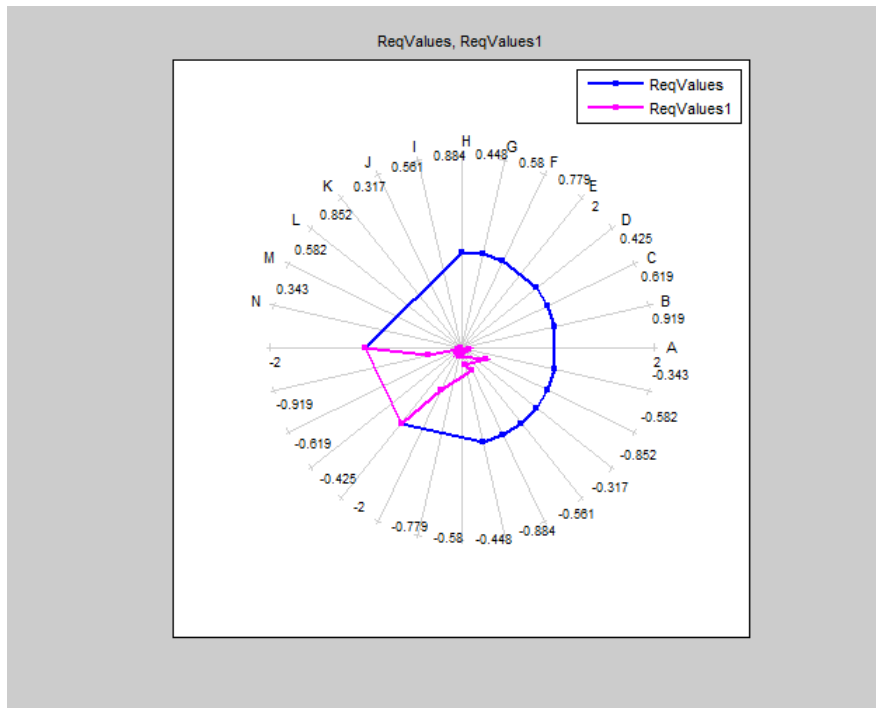
### Optimize the model.

Click **Optimize**.

A new variable, ReqValues1, containing the evaluation of the requirements using the optimized design variables appears in the **Design Optimization Workspace**.

### Compare the requirement values before and after optimization using the spider plot.

- In the **Data to Plot** list, select ReqValues1.
- In the **Add Plot** list, select Spider plot 1.

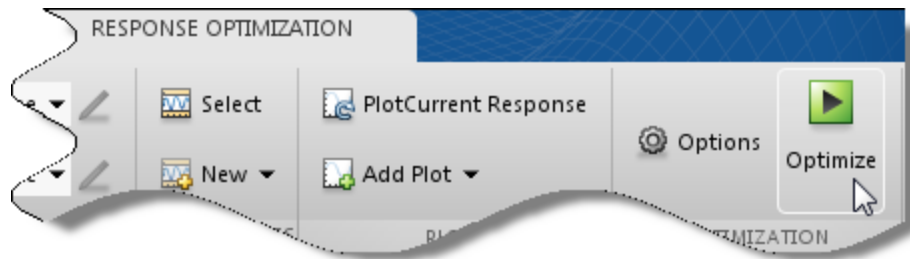


The optimized requirement values, `ReqValues1`, are all negative or zero, indicating that all the constraints are satisfied.

## Run the Optimization (GUI)

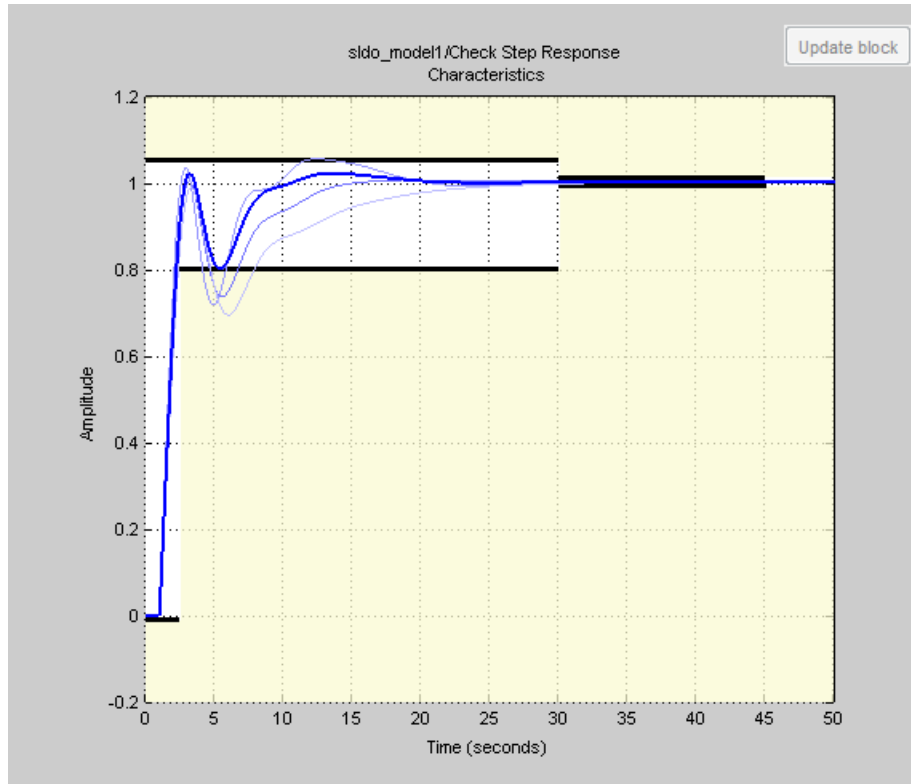
After you specify design requirements and design variables and optionally uncertain variables and optimization settings, you can optimize the design.

Run the optimization by clicking **Optimize** in the Design Optimization tool.



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. To learn more about how the software formulates the optimization problems, see “How the Optimization Algorithm Formulates Minimization Problems” on page 3-4.

Simulink Design Optimization software begins by plotting the initial response. During the optimization, intermediate responses are also plotted. The plot shows the plots of the last five iterations. If you include uncertain variables, 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 design variables and displays the optimization termination information in the optimization progress window. You can click the design variable set name in the **Design Optimization Workspace** 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 values of the parameters as initial values.

---

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 3-65 and the discussion of **Display level** in “Selecting Additional Optimization Options” on page 3-67.

Iter	F-count	f(x)	max constraint
0	5		431.9566
1	12		264.3424
2	19		6.3727
3	26		1.0278
4	33		0.1333
5	40		0.1049
6	48		0.0805
7	55		0.0030
8	62		-1.1153e-06

Optimization started 31-May-2011 16:55:07  
 Optimization converged, 31-May-2011 17:03:36

Optimized variable values written to 'DesignVars' in SDOTOOL workspace  
 'sldo\_model1' updated with optimized values

The Gradient descent optimization method may violate the bounds on parameter values when it cannot satisfy the design requirements and the bounds on parameter values simultaneously. To learn how to troubleshoot this problem, see “Troubleshooting Optimization Results” on page 3-186

---

**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 3-186.

## Design Optimization to Meet Time- and Frequency-Domain Requirements

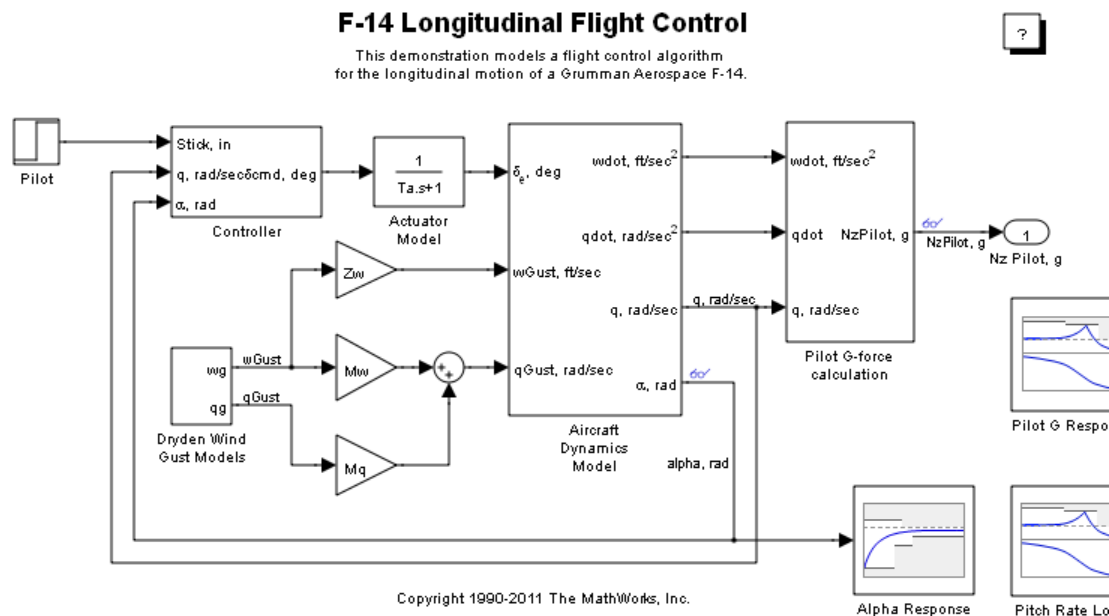
This example shows how to tune a controller to satisfy time- and frequency-domain design requirements using the Design Optimization tool.

The example requires Simulink Control Design™.

### F-14 Longitudinal Flight Control Model

Open the Simulink model.

```
sys = 'sdoF14';
open_system(sys);
```



The F-14 model is based on the Simulink `sldemo_f14` model. The model includes:

- Subsystems to model aircraft dynamics (Aircraft Dynamics Model), wind gusts (Dryden Wind Gust Models), and pilot G-forces (Pilot G-force calculation). For more information on the subsystems, see "F-14 Longitudinal Flight Control".
- A step change applied to the aircraft joystick at 1 second into the simulation that causes the aircraft to pitch upward.

### **Controller Design Problem**

You tune the controller gains to meet the following time- and frequency-domain design requirements:

- Angle-of-attack  $\alpha$  response to a step change in the joystick has a rise time of less than 1 second, less than 1% overshoot, and settles to within 1% of steady state within less than 5 seconds
- Pitch-rate control loop has good tracking below 1 rad/s and 20db noise rejection above 100 rad/s
- Closed loop response from joystick to pilot G-Force is below 0db above 5 rad/s.

These requirements reduce the high frequency G-forces experienced by the pilot in response to joystick changes while still maintaining flight performance.

The model includes the following blocks (from Simulink Design Optimization™ and Simulink Control Design Model Verification libraries):

- Alpha Response specifies the alpha step response requirement.

**Check Step Response Characteristics**  
Assert that the input signal satisfies bounds specified by step response characteristics.

Bounds **Assertion**

Include step response bound in assertion

Step time (seconds):

Initial value:  Final value:

Rise time (seconds):  % Rise:

Settling time (seconds):  % Settling:

% Overshoot:  % Undershoot:

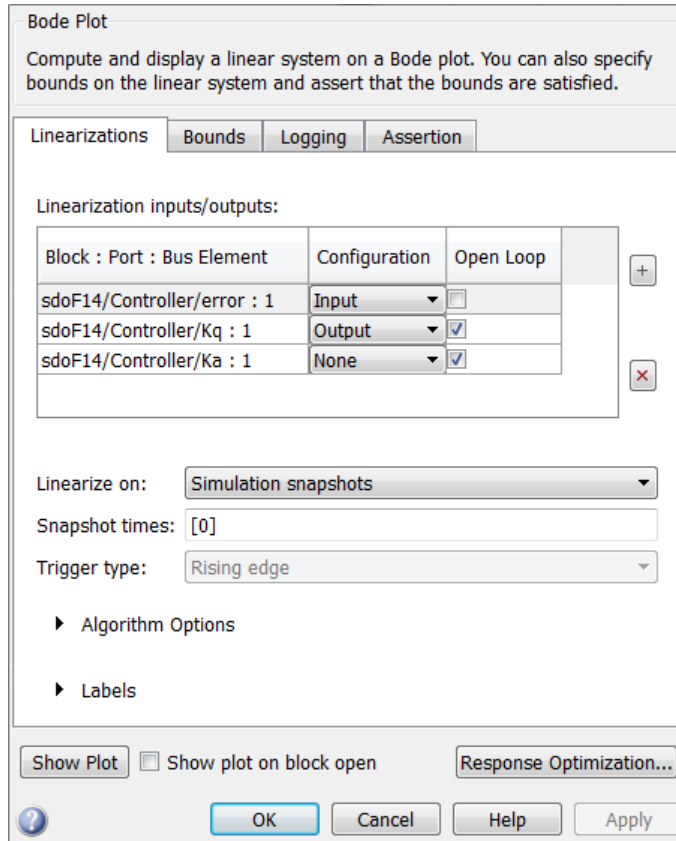
Enable zero-crossing detection

Show plot on block open

- Pitch Rate Loop specifies the pitch-rate performance requirement.

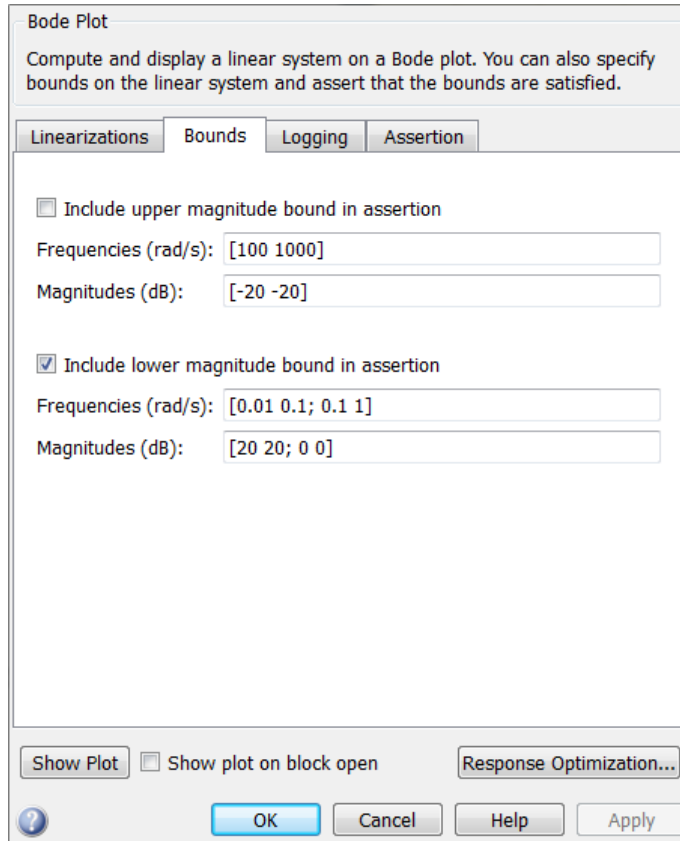
The linearization inputs/outputs are already selected in the **Linearizations** tab. The pitch-rate loop starts from the input of the controller (the controller error signal) and ends at the output of the pitch-rate sensor. The angle-of-attack loop is opened signal so that the block only computes the pitch-rate loop response. The linear system is computed at a simulation time of 0.





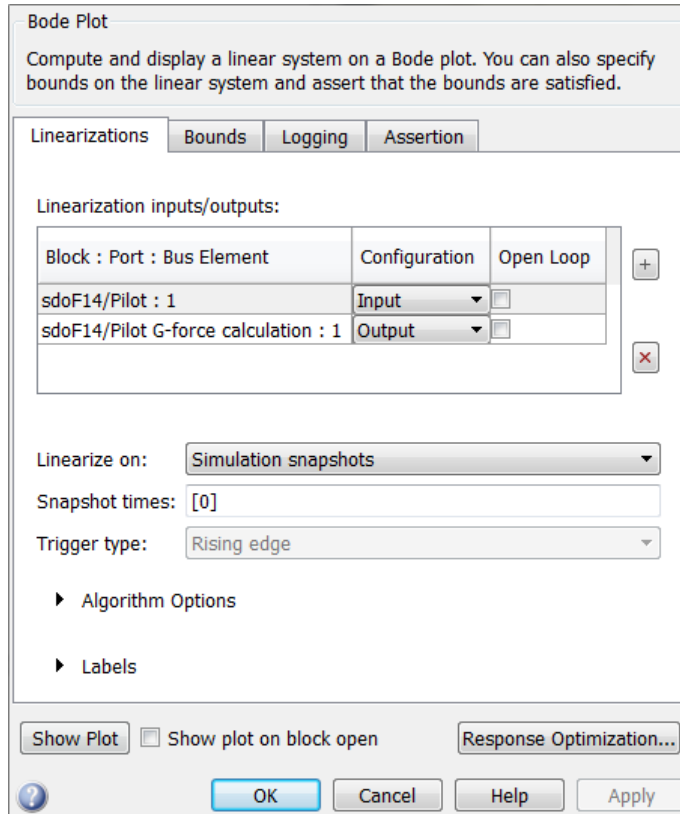
The **Bounds** tab specifies the following pitch-rate loop shape requirements:

- Greater than 20db over the range 0.01 rad/s to 0.1 rad/s
- Greater than 0db over the range 0.1 rad/s to 1 rad/s
- Less than -20db over the range 100 rad/s to 1000 rad/s

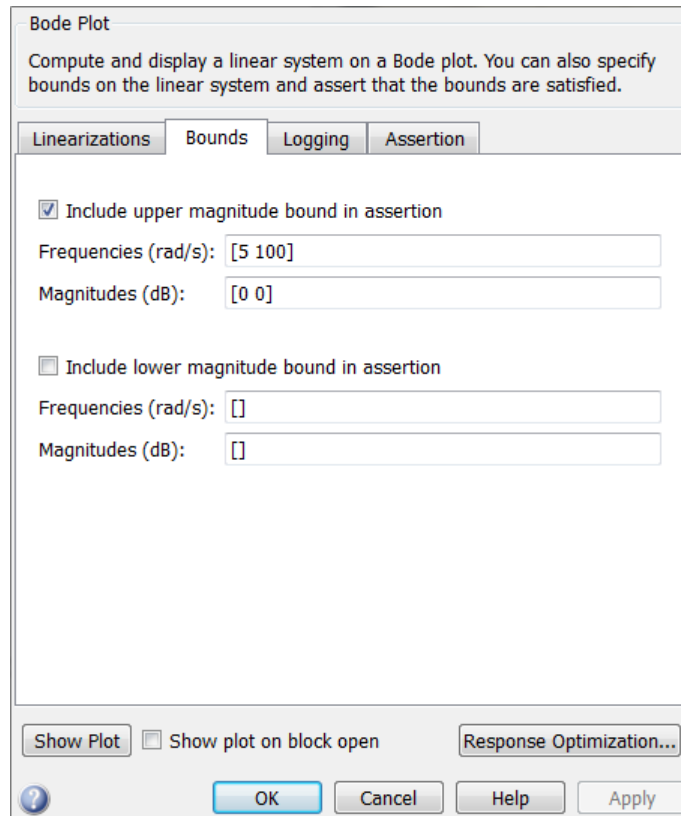


- Pilot G Response specifies the G-force requirement.

The linearization inputs/outputs are already selected in the **Linearizations** tab. The linear system is computed at a simulation time of 0.



The **Bounds** tab specifies the G-force requirements of less than 0db over the range 5 rad/s 100 rad/s.



#### Open the Design Optimization Tool

Open the Design Optimization tool to configure and run design optimization problems interactively. Click **Response Optimization** on the Block Parameters dialog of Alpha Response, Pitch Rate Loop or Pilot G Response block. Alternatively, type `sdotool('sdoF14')`. To show multiple requirement plots at the same time, use the plot layout widgets at the top-right of the tool.

The screenshot displays the Design Optimization tool interface, which is divided into two main tabs: DESIGN OPTIMIZATION and RESPONSE OPTIMIZATION. The DESIGN OPTIMIZATION tab is active, showing a toolbar with buttons for 'Design Variables Set' (set to 'None'), 'Uncertain Variables Set' (set to 'None'), 'Select', 'New', 'Evaluate Requirements', 'Data to Plot' (set to 'No data selected'), 'Add Plot', and 'Plot Current Response'. Below the toolbar, the interface is organized into three sections: VARIABLES, REQUIREMENTS, and PLOTS. The Data Browser is open, showing a search bar for workspace variables. It is divided into four sections: MATLAB Workspace, Model Workspace (sdoHydraulicCylinder), Design Optimization Workspace, and Variable Preview. The Model Workspace section contains a table of variables:

Name	Value
A	0.0100
Ac	1.0000e-03
Beta	700000000
C1	2.0000e-08

The tool detects the requirements specified in the Model Verification blocks and automatically includes them as requirements to satisfy.

### **Specify Design Variables**

Specify the following model parameters as design variables for optimization:

- Controller gains  $K_i$  and  $K_f$
- Pitch-rate sensor gain  $K_q$
- Alpha sensor gain  $K_a$

In the **Design Variables Set** drop-down list, select **New**. A dialog to select model parameters for optimization opens.

Select  $K_i$ ,  $K_f$ ,  $K_q$  and  $K_a$ . Click << to add the selected parameters to the design variables set.

Specify minimum and maximum gain values, the  $K_i$  and  $K_f$  values must remain negative while  $K_a$  and  $K_q$  must remain positive.

Press **Enter** after you enter the values.

Create Design Variables set: DesignVars

<input checked="" type="checkbox"/>	Variable	Value	Minimum	Maximum	Scale
<input checked="" type="checkbox"/>	Ka	0.677	0	Inf	1
<input checked="" type="checkbox"/>	Kf	-1.746	-Inf	0	1
<input checked="" type="checkbox"/>	Ki	-3.864	-Inf	0	1
<input checked="" type="checkbox"/>	Kq	0.8156	0	Inf	1

Update model variable values

Variable Detail

Specify expression (e.g., s.x or

Click **OK**. A new variable DesignVars appears in the **SDOTOOL Workspace**.

#### Evaluate the Initial Design

Click **Plot Current Response** to simulate the model and check how well the initial design satisfies the design requirements.

DESIGN OPTIMIZATION
RESPONSE OPTIMIZATION
PLOT

Design Variables Set: DesignVars

Uncertain Variables Set: None

Select Evaluate Requirements

New

Data to Plot: No data selected

Add Plot Plot Current Response

VARIABLES
REQUIREMENTS
PLOTS

*Data Browser*

Search workspace variables

▼ MATLAB Workspace

Name	Value
A	[8;0;0;0;0;...]
Ap	[-1.0189,0....]
B	[984;0;0;-0....]
Beamwidth	0.1745

▼ Model Workspace (sdoF14)

Name	Value
Beta	426.4352
Gamma	0.0100
Ka	0.6770
Kf	-1.7460

▼ Design Optimization Workspace

Name	Value
DesignVars	<4x1 para...>

▼ Variable Preview

Time plot 1 x Bode plot 1 x Bode plot 2 x

**Time plot 1**

sdoF14/Alpha Response

**Bode plot 2**

sdoF14/Pitch Rate Loop



The plots indicate that the current design does not satisfy the pilot G-force requirement and the alpha step response overshoot requirement is violated.

### **Optimize the Design**

Create a plot to display how the controller variables are modified during the optimization. In the **Add plot** drop-down list, select **DesignVars**, which contains the optimization design variables  $K_i$ ,  $K_f$ ,  $K_q$  and  $K_a$ .

DESIGN OPTIMIZATION      RESPONSE OPTIMIZATION      PLOT

Design Variables Set: DesignVars    Select    Evaluate Requirements    Data to Plot: DesignVars    Add Plot    Plot Current Response

Uncertain Variables Set: None    New    REQUIREMENTS    PLOTS

VARIABLES    REQUIREMENTS    PLOTS

Data Browser

Search workspace variables

▼ MATLAB Workspace

Name	Value

▼ Model Workspace (sdoHydraulicCylinder)

Name	Value
A	0.0100
Ac	1.0000e-03
Beta	700000000
C1	2.0000e-08

▼ Design Optimization Workspace

Name	Value
DesignVars	<2x1 para...>
MaxPressure	<1x1 sdo.r...>
MinimizeAC	<1x1 sdo.r...>
PistonPosition	<1x1 Simul...>

▼ Variable Preview

Time plot 1    Time plot 2    Iteration plot 1    Iteration plot 2

Time plot 1

sdoHydraulicCylinder/Cylinder Assembly:1

Time plot 2

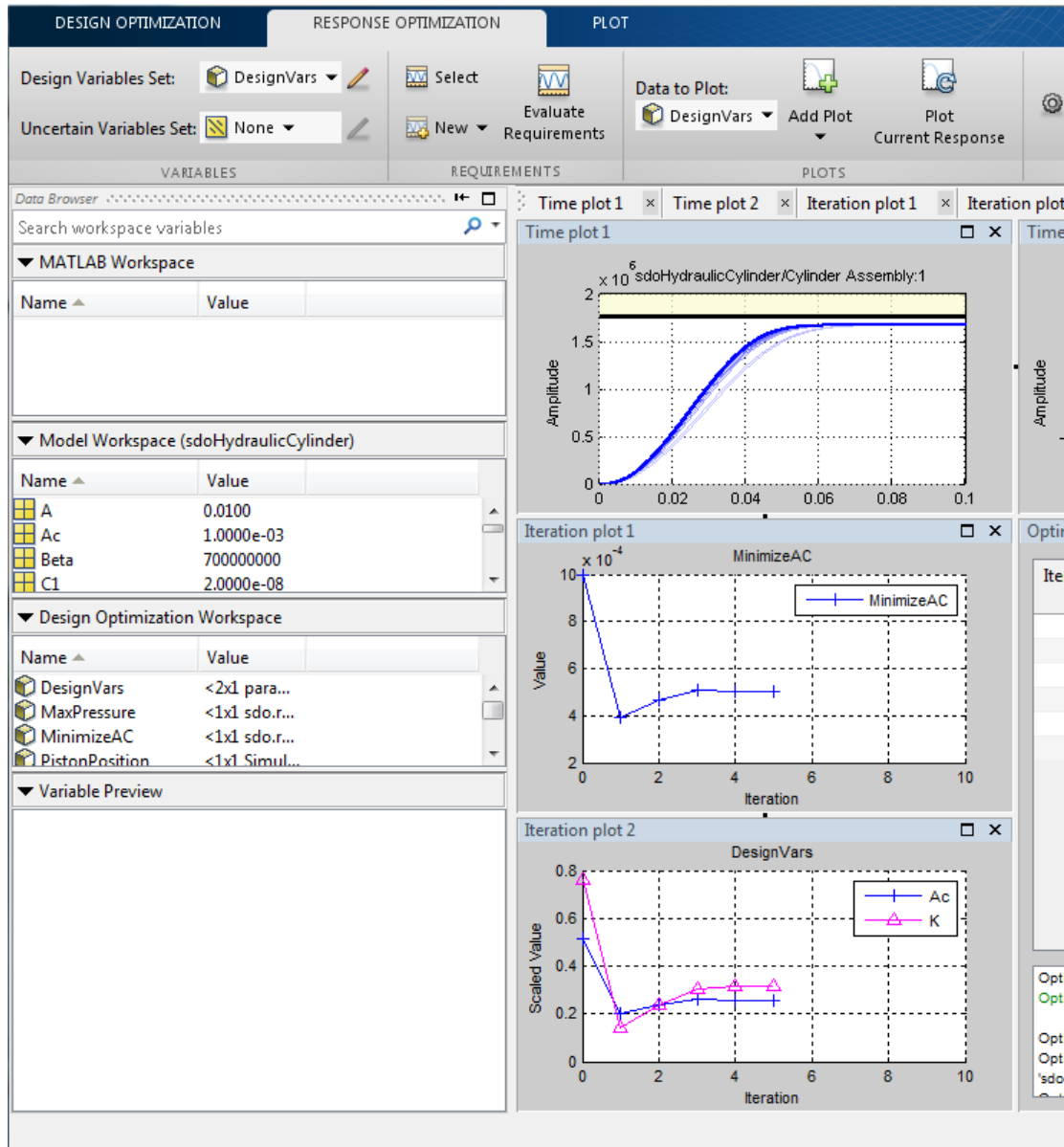
sdoHydr...

Iteration plot 1

There is no data for MinimizeAC, run the optimization

Iteration plot 2

Click **Optimize**.



To load a pre-configured file and run the optimization, click **Open** in the **Design Optimization** tab and select `sdoF14_sdosession.mat`. Alternatively load the project by typing:

```
>> load sdoF14_sdosession
```

```
>> sdotool(SDOSessionData)
```

The optimization progress window updates at each iteration and shows that the optimization converged after 5 iterations.

The Alpha Response and Pilot G Response plots indicate that the design requirements are satisfied. The DesignVars plot shows that the controller gains converged to new values.

To view the optimized design variable values, click **DesignVars** in the **SDOTOOL Workspace**. The optimized values of the design variables are automatically updated in the Simulink model.

```
% Close the model  
setOption(sdotool('sdoF14'),'NoPromptClose',true)  
bdclose('sdoF14')
```

## Design Optimization to Meet a Custom Objective Using the GUI

This example shows how to optimize a design to meet a custom objective using the Design Optimization tool. You optimize the cylinder parameters to minimize the cylinder geometry and satisfy design requirements.

### Hydraulic Cylinder Model

The hydraulic cylinder model is based on the Simulink model `sldemo_hydcyl`. The model includes:

- Pump and Cylinder Assembly subsystems. For more information on the subsystems, see "Single Hydraulic Cylinder Simulation".
- A step change applied to the cylinder control valve orifice area that causes the cylinder piston position to change.

### Hydraulic Cylinder Design Problem

You tune the cylinder cross-sectional area and piston spring constant to meet the following design requirements:

- Ensure that the piston position has a step response rise time of less than 0.04 seconds and setting time of less than 0.05 seconds.
- Limit the maximum cylinder pressures to  $1.75e6$  N/m.
- Minimize the cylinder cross-sectional area.

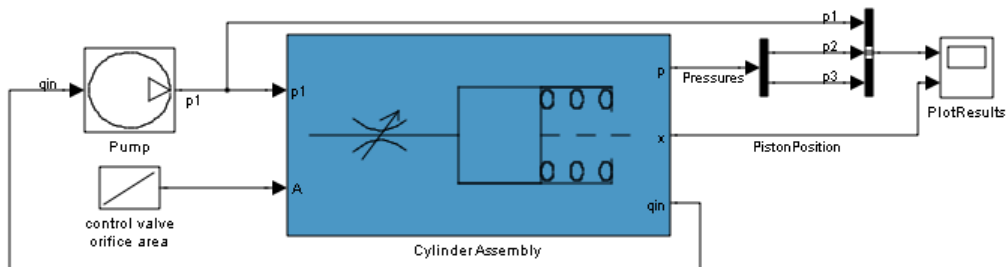
### Open the Design Optimization Tool

Open the Design Optimization tool to configure and run design optimization problems interactively.

```
sdotool('sdoHydraulicCylinder')
```



### Single Hydraulic Cylinder Simulation



Copyright 1990-2011 The MathWorks, Inc.

The screenshot shows the 'DESIGN OPTIMIZATION' software interface. The 'RESPONSE OPTIMIZATION' tab is active. The 'Design Variables Set' is set to 'None'. The 'Data Browser' panel is expanded to show the 'Model Workspace (sdoHydraulicCylinder)' with the following variables:

Name	Value
A	0.0100
Ac	1.0000e-03
Beta	700000000
C1	2.0000e-08

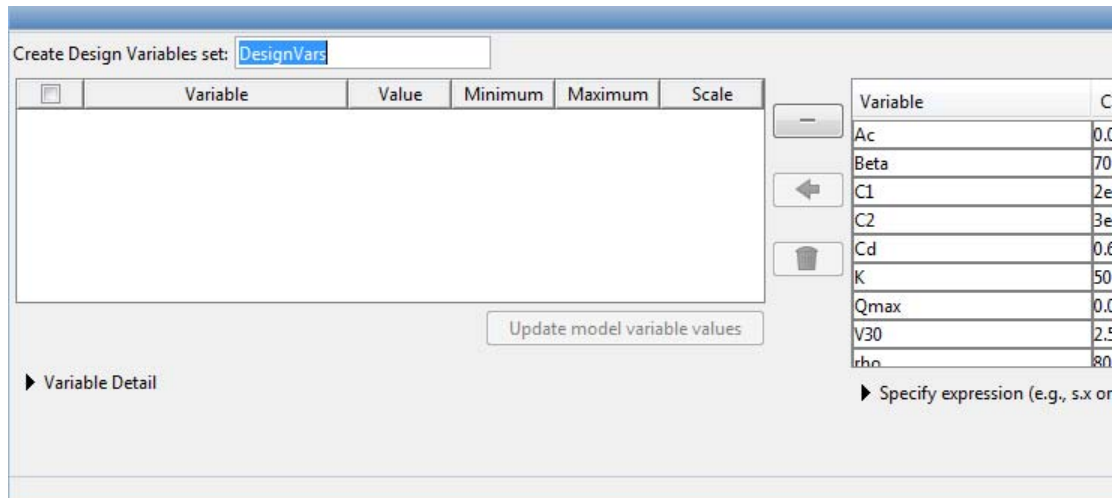
### Specify Design Variables

Specify the following model parameters as design variables for optimization:

- Cylinder cross-sectional area  $A_c$

- Piston spring constant K

In the **Design Variables Set** drop-down list, select **New**. A dialog to select model parameters for optimization opens.



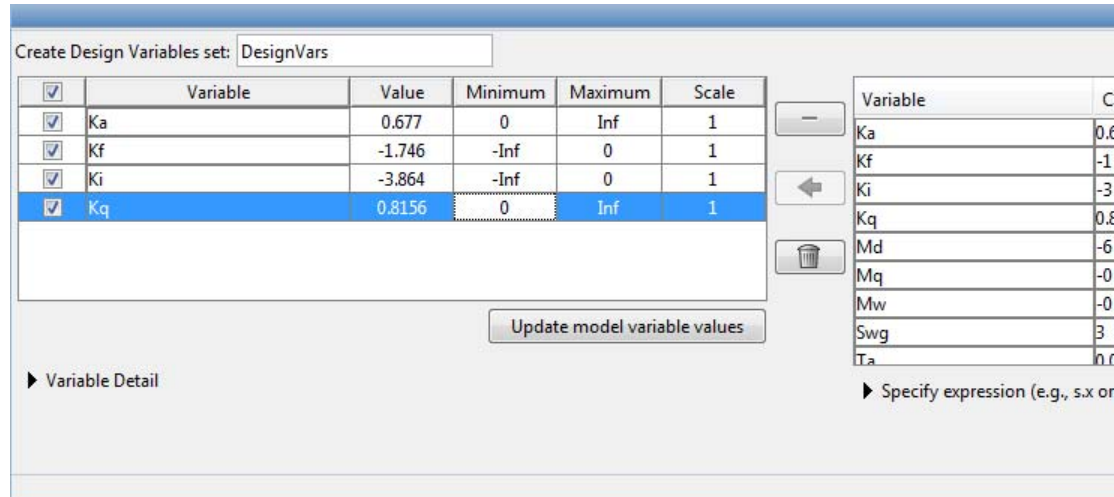
Select Ac and K. Click <- to add the selected parameters to the design variables set.

Limit the cylinder cross-sectional area to circular area with radius between 1 and 2 centimeters and the piston spring constant to a range of  $1e4$  to  $10e4$  N/m. To do so, specify the maximum and minimum for the corresponding variable in the **Maximum** and **Minimum** columns.

Because the variable values are different orders of magnitude, scale Ac by  $1e-3$  and K by  $1e5$ .

Press **Enter** after you specify the values.





Click **OK**. A new variable `DesignVars` appears in the **SDOTOOL Workspace**.

### Specify Design Requirements

The design requirements require logged model signals. During optimization, the model is simulated using the current value of the design variables and the logged signal is used to evaluate the design requirements.

Log the cylinder pressures, which is the first output port of the **Cylinder Assembly** block.

In the **New** drop-down list, select **Signal**. A dialog to select model signals to log opens.

DESIGN OPTIMIZATION      RESPONSE OPTIMIZATION

Design Variables Set: DesignVars

Uncertain Variables Set: None

DATA TO PLOT: No data selected Add Plot Plot Current Response

SELECT      EVALUATE REQUIREMENTS

NEW      NEW REQUIREMENT

VARIABLES

Data Browser

Search workspace variables

▼ MATLAB Workspace

Name ▲	Value

▼ Model Workspace (sdoHydraulicCylinder)

Name ▲	Value
A	0.0100
Ac	1.0000e-03
Beta	700000000
C1	2.0000e-08

▼ Design Optimization Workspace

Name ▲	Value
DesignVars	<2x1 para...

▼ Variable Preview

NEW REQUIREMENT

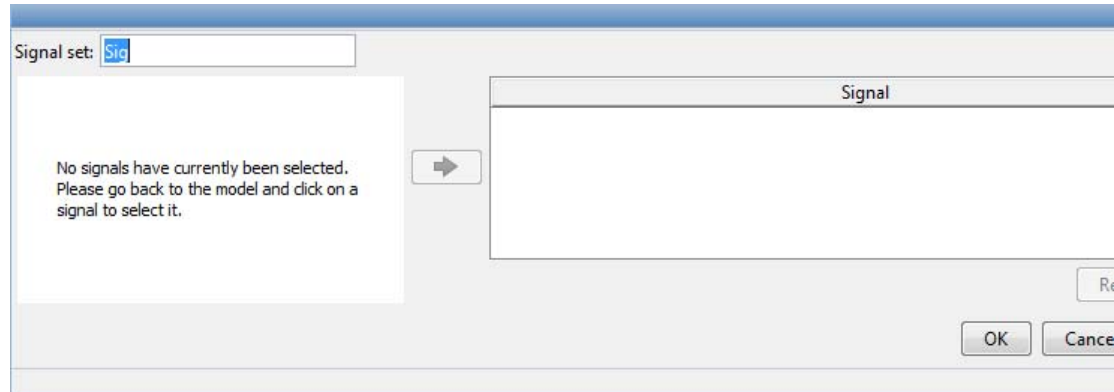
- Signal Bound**  
Use to specify a piecewise linear bound on a signal.
- Step Response Envelope**  
Use to specify a step response envelope on a signal.
- Signal Tracking**  
Use to specify a tracking requirement on a signal.
- Custom Requirement**  
Use to create a custom requirement.

NEW SIGNAL

- Signal**  
Specify Simulink Signals to log so they can be used in design requirements.

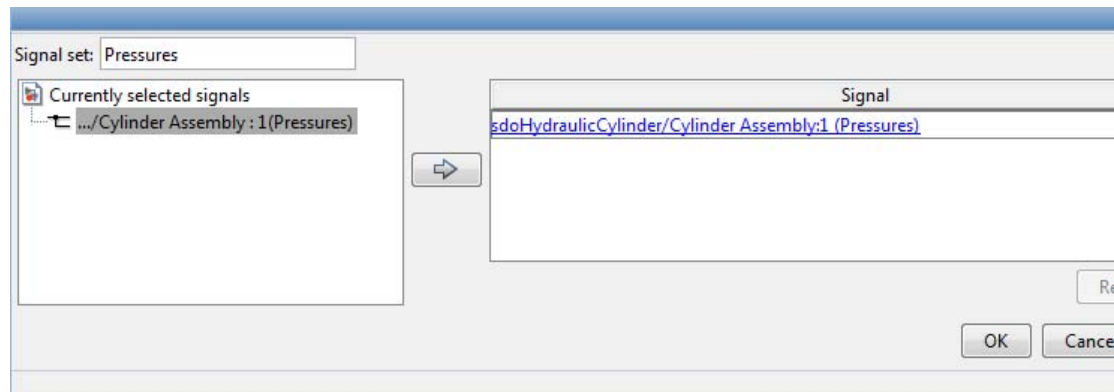
NEW REQUIREMENT USING MODEL BLOCKS

- Time Domain Check Blocks**  
Use the Simulink Design Optimization library to add time domain check blocks to a Simulink model.



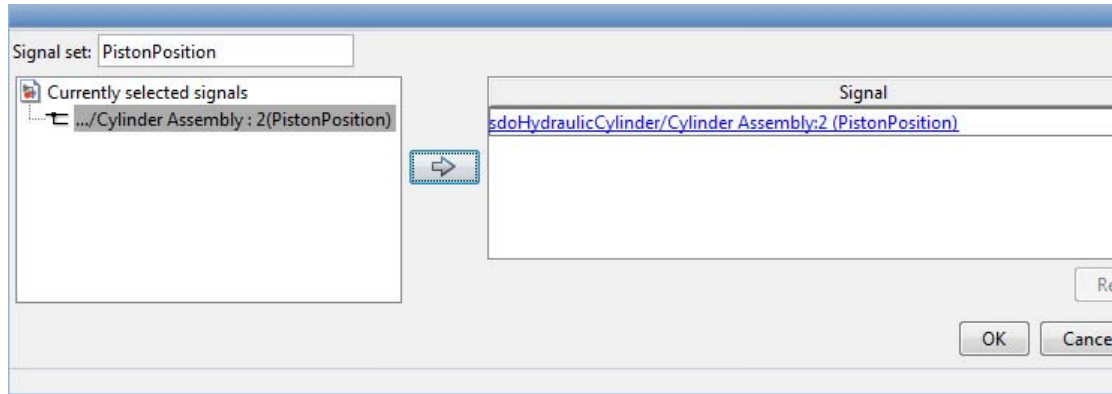
Enter **Pressures** as the signal name in the **Signal set** field. Then, in the Simulink model, click the first output port of the **Cylinder Assembly** block named **Pressure**. The dialog updates to display the selected signal.

Select the signal in the dialog and click -> to add it to the signal set.



Click **OK**. A new variable **Pressures** appears in the **SDOTOOL Workspace**.

Similarly, log the piston position, which is the second output of the **Cylinder Assembly** block, in a variable named **PistonPosition**.



Specify the maximum cylinder pressure requirement of less than  $1.75e6$  N/m.

In the **New** drop-down list, select **Signal Bound**. A dialog to create a signal bound requirement opens.

DESIGN OPTIMIZATION      RESPONSE OPTIMIZATION

Design Variables Set: DesignVars

Uncertain Variables Set: None

**VARIABLES**

Data Browser

Search workspace variables

▼ MATLAB Workspace

Name ▲	Value

▼ Model Workspace (sdoHydraulicCylinder)

Name ▲	Value
A	0.0100
Ac	1.0000e-03
Beta	700000000
C1	2.0000e-08

▼ Design Optimization Workspace

Name ▲	Value
DesignVars	<2x1 para...
PistonPosition	<1x1 Simul...
Pressures	<1x1 Simul...

▼ Variable Preview

**RESPONSE OPTIMIZATION**

Select Evaluate Requirements

Data to Plot: No data selected Add Plot Plot Current Response

**New Requirement**

- Signal Bound**  
Use to specify a piecewise linear bound on a signal.
- Step Response Envelope**  
Use to specify a step response envelope on a signal.
- Signal Tracking**  
Use to specify a tracking requirement on a signal.
- Custom Requirement**  
Use to create a custom requirement.

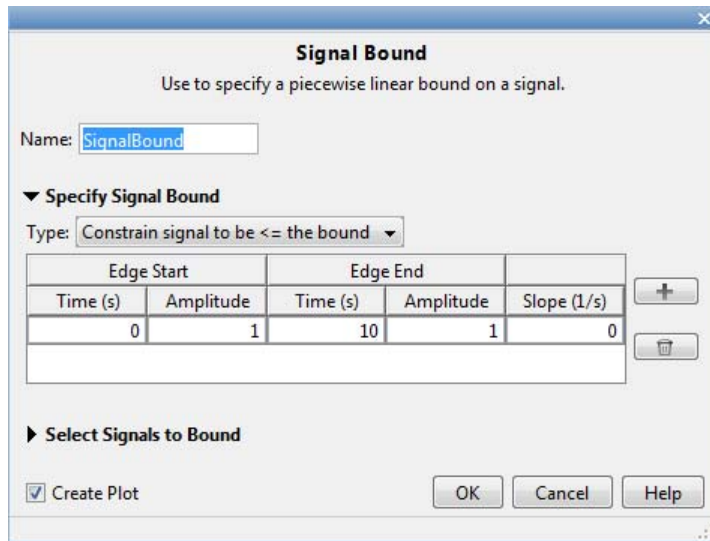
**New Signal**

- Signal**  
Specify Simulink Signals to log so they can be used in design requirements.

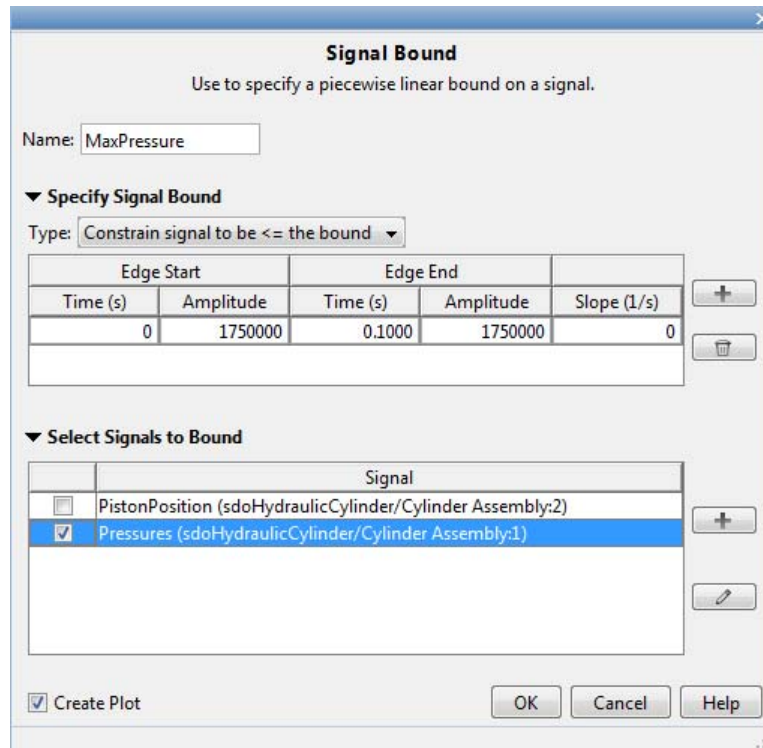
**New Requirement Using Model Blocks**

- Time Domain Check Blocks**  
Use the Simulink Design Optimization library to add time domain check blocks to a Simulink model.

**PLOTS**



In the **Amplitude** columns, enter the maximum pressure requirement of  $1.75 \times 10^6$  N/m and **Requirement Name** as MaxPressure. In the **Select Model Signals** area, select Pressures, the signal on which this requirement applies.



Click **OK**.

- A new MaxPressure variable appears in the **SDOTOOL Workspace**.
- A graphical view of the maximum pressure requirement is automatically created.

The screenshot displays the ANSYS Workbench interface for Response Optimization. The 'DESIGN OPTIMIZATION' tab is selected, showing 'Design Variables Set' as 'DesignVars' and 'Uncertain Variables Set' as 'None'. The 'RESPONSE OPTIMIZATION' tab is also visible, with a 'New' dropdown menu open. The 'PLOT' tab shows a 'Time plot 1' window with a message: 'There is no data for Pressures, use "Plot Current Response" or run the optimization to...'. The plot area shows a grid with 'Amplitude' on the y-axis (scaled by  $10^6$ ) and 'Time (seconds)' on the x-axis. The 'Data Browser' on the left shows the 'Design Optimization Workspace' with variables: DesignVars, MaxPressure, PistonPosition, and Pressures.

Specify the piston position step response requirement of rise time of less than 0.04 seconds and a settling time of less than 0.05 seconds.

In the **New** drop-down list of the **Response Optimization** tab, select **Step Response Envelope**. A dialog to create a step response requirement opens.



Specify a requirement name `PistonResponse`, the required rise and settling time bounds. Select `PistonPosition` as the signal to apply the step response requirement to.

**Step Response Envelope**  
Use to specify a step response envelope on a signal.

Name:

▼ **Specify Step Response Characteristics**

Initial value:       Final value:

Step time:  seconds

Rise time:  seconds      % Rise:

Settling time:  seconds      % Settling:

% Overshoot:       % Undershoot:

▼ **Select Signals to Bound**

	Signal	
<input checked="" type="checkbox"/>	PistonPosition (sdoHydraulicCylinder/Cylinder Assembly:2)	<input type="button" value="+"/>
<input type="checkbox"/>	Pressures (sdoHydraulicCylinder/Cylinder Assembly:1)	<input type="button" value="✎"/>

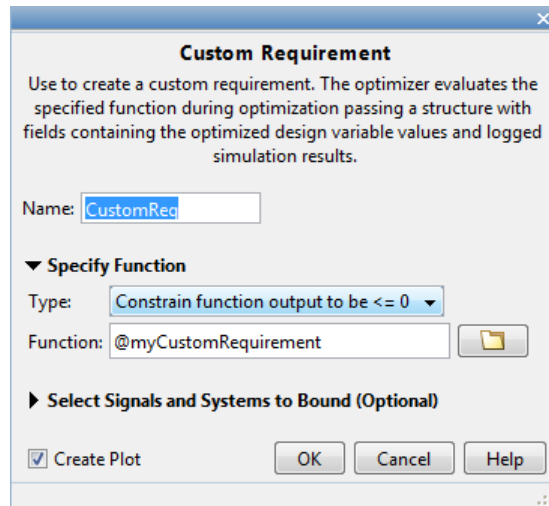
Create Plot                 

Click **OK**.

### Specify Custom Objective

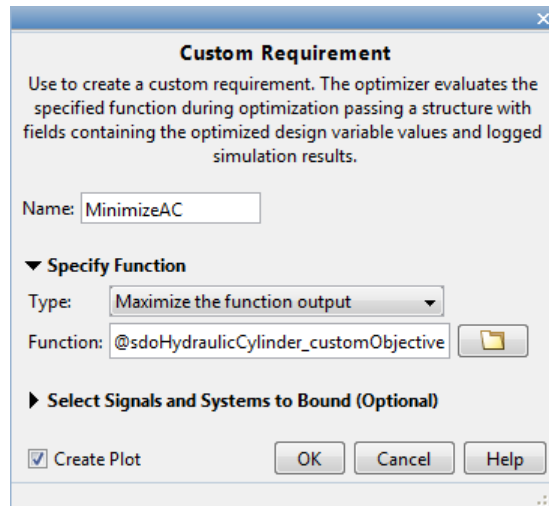
The custom objective is to minimize the cylinder cross-sectional area.

In the **New** drop-down list, select **Custom Requirement**. A dialog to create custom requirement opens.



Specify a function to call during optimization in the **Requirement function** field. At each optimization iteration, the software calls the function and passes the current design variable values. You can also optionally pass logged signals to the custom requirement. Here, you use `sdoHydraulicCylinder_customObjective` as the custom requirement function, which returns the value of the cylinder cross-sectional area.

In the **Requirement type** drop-down list, specify whether the requirement is an objective to minimize (min), an inequality constraint (<=), or an equality constraint (==).



```
type sdoHydraulicCylinder_customObjective
```

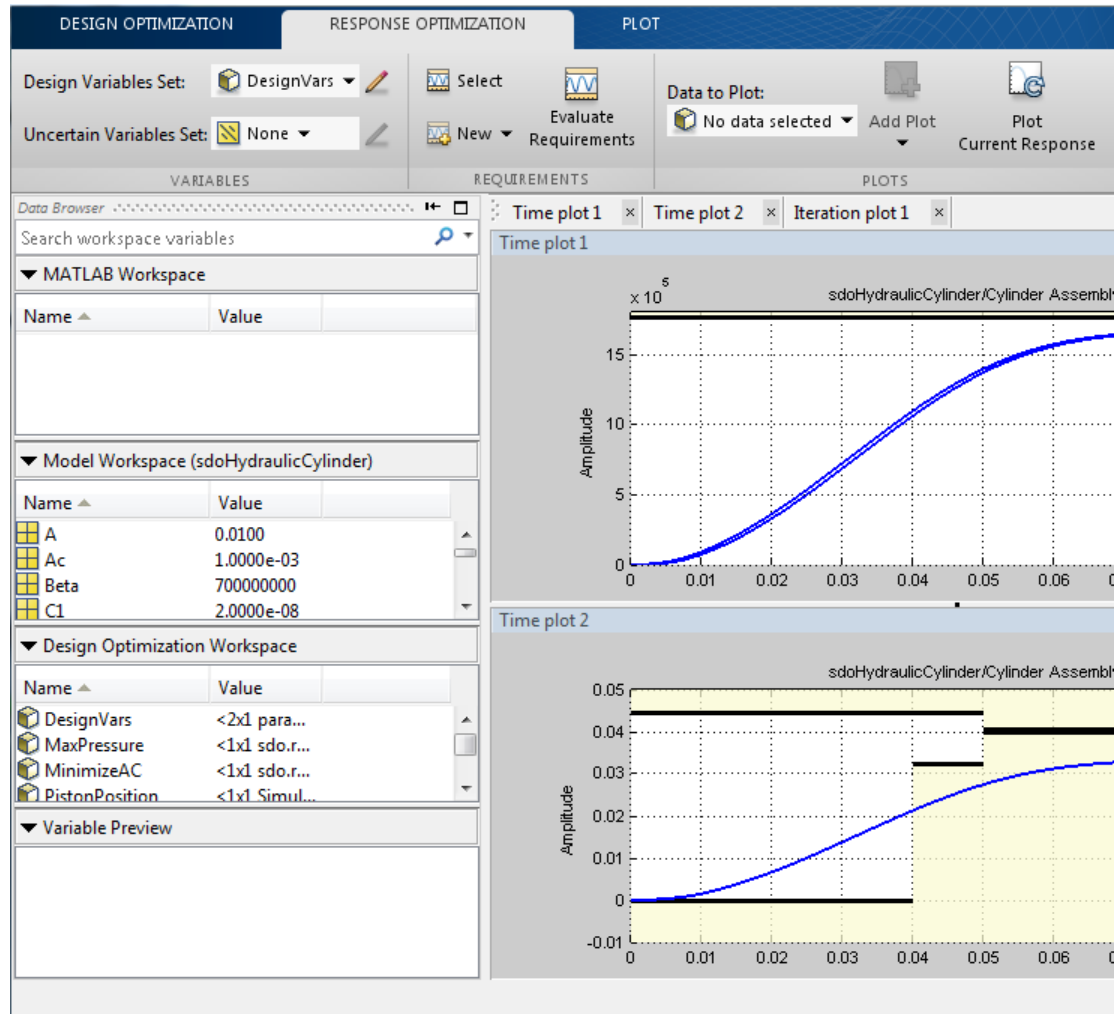
```
function objective = sdoHydraulicCylinder_customObjective(data)
%SDOHYDRAULICCYLINDER_CUSTOMOBJECTIVE
%
% The sdoHydraulicCylinder_customObjective function is used to define a
% custom requirement that can be used in the graphical SDTOOL environment.
%
% The |data| input argument is a structure with fields containing the
% design variable values chosen by the optimizer.
%
% The |objective| return argument is the objective value to be minimized by
% the SDOTool optimization solver.
%
% Copyright 2011 The MathWorks, Inc.
% $Revision: 1.1.8.1 $ $Date: 2011/05/09 01:20:51 $

%For the cylinder design problem we want to minimize the cylinder
%cross-sectional area so return the cylinder cross-sectional area as an
%objective value.
```

```
Ac = data.DesignVars(1);  
objective = Ac.Value;  
end
```

### **Evaluate the Initial Design**

Click **Plot Current Response** to simulate the model and check how well the initial design satisfies the design requirements. To show both requirement plots at the same time, use the plot layout widgets at the top-right of the tool.



From the plots, see that the maximum pressure requirement is satisfied but the piston position step response requirement is not satisfied.

### Optimize the Design

Create a plot to display how the cylinder cross-sectional area and piston spring constant are modified during optimization.

In the **Add plot** drop-down list, select **DesignVars**, which contains the optimization design variables Ac and K.

The screenshot shows the ANSYS Workbench Design Optimization interface. The 'PLOT' tab is active, displaying a toolbar with options like 'Design Variables Set', 'Uncertain Variables Set', 'Data to Plot', and 'Add Plot'. The 'Data to Plot' dropdown is set to 'DesignVars'. Below the toolbar is the 'Data Browser' with three sections: 'MATLAB Workspace', 'Model Workspace (sdoHydraulicCylinder)', and 'Design Optimization Workspace'. The 'Model Workspace' table is as follows:

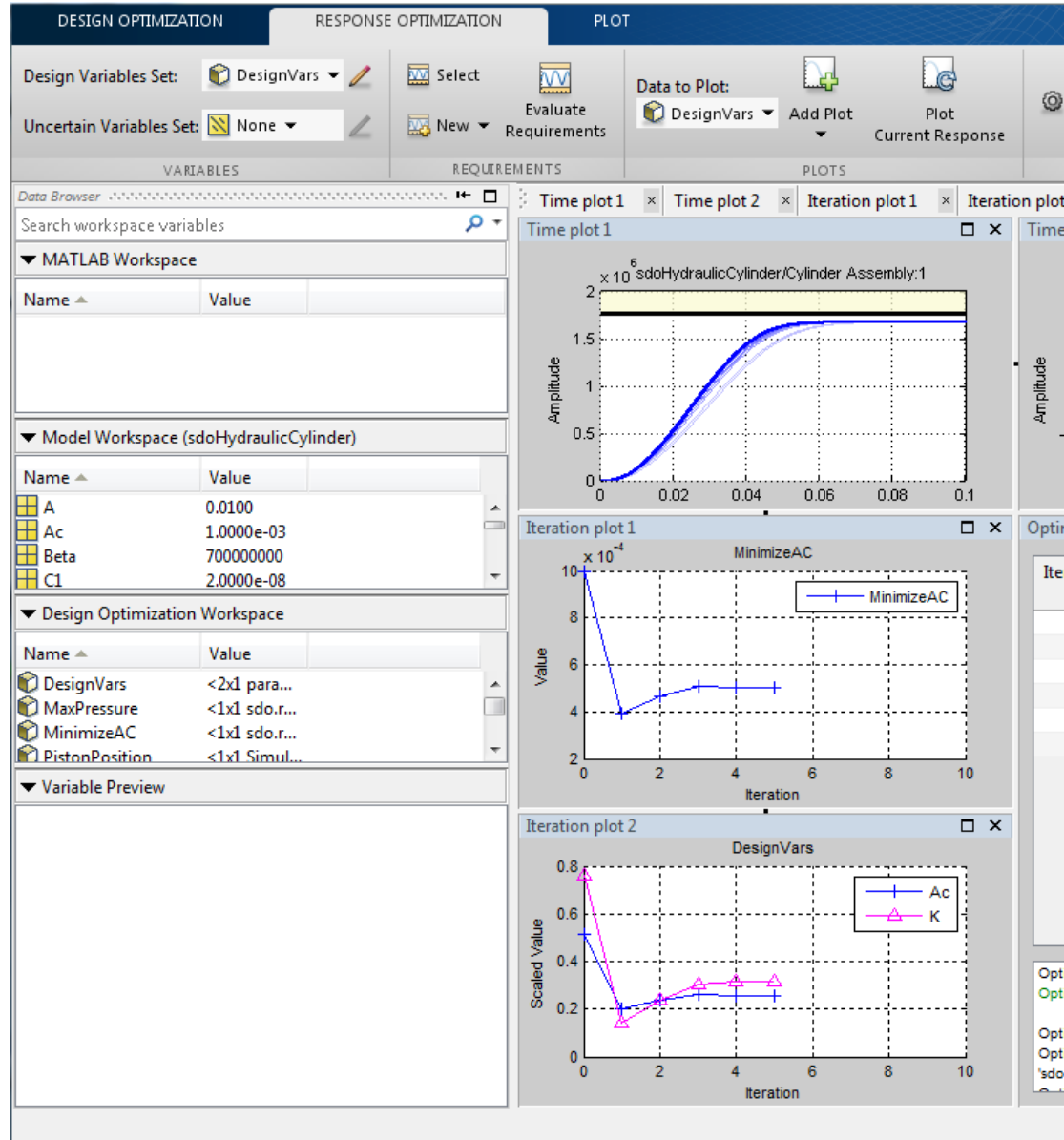
Name	Value
A	0.0100
Ac	1.0000e-03
Beta	700000000
C1	2.0000e-08

The 'Design Optimization Workspace' table is as follows:

Name	Value
DesignVars	<2x1 para...>
MaxPressure	<1x1 sdo.r...>
MinimizeAC	<1x1 sdo.r...>
PistonPosition	<1x1 Simul...>

Four plots are visible: 'Time plot 1' (Amplitude vs. Time), 'Time plot 2' (Amplitude vs. Time), 'Iteration plot 1' (Value vs. Iteration), and 'Iteration plot 2' (Scaled Value vs. Iteration). A yellow warning box in 'Iteration plot 1' states: 'There is no data for MinimizeAC, run the optimization'.

Click **Optimize**.



The optimization progress window updates at each iteration and shows that the optimization converged after 5 iterations.

The `Pressures` and `PistonPosition` plots indicate that the design requirements are satisfied. The `DesignVars` plot shows that the cylinder cross-sectional area `Ac` is minimized.

To view the optimized design variable values, click the variable name in the **SDOTOOL Workspace**. The optimized values of the design variables are automatically updated in the Simulink model.

### **Related Examples**

To learn how to optimize the cylinder design using the `sdo.optimize` command, see "Design Optimization to Meet Custom Objective at the Command Line".

```
% Close the model
setOption(sdotool('sdoHydraulicCylinder'),'NoPromptClose',true)
bdclose('sdoHydraulicCylinder')
```



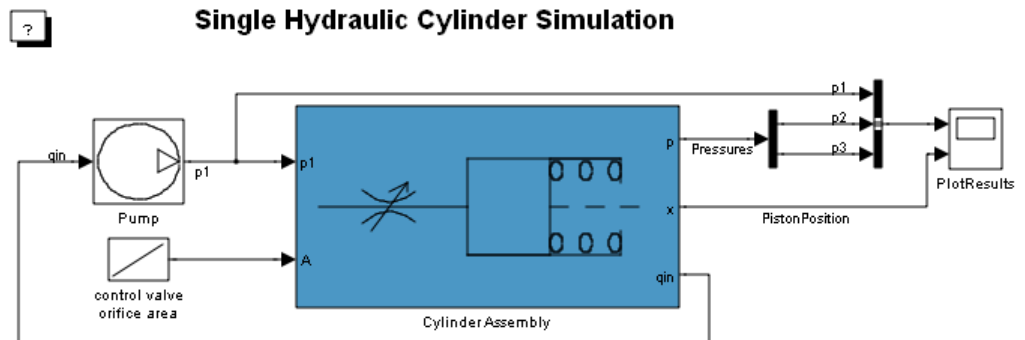
## Design Optimization to Meet a Custom Objective at the Command Line

This example shows how to optimize a design to meet custom objective using `sdo.optimize`. You optimize the cylinder parameters to minimize the cylinder geometry and satisfy design requirements.

### Hydraulic Cylinder Model

Open the Simulink model.

```
sys = 'sdoHydraulicCylinder';
open_system(sys);
```



Copyright 1990-2011 The MathWorks, Inc.

The hydraulic cylinder model is based on the Simulink model `sldemo_hydcyl`. The model includes:

- Pump and Cylinder Assembly subsystems. For more information on the subsystems, see "Single Hydraulic Cylinder Simulation".
- A step change applied to the cylinder control valve orifice area that causes the cylinder piston position to change.

### Hydraulic Cylinder Design Problem

You tune the cylinder cross-sectional area and piston spring constant to meet the following design requirements:

- Ensure that the piston position has a step response rise time of less than 0.04 seconds and settling time of less than 0.05 seconds.
- Limit the maximum cylinder pressures to 1.75e6 N/m.
- Minimize the cylinder cross-sectional area.

#### **Specify Design Variables**

Select the following model parameters as design variables for optimization:

- Cylinder cross-sectional area  $A_c$
- Piston spring constant  $K$

```
Ac = sdo.getParameterFromModel('sdoHydraulicCylinder','Ac');  
K = sdo.getParameterFromModel('sdoHydraulicCylinder','K');
```

Limit the cylinder cross-sectional area to a circular area with radius between 1 and 2 centimeters.

```
Ac.Minimum = pi*1e-2^2; % m^2  
Ac.Maximum = pi*2e-2^2; % m^2
```

Limit the piston spring constant to a range of 1e4 to 10e4 N/m.

```
K.Minimum = 1e4; % N/m  
K.Maximum = 10e4; % N/m
```

#### **Specify Design Requirements**

The design requirements require logged model signals. During optimization, the model is simulated using the current value of the design variables and the logged signal is used to evaluate the design requirements.

Log the following signals:

- Cylinder pressures, available at the first output port of the Cylinder Assembly block

```
Pressures = Simulink.SimulationData.SignalLoggingInfo;
Pressures.BlockPath      = 'sdoHydraulicCylinder/Cylinder Assembly';
Pressures.OutputPortIndex = 1;
```

- Piston position, available at the second output port of the Cylinder Assembly block

```
PistonPosition = Simulink.SimulationData.SignalLoggingInfo;
PistonPosition.BlockPath      = 'sdoHydraulicCylinder/Cylinder Assembly';
PistonPosition.OutputPortIndex = 2;
```

Create an object to store the logging information and use later to simulate the model

```
simulator = sdo.SimulationTest('sdoHydraulicCylinder');
simulator.LoggingInfo.Signals = [PistonPosition, Pressures];
```

Specify the piston position step response requirement of rise time of less than 0.04 seconds and settling time less than of 0.05 seconds.

```
PistonResponse = sdo.requirements.StepResponseEnvelope;
set(PistonResponse, ...
    'RiseTime',      0.04, ...
    'FinalValue',    0.04, ...
    'SettlingTime',  0.05, ...
    'PercentSettling', 1);
```

Specify the maximum cylinder pressure requirement of less than 1.75e6 N/m.

```
MaxPressure = sdo.requirements.SignalBound;
set(MaxPressure, ...
    'BoundTimes',      [0 0.1], ...
    'BoundMagnitudes', [1.75e6 1.75e6], ...
    'Type',             '<=');
```

For convenience, collect the performance requirements into a single structure to use later.

```
requirements = struct(...  
    'PistonResponse', PistonResponse, ...  
    'MaxPressure',    MaxPressure);
```

### **Create Objective/Constraint Function**

To optimize the cylinder cross-sectional area and piston spring constant, create a function to evaluate the cylinder design. This function is called at each optimization iteration.

Here, use an anonymous function with one argument that calls the `sdoHydraulicCylinder_design` function.

```
evalDesign = @(p) sdoHydraulicCylinder_design(p,simulator,requirements);
```

The function:

- Has one input argument that specifies the cylinder cross-sectional area and piston spring constant values.
- Returns the optimization objective value and optimization constraint violation values.

The optimization solver minimizes the objective value and attempts to keep the optimization constraint violation values negative. Type `help sdoExampleCostFunction` for more details on how to write the objective/constraint function.

The `sdoHydraulicCylinder_design` function uses the `simulator` and `requirements` objects to evaluate the design. Type `edit sdoHydraulicCylinder_design` to examine the function in more detail.

```
type sdoHydraulicCylinder_design
```

```
function design = sdoHydraulicCylinder_design(p,simulator,requirements)  
%SDOHYDRAULICCYLINDER_DESIGN  
%  
% The sdoHydraulicCylinder_design function is used to evaluate a cylinder  
% design.
```

```
%
% The |p| input argument is the vector of cylinder design parameters.
%
% The |simulator| input argument is a sdo.SimulinkTest object used to
% simulate the |sdoHydraulicCylinder| model and log simulation signals
%
% The |requirements| input argument contains the design requirements used
% to evaluate the cylinder design
%
% The |design| return argument contains information about the design
% evaluation that can be used by the |sdo.optimize| function to optimize
% the design.
%
% see also sdo.optimize, sdoExampleCostFunction

% Copyright 2011 The MathWorks, Inc.
% $Revision: 1.1.8.2 $ $Date: 2011/06/30 16:51:02 $

%% Simulate the model
%
% Use the simulator input argument to simulate the model and log model
% signals.
%
% First ensure that we simulate the model with the parameter values chosen
% by the optimizer.
%
simulator.Parameters = p;
% Simulate the model and log signals.
%
simulator = sim(simulator);
% Get the simulation signal log, the simulation log name is defined by the
% model |SignalLoggingName| property
%
logName = get_param('sdoHydraulicCylinder','SignalLoggingName');
simLog = get(simulator.LoggedData,logName);

%% Evaluate the design requirements
%
% Use the requirements input argument to evaluate the design requirements
%
```

```
% Check the PistonPosition signal against the stepresponse requirement
%
PistonPosition = get(simLog,'PistonPosition');
cPiston = evalRequirement(requirements.PistonResponse,PistonPosition.Values);
% Check the Pressure signals against the maximum requirement
%
Pressures = find(simLog,'Pressures');
cPressure = evalRequirement(requirements.MaxPressure,Pressures.Values);
% Use the PistonResponse and MaxPressure requirements as non-linear
% constraints for optimization.
design.Cleq = [cPiston(:);cPressure(:)];
% Add design objective to minimize the Cylinder cross-sectional area
Ac = p(1); %Since we called sdo.optimize(evalDesign,[Ac;K])
design.F = Ac.Value;
end
```

#### Evaluate the Initial Design

Call the objective function with the initial cylinder cross-sectional area and initial piston spring constant.

```
initDesign = evalDesign([Ac;K]);
```

The function simulates the model and evaluates the design requirements. The scope shows that the maximum pressure requirement is satisfied but the piston position step response requirement is not satisfied.

initDesign is a structure with the following fields:

- Cleq shows that some of the inequality constraints are positive indicating they are not satisfied by the initial design.

```
initDesign.Cleq
```

```
ans =
```

```
-0.3839
-0.1861
-0.1836
```

```

-1.0000
 0.3033
 0.2909
 0.1671
 0.2326
-0.0480
-0.0480

```

- F shows the optimization objective value (in this case the cylinder cross-sectional area). The initial design cross-sectional area, as expected, has the same value as the initial cross-sectional area parameter Ac.

```
initDesign.F
```

```
ans =
```

```
1.0000e-03
```

### Optimize the Design

Pass the objective function, initial cross-sectional area and piston spring constant values to `sdo.optimize`.

```
[pOpt,optInfo] = sdo.optimize(evalDesign,[Ac;K]);
```

```
Optimization started 04-Jan-2012 12:33:08
```

Iter	F-count	f(x)	max constraint	Line search Steplength	Directional derivative	First-o optimal
0	5	0.001	0.3033			
1	10	0.000390105	3.762	0.696	-0.000876	0.001
2	15	0.000462964	0.348	0.106	0.000689	0.003
3	20	0.000509424	0.04789	0.0706	0.000658	0.01
4	25	0.000503622	0.001654	0.0109	-0.000531	0.05
5	30	0.000500809	0.0004008	0.00197	-0.00143	0.001

```
Local minimum possible. Constraints satisfied.
```

```
fmincon stopped because the size of the current search direction is less than  
twice the selected value of the step size tolerance and constraints are  
satisfied to within the selected value of the constraint tolerance.
```

The optimization repeatedly evaluates the cylinder design by adjusting the cross-sectional area and piston spring constant to meet the design requirements. From the scope, see that the maximum pressure and piston response requirements are met.

The `sdo.optimize` function returns:

- `pOpt` shows the optimized cross-sectional area and piston spring constant values.

`pOpt`

```
pOpt(1,1) =
```

```
    Name: 'Ac'  
    Value: 5.0081e-04  
  Minimum: 3.1416e-04  
  Maximum: 0.0013  
    Free: 1  
    Scale: 0.0020  
    Info: [1x1 struct]
```

```
pOpt(2,1) =
```

```
    Name: 'K'  
    Value: 2.0651e+04  
  Minimum: 10000  
  Maximum: 100000  
    Free: 1  
    Scale: 65536  
    Info: [1x1 struct]
```



```
2x1 param.Continuous
```

- `optInfo` is a structure that contains optimization termination information such as number of optimization iterations and the optimized design.

```
optInfo
```

```
optInfo =
```

```
    Cleq: [10x1 double]
         F: 5.0081e-04
  Gradients: [1x1 struct]
    exitflag: 4
  iterations: 5
 SolverOutput: [1x1 struct]
         Stats: [1x1 struct]
```

For example, the `Cleq` field shows the optimized non-linear inequality constraints are all non-positive to within optimization tolerances, indicating that the maximum pressure and piston response requirements are satisfied.

```
optInfo.Cleq
```

```
ans =
```

```
-0.1093
 0.0003
 0.0003
-1.0000
-0.1186
 0.0004
-0.0205
 0.0002
-0.0476
-0.0476
```

The F field contains the optimized cross-sectional area. The optimized cross-sectional area value is nearly 50% less than the initial value.

```
optInfo.F
```

```
ans =
```

```
5.0081e-04
```

#### **Update the Model Variable Values**

By default, the model variables Ac and K are not updated at the end of optimization. Use the `setValueInModel` command to update the model variable values.

```
sdo.setValueInModel('sdoHydraulicCylinder',pOpt)
```

#### **Related Examples**

To learn how to optimize the cylinder design using the Design Optimization tool, see "Design Optimization to Meet Custom Objective Using the Design Optimization Tool".

```
% Close the model  
bdclose('sdoHydraulicCylinder')
```

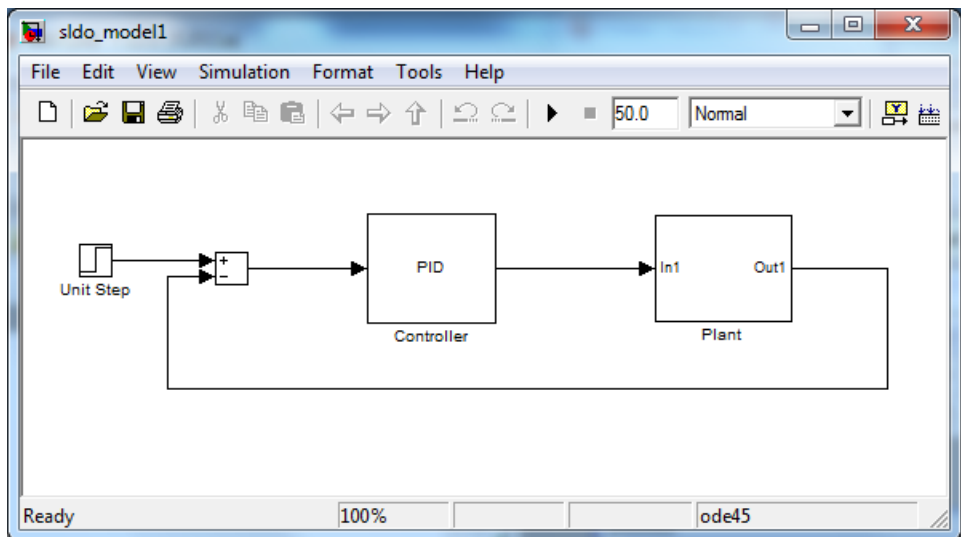
## Design Optimization to Meet Custom Signal Requirements (GUI)

This example shows how to optimize a design to meet a custom signal requirement. You optimize the controller parameters to minimize the plant actuation signal energy while satisfying step response requirements.

- 1 Load a saved Design Optimization tool session.

```
load sldo_model1_custom_signal_session
sdotool(SDOSessionData);
```

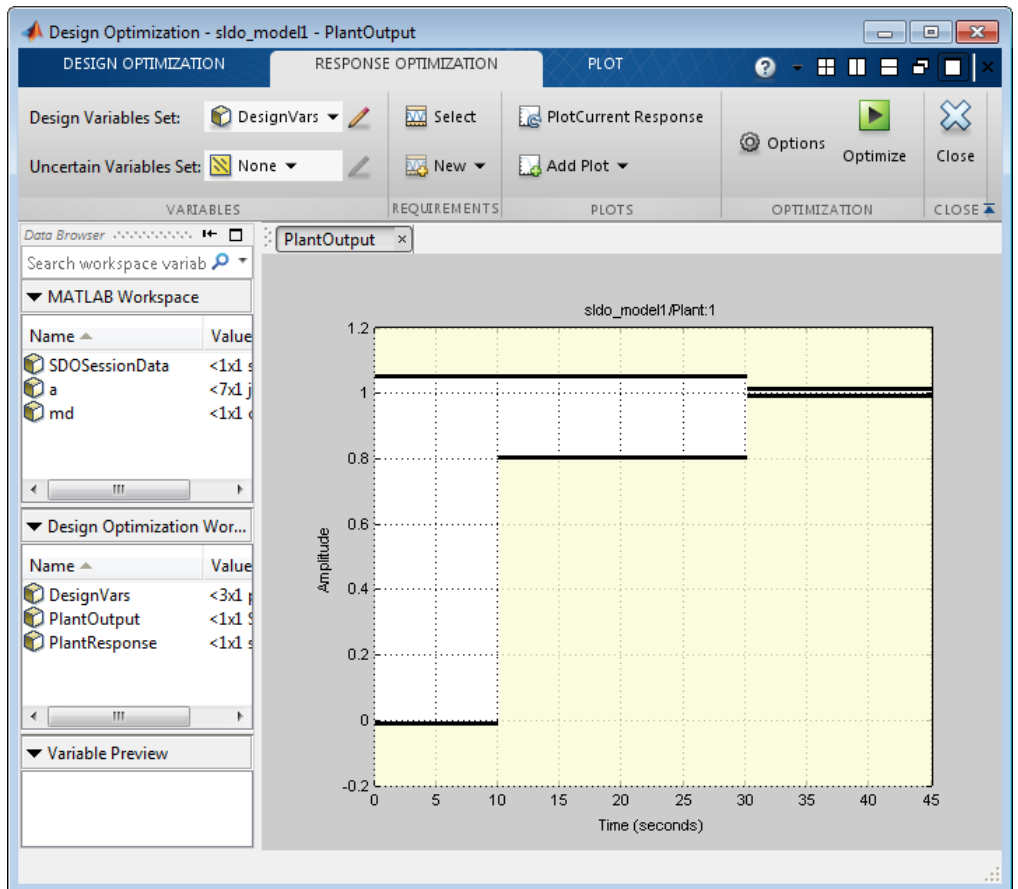
The following Simulink model opens.



The Design Optimization tool, configured with the following settings, also opens:

- Step response characteristics, specified on the output of the Plant block, that the model output must satisfy:
  - Maximum overshoot of 5%
  - Maximum rise time of 10 seconds

- Maximum settling time of 30 seconds
- Design variable set with the controller parameters Kp, Ki and Kd. These parameters have a minimum value of 0.
- The variables for step requirement (PlantResponse), logged signal (PlantOutput) and design variables (DesignVars) which appear in the **Design Optimization Workspace**.



- 2 Specify a signal to log. You apply the custom requirement on this logged signal.

**a** Select **New > Signal**.

A window opens where you select a signal to log.

**b** In the Simulink model window, click the output of the Controller block.

The window updates to display the selected signal.

**c** Select the signal and click  to add it to the signal set.

**d** In **Signal set**, enter PlantActuator.

Click **OK**. A new variable PlantActuator appears in the **Design Optimization Workspace** of the Design Optimization tool window.

**3** Specify the custom requirement to apply to the signal.

The custom requirement calls the objective function `sldo_model1_minimize_energy` which returns the energy in the PlantActuator signal. The signal energy is minimized. This function accepts:

- An input argument `data` which is a structure with fields for the design variables in the **Design Optimization Workspace**. Signals are logged for the nominal and uncertain parameter values if there are any.
- Returns the objective value to be minimized.

---

**Tip** Type `type sldo_model1_minimize_energy` to see the contents of this function.

---

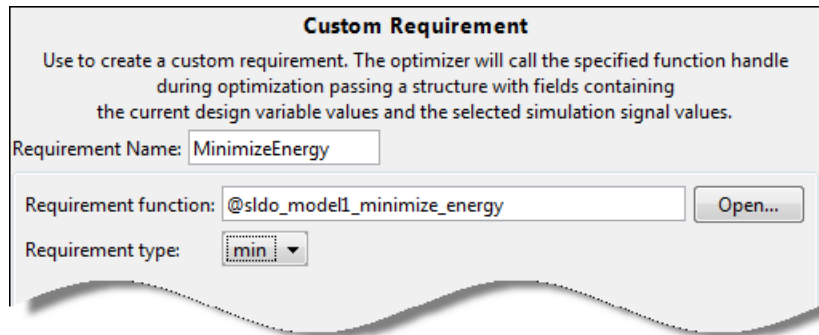
**a** Select **New > Custom Requirement**.

A window opens where you specify the custom requirement.

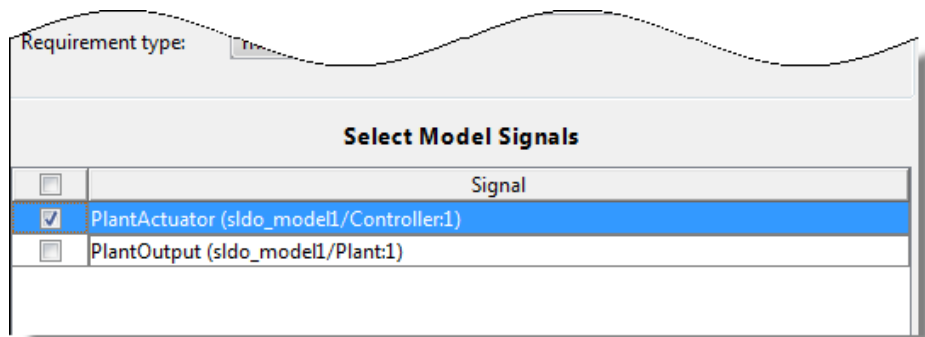
**b** Specify `MinimizeEnergy` as the **Requirement Name**.

**c** Specify `@sldo_model1_minimize_energy` as the **Requirement function**.

**d** Select `min` as the **Requirement type**.

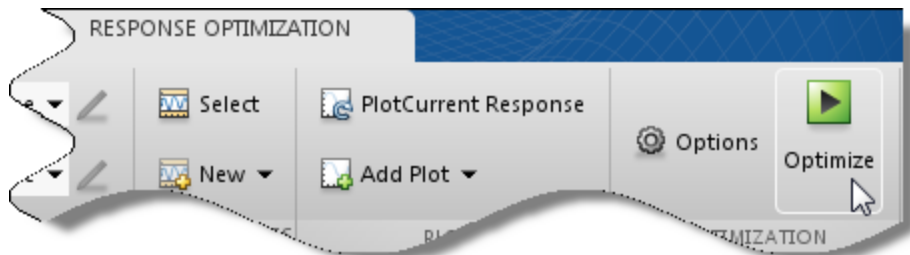


- 4 In the **Select Model Signals** area, select the PlantActuator check box to associate the custom requirement with that signal.

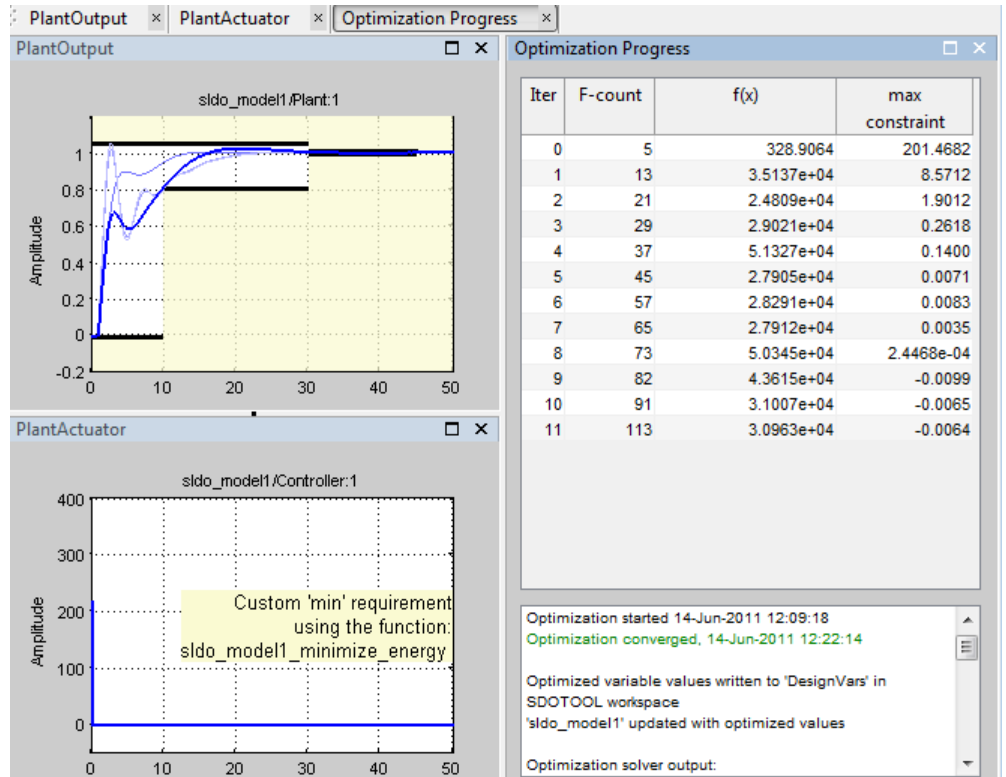


Click **OK**. A new variable appears in the **Design Optimization Workspace**. The window also updates to graphically display the custom signal requirement.

- 5 Click **Optimize**.



After a few iterations, the optimization converges to meet both the custom signal and step response requirements.



**6** Close the model.

```

setOption(sdtool('sldo_model1'),'NoPromptClose',true)
bdclose('sldo_model1')
  
```

## Design Optimization to Meet Frequency-Domain Requirements

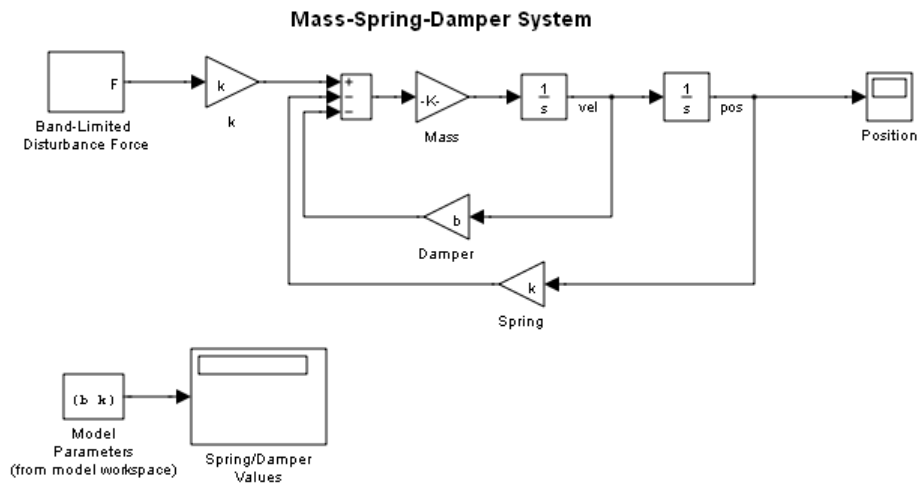
This example shows how to tune model parameters to meet frequency-domain requirements using the Design Optimization tool.

The example requires Simulink Control Design™

### Mass-Spring-Damper Model

Open the Simulink Model.

```
open_system('sdoMassSpringDamper')
```



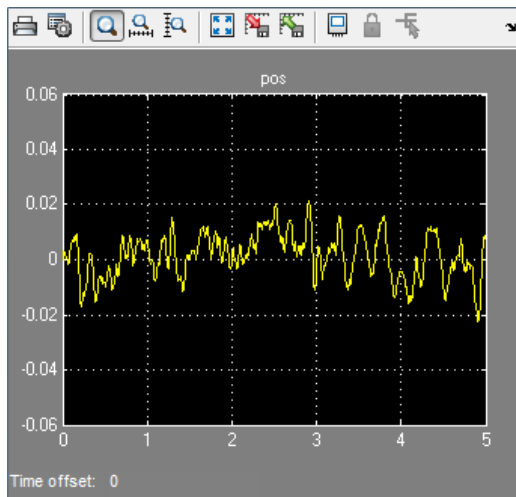
Copyright 2002-2011 The MathWorks, Inc.

Mass-spring-damper models represent simple suspension systems and for this example we tune the system to meet typical suspension requirements. The model implements the second order system representing a mass-spring-damper using Simulink blocks and includes:



- a Mass gain block parameterized by the total suspended mass,  $m_0+m_{load}$ . The total mass is the sum of a nominal mass  $m_0$  and a variable load mass  $m_{Load}$ .
- a Damper gain block parameterized by the damping coefficient,  $b$ .
- a Spring gain block parameterized by the spring constant,  $k$ .
- two integrator blocks to compute the mass velocity and position.
- a Band-Limited Disturbance Force block applying a disturbance force to the Mass. The disturbance force is assumed to be band-limited white noise.

Simulate the model to view the system response to the applied disturbance force.



### Design Problem

The initial system has a bandwidth that is too high. This can be seen from the spiky position signal. You tune the spring and damper values to meet the following requirements:

- The -3dB system bandwidth must not exceed 10 rad/s.

- The damping ratio of the system must be less than  $1/\sqrt{2}$ . This ensures that no frequencies in pass band are amplified by the system.
- Minimize the expected failure rate of the system. The expected failure rate is described by a Weibull distribution dependent on the mass, spring, and damper values.
- These requirements must all be satisfied as the load mass ranges from 0 to 20.

#### Open the Design Optimization Tool

In the Simulink model **Tools** menu, select **Response Optimization**.

#### Specify Design Variables

In the **Design Variables Set** list, select **New**. Add the b and k model variables to the design variable set.

- Specify the Minimum and Maximum values for the b variable as 100 and 10000 respectively.
- Specify the Minimum and Maximum values for the k variable as 10000 and 100000 respectively.

Create Design Variables set: DesignVars

<input checked="" type="checkbox"/>	Variable	Value	Mini...	Maxi...	Scale
<input checked="" type="checkbox"/>	b	6283	100	10000	8192
<input checked="" type="checkbox"/>	k	98696	10000	100000	131072

Update model variable val...

Variable	Current val...	
NoisePower	100000	...Band-Limited Disturba
b	6283	...sdoMassSpringDamper/...ssSpringDamper/Mode
k	98696	...Band-Limited Disturba ...ssSpringDamper/Mode workspace) ...sdoMassSpringDamper/...

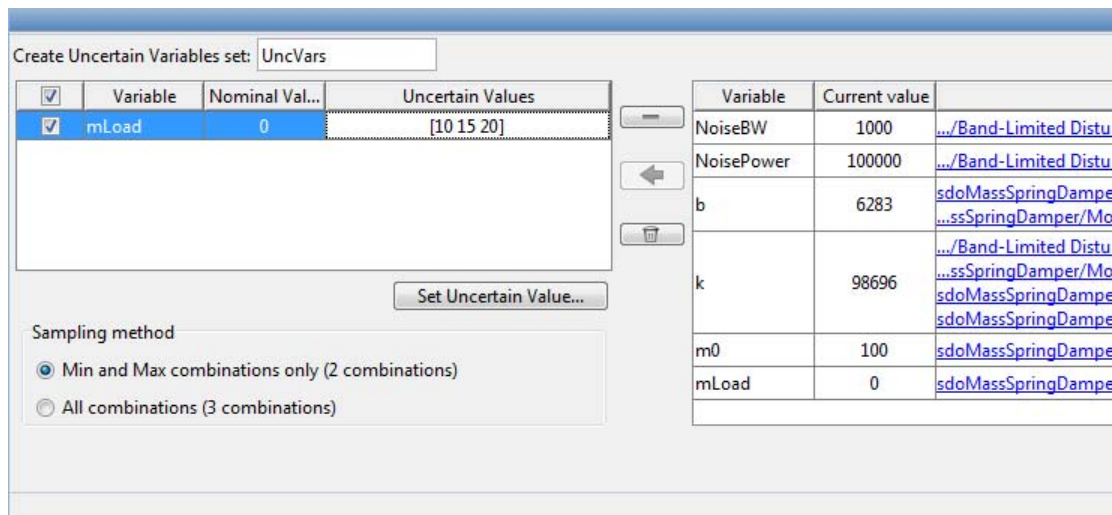
Variable Detail

Specify expression (e.g., s.x or a(3))

Click **OK**. A new variable, DesignVars, appears in the **Design Optimization Workspace**.

In the **Uncertain Variables Set** list, select **New**. Add the mLoad variable to the uncertain variables set.

- Specify the Uncertain Values value for the mLoad variable as [10 15 20]



Click **OK**. A new variable, UncVars, appears in the **Design Optimization Workspace**.

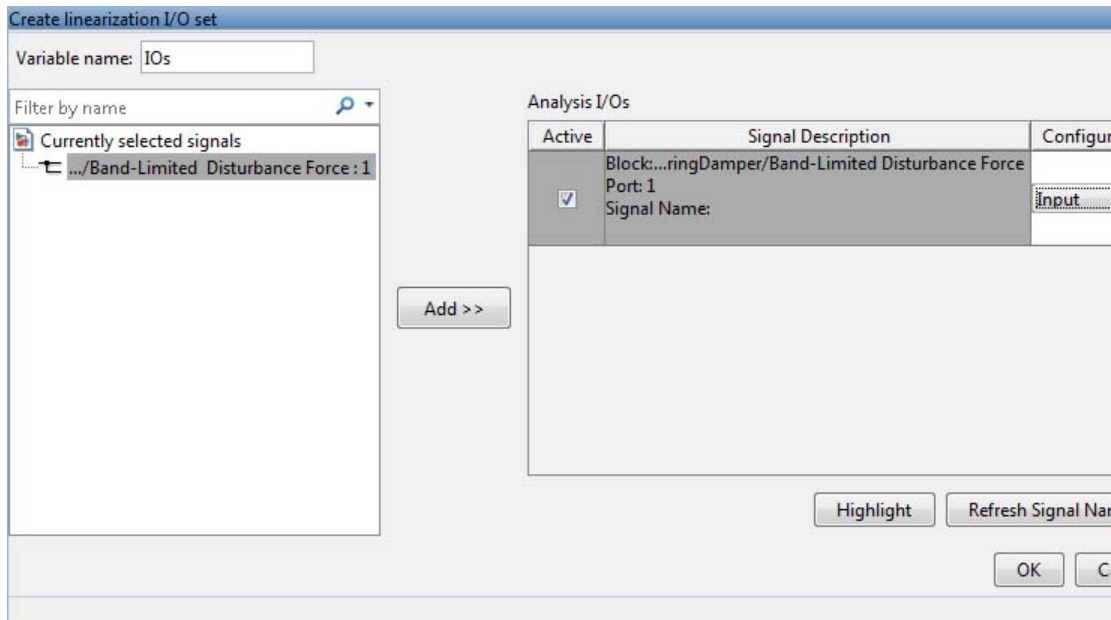
### Specify Linear Analysis Input/Output Points

Specify the input/output points defining the linear system used to compute the bandwidth and damping ratio.

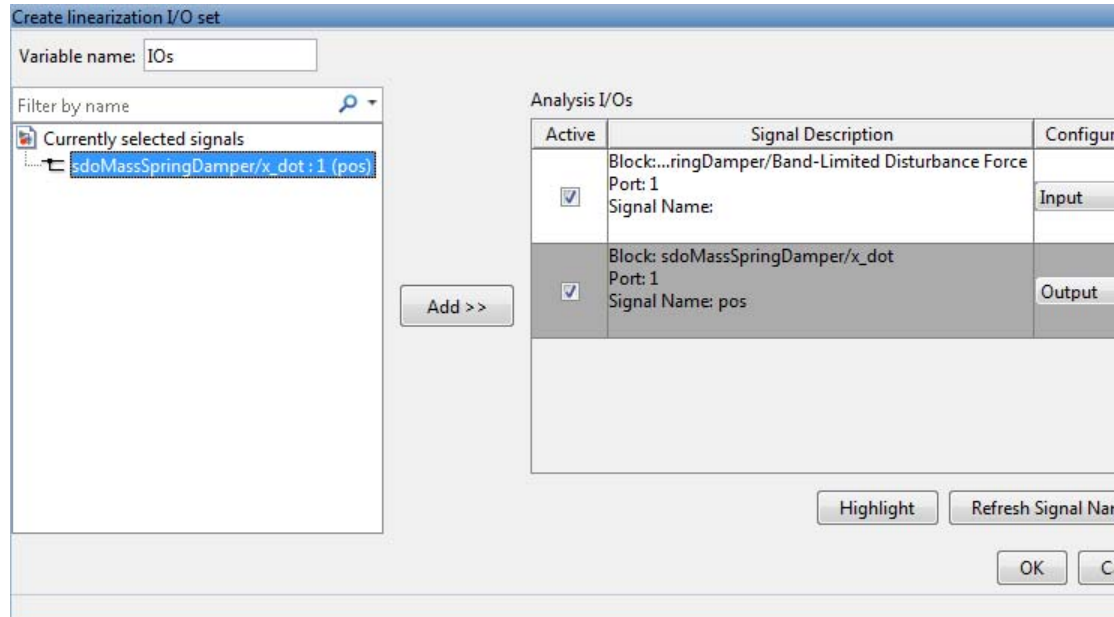
To specify the input/output points:

- In the **New** list, select **Linearization I/Os**.

- In the Simulink model, click the signal at the output of the **Band-Limited Disturbance Force** block. The **Create linearization I/O set** dialog box is updated and the chosen signal appears in it.
- In the **Create linearization I/O** dialog box, select the signal and click **Add**.
- In the **Configuration** list for the selected signal, choose **Input** to specify it as an input signal.



- Similarly, add the **pos** signal from the Simulink model. Specify this signal as an output. In the **Configuration** list, select **Output**.



- Click **OK**. A new variable, IOs, appears in the **Design Optimization Workspace**.

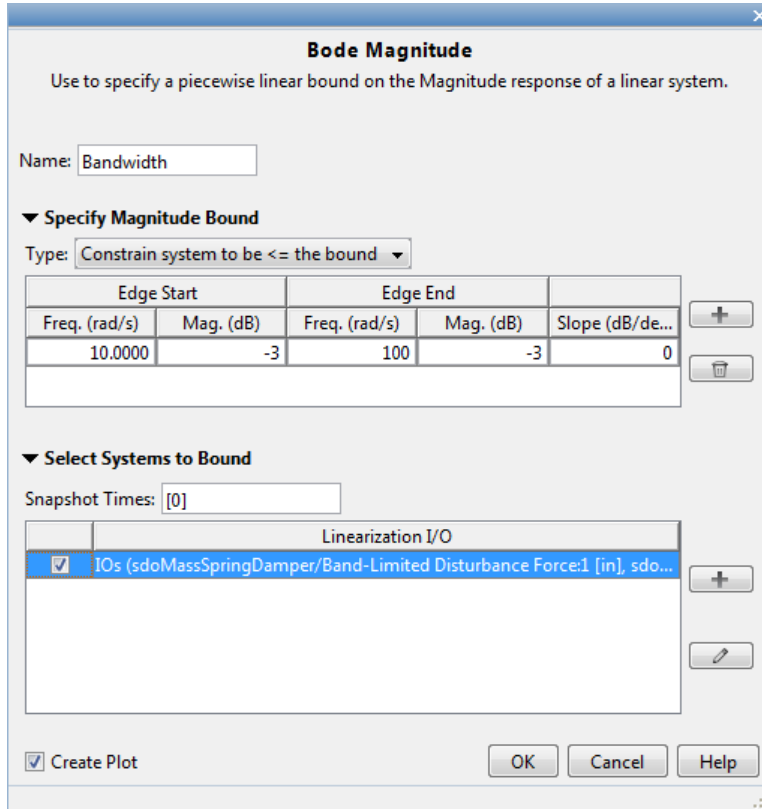
### Add Bandwidth and Damping-Ratio Requirements

Tune the spring and damper values to satisfy bandwidth and damping ratio requirements.

To specify the bandwidth requirement:

- Open a dialog to specify bounds on the Bode magnitude. In the **New** list, select **Bode Magnitude**.
- Specify the requirement name as **Bandwidth**.
- Specify the edge start frequency and magnitude as 10 rad/s and -3db, respectively.

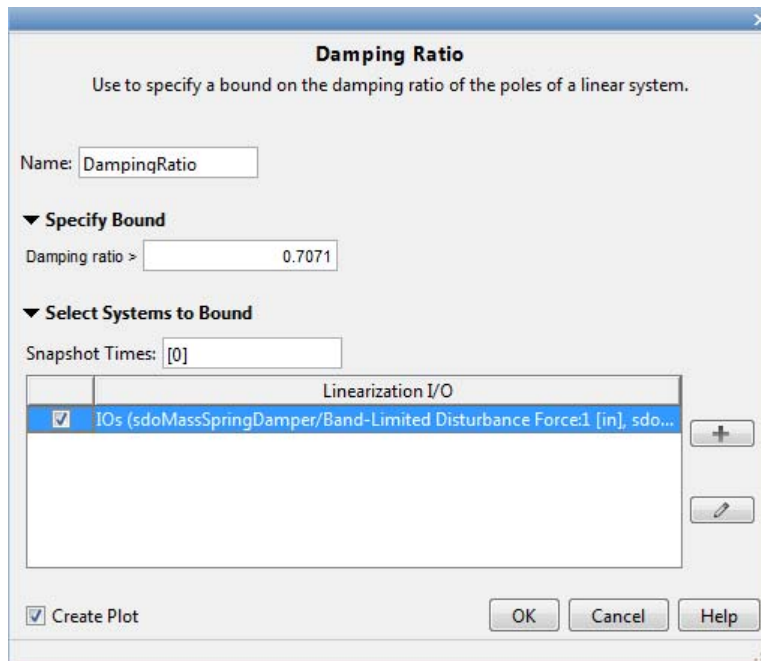
- Specify the edge end frequency and magnitude as 100 rad/s and -3db, respectively.
- Specify the input/output set to which the requirement applies by clicking **Select Systems to Bound**. Select the IOs check box .



- Click **OK**. A new requirement, **Bandwidth**, appears in the **Design Optimization Workspace** and a graphical view of the bandwidth requirement is automatically created.

To specify the damping ratio requirement:

- Open a dialog to specify bounds on the damping ratio. In the **New** list, select **Damping Ratio**.
- Specify the damping ratio bound value as 0.7071.
- Specify the input/output set to which the requirement applies by clicking **Select Systems to Bound**. Select the IOs check box .



- Click **OK**. A new requirement, **DampingRatio**, appears in the **Design Optimization Workspace** and a graphical view of the damping ratio requirement is automatically created.

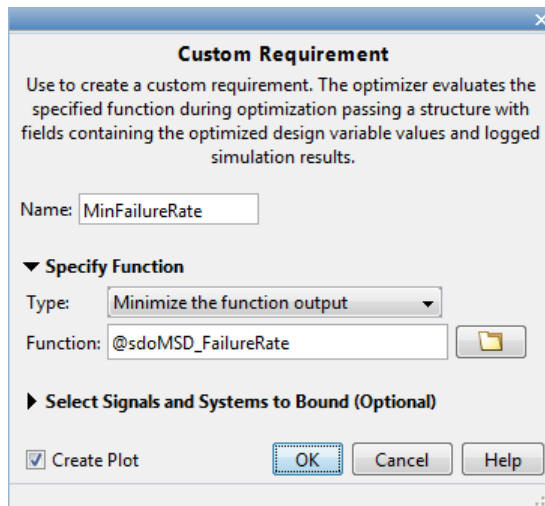
### Add a Reliability Requirement

Tune the spring and damper values to minimize the expected failure rate over a lifetime of 100e3 miles. The failure rate is computed using a Weibull

distribution on the damping ratio of the system. As the damping ratio increases the failure rate is expected to increase.

Specify the reliability requirement as a custom requirement:

- Open a dialog box to specify the custom requirements. In the **New** list, select **Custom Requirement**.
- Specify the custom requirement name as `MinFailureRate`.
- In the **Specify Function** area, select **Minimize the function output** from the **Type** list.
- Specify the function as `@sdoMSD_FailureRate`.



- Click **OK**. A new requirement, `MinFailureRate`, appears in the **Design Optimization Workspace** and a graphical view of the custom requirement is automatically created.

The `@sdoMSD_FailureRate` function returns expected mass-spring-damper failure rate for a lifetime of 100e3 miles.



```
type sdoMSD_FailureRate
```

```
function pFailure = sdoMSD_FailureRate(data)
%SDOMSD_FAILURERATE
%
% The sdoMSD_FailureRate function is used to define a custom requirement
% that can be used in the graphical SDTOOL environment.
%
% The |data| input argument is a structure with fields containing the
% design variable values chosen by the optimizer.
%
% The |pFailure| return argument is the failure rate to be minimized by the
% SDTOOL optimization solver. The failure rate is given by a Weibull
% distribution that is a function of the mass, spring and damper values.
% The design minimizes the failure rate for a 100e3 mile lifetime.
%
%
% Copyright 2011 The MathWorks, Inc.
% $Revision: 1.1.8.1 $ $Date: 2011/11/17 21:29:26 $

%Get the spring and damper design values
allVarNames = {data.DesignVars.Name};
idx          = strcmp(allVarNames,'k');
k            = data.DesignVars(idx).Value;
idx          = strcmp(allVarNames,'b');
b            = data.DesignVars(idx).Value;

%Get the nominal mass from the model
m = slResolve('m0','sdoMassSpringDamper');

%The expected failure rate is defined by the Weibull cumulative
%distribution function, 1-exp(-(x/l)^k), where k=3, l is a function of the
%mass, spring and damper values, and x the lifetime.
d          = b/2/sqrt(m*k);
pFailure = 1-exp(-(100e3*d/250e3)^3);
end
```

**Optimize the Design**

Before running the optimization be sure to have completed the earlier steps. Alternatively, you can load the `sdoMassSpringDamper_sdoSession` from the model workspace into the Design Optimization tool.

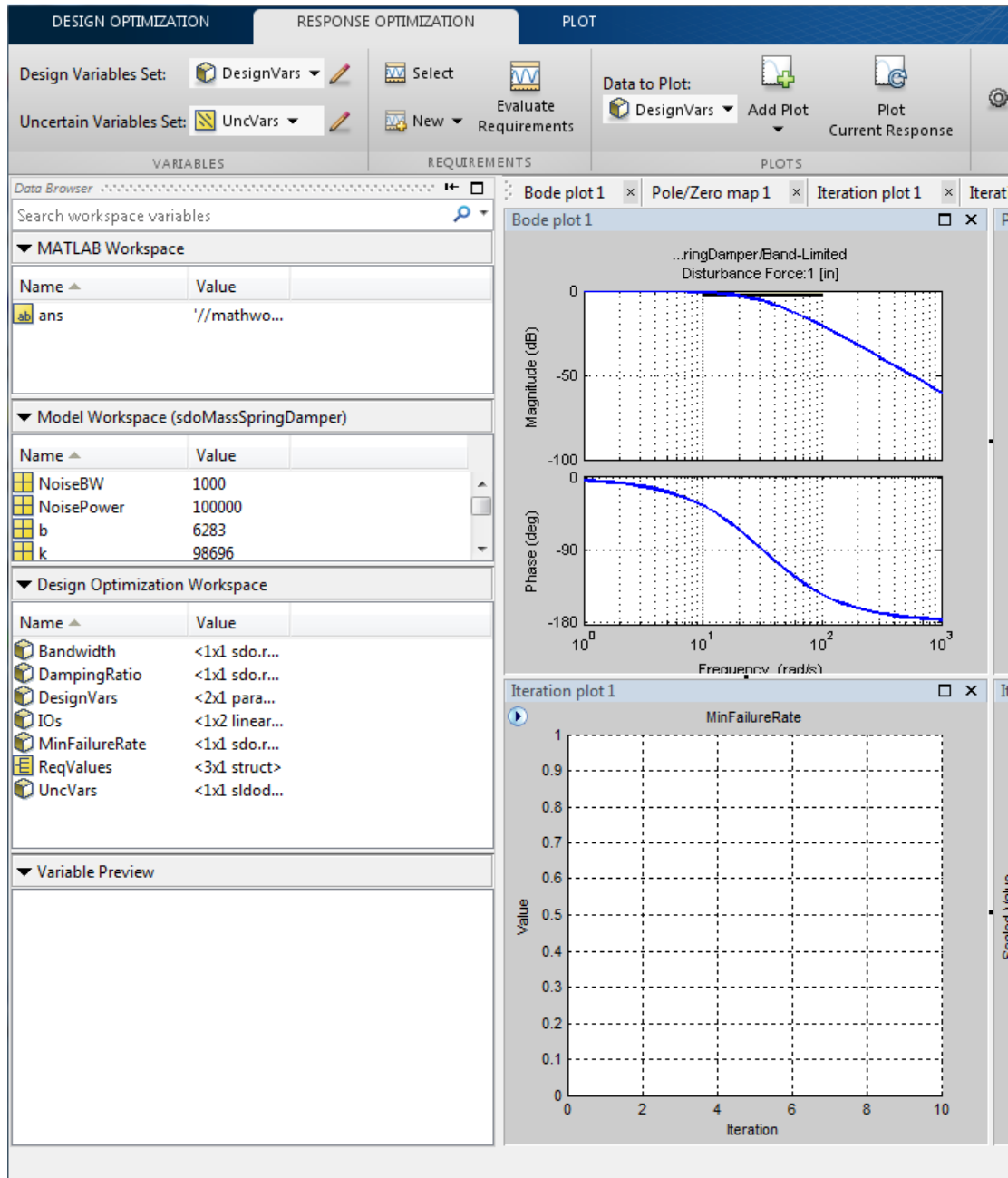
To save the initial design variable values and later compare them with the optimized values configure the optimization.

- Click **Options**.
- Select the **Save optimized variable values as new design variable set** option.

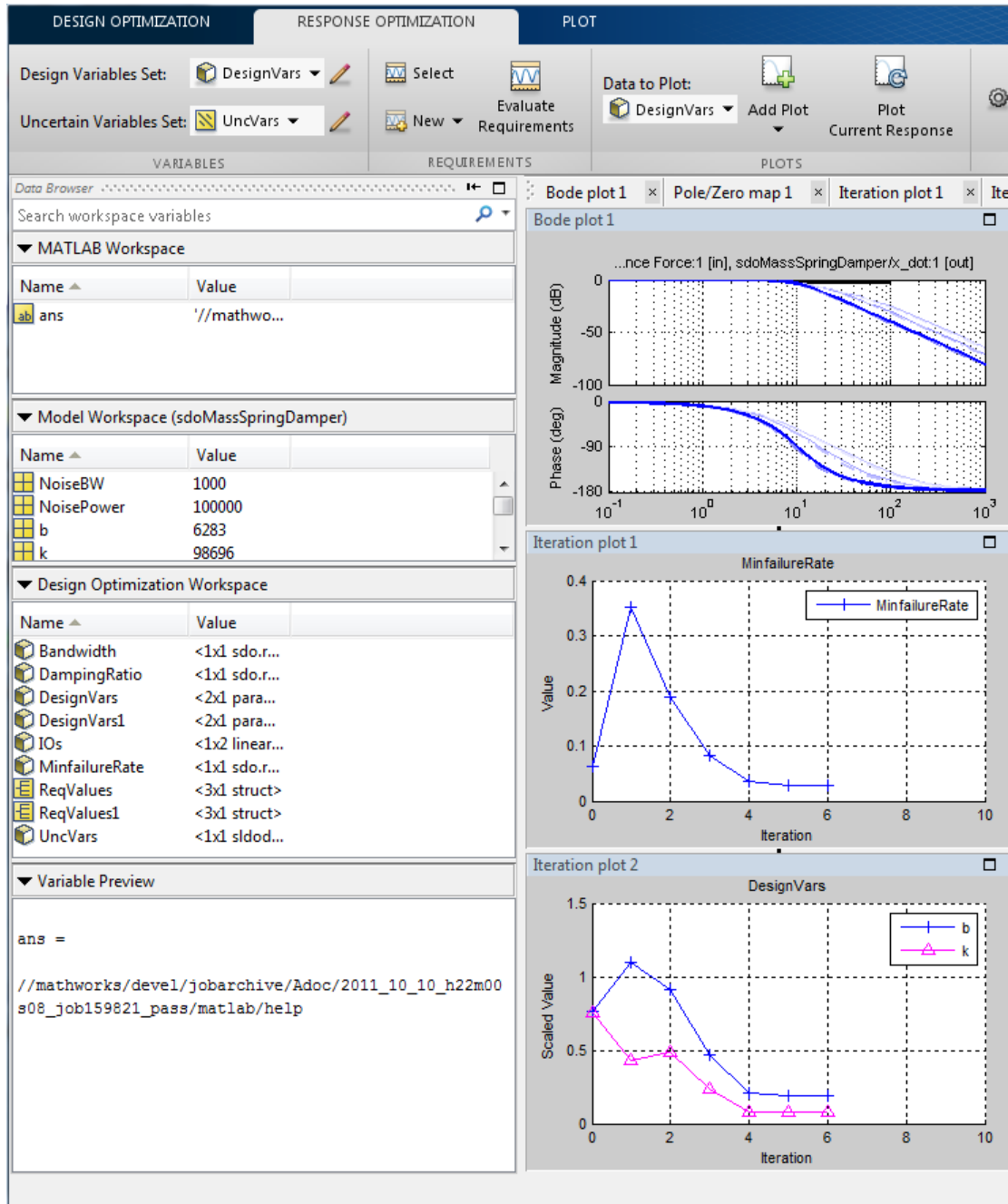
To study how the design variable values change during optimization:

- In the **Data to Plot** list, select **DesignVars**.
- In the **Add Plot** list, and select **Iteration Plot**.
- View the design variables in an appropriately scaled manner. Right-click on the **DesignVars** plot and select **Show scaled values**.

To evaluate the requirements at the initial design point, click **Evaluate Requirements**. The requirement plots are updated and a **ReqValues** variable is added to the **Design Optimization Workspace**.



To optimize the design, click **Optimize**. The plots are updated during optimization. At the end of optimization, the optimal design values are written to the `DesignVars1` variable. The requirement values for the optimized design are written to the `ReqValues1` variable.



### Analyze the Design

To compare design variables before and after optimization:

- In the **Data to Plot** list, select **DesignVars**.
- In the **Add Plot** list, select **Spider Plot**.
- To add the optimized design variables to the same plot, select **DesignVars1** in the **Design Optimization Workspace** and drag it onto the Spider plot. Alternatively, in the **Data to Plot** list, select **DesignVars1**. Then, in the **Add Plot** list, select **Spider plot 1** from the **Add to Existing Plot** section.

DESIGN OPTIMIZATION      RESPONSE OPTIMIZATION      PLOT

Design Variables Set: DesignVars      Select      Evaluate Requirements      Data to Plot: DesignVars      Add Plot      Plot Current Response

Uncertain Variables Set: UncVars      New

VARIABLES      REQUIREMENTS      PLOTS

Data Browser

Search workspace variables

▼ MATLAB Workspace

Name	Value
ans	//mathwo...

▼ Model Workspace (sdoMassSpringDamper)

Name	Value
NoiseBW	1000
NoisePower	100000
b	6283
k	98696

▼ Design Optimization Workspace

Name	Value
Bandwidth	<1x1 sdo.r...
DampingRatio	<1x1 sdo.r...
DesignVars	<2x1 para...
DesignVars1	<2x1 para...
IOs	<1x2 linear...
MinfailureRate	<1x1 sdo.r...
ReqValues	<3x1 struct>
ReqValues1	<3x1 struct>
UncVars	<1x1 sldod...

▼ Variable Preview

```
DesignVars1(1,1) =
    Name: 'b'
    Value: 1.5492e+03
    Minimum: 100
    Maximum: 10000
    Free: 1
    Scale: 8192
    Info: [1x1 struct]
```

Boode plot 1      Pole/Zero map 1      Iteration plot 1

The plot shows that the optimizer reduced both the k and b values for the optimal design.

To compare requirements before and after optimization:

- In the **Data to Plot** list, select **ReqValues**.
- In the **Add Plot** list, select **Spider Plot**.
- To add the optimized requirement values to the same plot, select **ReqValues1** in the **Design Optimization Workspace** and drag it onto the Spider plot. Alternatively, in the **Data to Plot** list, select **ReqValues1**. Then, in the **Add Plot** list, select **Spider plot 2** from the **Add to Existing Plot** section.



DESIGN OPTIMIZATION      RESPONSE OPTIMIZATION      PLOT

Design Variables Set: DesignVars      Select      Evaluate Requirements      Data to Plot: ReqValues      Add Plot      Plot Current Response

Uncertain Variables Set: UncVars      New

VARIABLES      REQUIREMENTS      PLOTS

Data Browser

Search workspace variables

▼ MATLAB Workspace

Name	Value
ans	'//mathwo...

▼ Model Workspace (sdoMassSpringDamper)

Name	Value
NoiseBW	1000
NoisePower	100000
b	6283
k	98696

▼ Design Optimization Workspace

Name	Value
Bandwidth	<1x1 sdo.r...
DampingRatio	<1x1 sdo.r...
DesignVars	<2x1 para...
DesignVars1	<2x1 para...
IOs	<1x2 linear...
MinfailureRate	<1x1 sdo.r...
ReqValues	<3x1 struct>
ReqValues1	<3x1 struct>
UncVars	<1x1 sldod...

▼ Variable Preview

```
ReqValues1 =
3x1 struct array with fields:
    Name
    Value
```

Pole/Zero map 1      Iteration plot 1      Iteration plot

ReqV

MinfailureRate 0.124

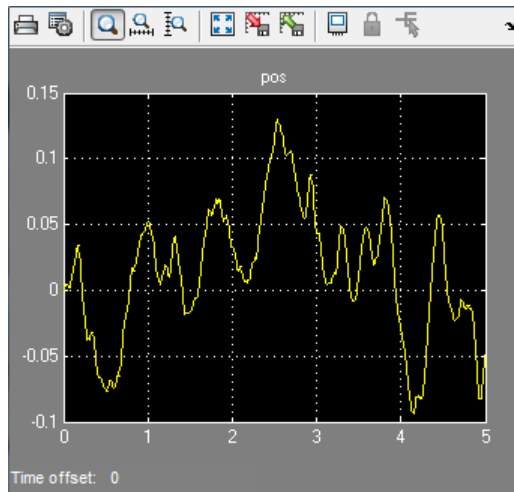
-1.44

-0.828

The plot shows that the optimal design has a lower failure rate (the MinFailureRate axis) and better satisfies the bandwidth requirement. The value plotted on the bandwidth axis is the difference between the bandwidth bound and the bandwidth value. The optimization satisfies the bound by keeping this value negative; a more negative value indicates better satisfaction of the bound.

The improved reliability and bandwidth are achieved by pushing the damping ratio closer to the damping ratio bound. The plot has two axes for the damping ratio requirement, one for each system pole, and the plotted values are the difference between the damping ratio bound and the damping ratio value. The optimization satisfies the bound by keeping this value negative.

Finally the simulated mass position is smoother than the initial position response (indication of a lower bandwidth as required) at the expense of larger position deflection.



```
% Close the model  
bdclose('sdoMassSpringDamper')
```

# Optimizing Parameters for Robustness

## In this section...

“What Is Robustness?” on page 3-157

“Sampling Methods for Uncertain Parameters” on page 3-158

“Optimize Parameters for Robustness (GUI)” on page 3-162

## What Is Robustness?

A design is *robust* when its response does not violate design requirements under model parameter variations. Your model may contain 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 use Simulink Design Optimization software to incorporate the parameter uncertainty to test the robustness of your design. When you optimize parameters for robustness, the optimization solver uses the responses computed using all the uncertain parameter values to adjust the design variable values.

You can specify the same parameter both as a design *and* uncertain variable. However, you cannot use a parameter both as a design and uncertain variable in the same optimization run. Also, you cannot add uncertainty to controller or plant parameters during optimization-based control design in the SISO Design Tool.

The uncertain variables can be scalar, vector, matrix or an expression.

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 computational 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.

**Related Examples**

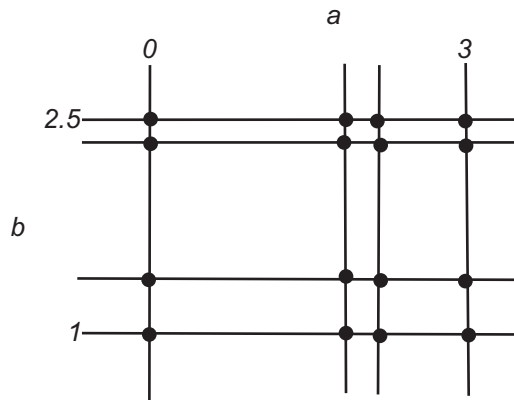
“Optimize Parameters for Robustness (GUI)” on page 3-162

**More About**

“Sampling Methods for Uncertain Parameters” on page 3-158

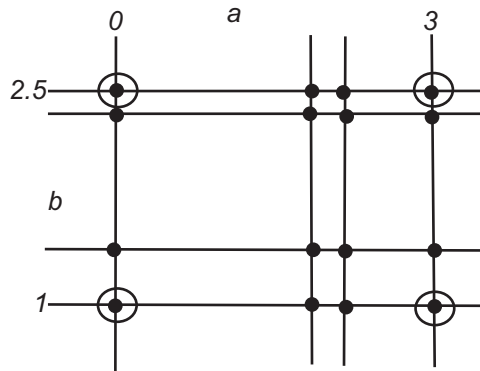
**Sampling Methods for Uncertain Parameters**

Sample values for uncertain parameters are a vector of numerical values. You can specify the vector yourself or generate a vector of random numbers using the software. The sample values you specify can be uniformly distributed or random. For example, four sample values for two uncertain parameters  $a$  and  $b$  in the range  $[0\ 3]$  and  $[1\ 2.5]$  may look like the following figure.



There are two methods to determine the number of sample values to use during optimization:

- Only the combination of minimum and maximum values (circled)



- Combination of the entire set of values (all solid dots in the previous figure)

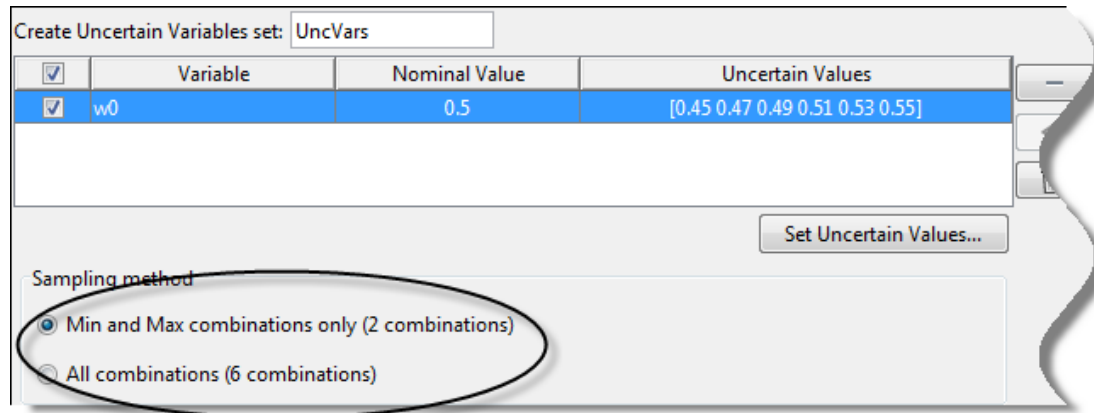
---

**Tip** Using only the minimum and maximum values during optimization increases the computation speed when compared to using the entire set of values.

---

For the previous example, there are 4 combinations using the minimum and maximum values and 16 combinations if you use all sample values.

In the Design Optimization tool, you specify the sampling method using the options as shown in the following figure.



#### Related Examples

“Optimize Parameters for Robustness (GUI)” on page 3-162

#### More About

- “What Is Robustness?” on page 3-157



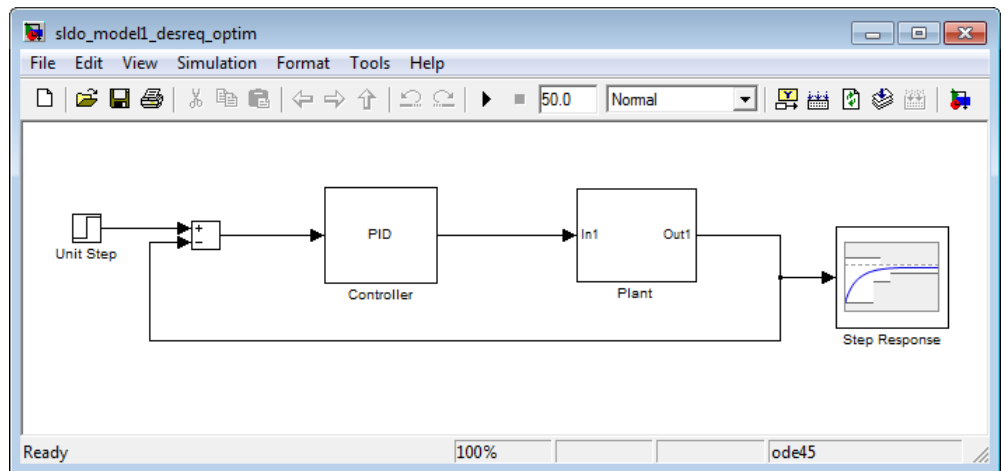
## Optimize Parameters for Robustness (GUI)

This example shows how to optimize parameters for model robustness.

1 Load a saved Design Optimization tool session.

```
load sldo_model1_desreq_optim_sdoession;  
sdotool(SDOSessionData);
```

The following Simulink model opens.

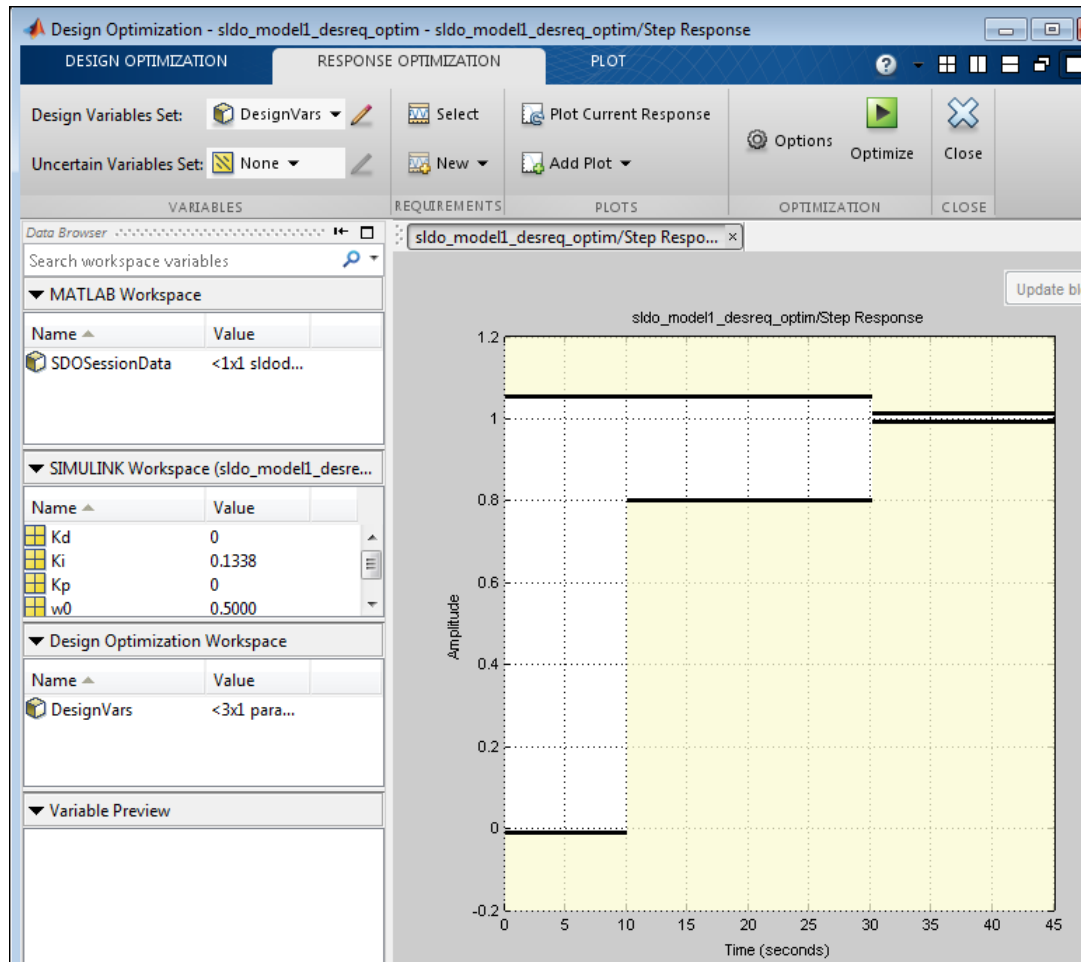


The Simulink model parameters  $K_p$ ,  $K_i$  and  $K_d$  have already been optimized to meet the following step response requirements:

- Maximum overshoot of 5%
- Maximum rise time of 10 seconds
- Maximum settling time of 30 seconds

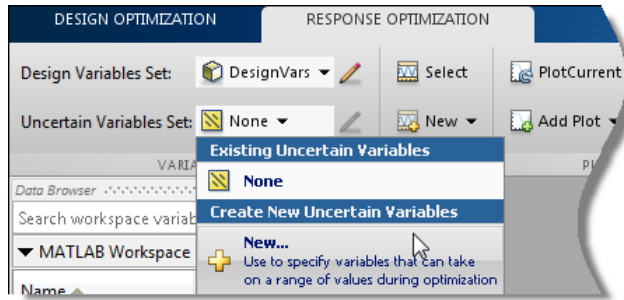
The command also opens the Design Optimization tool.



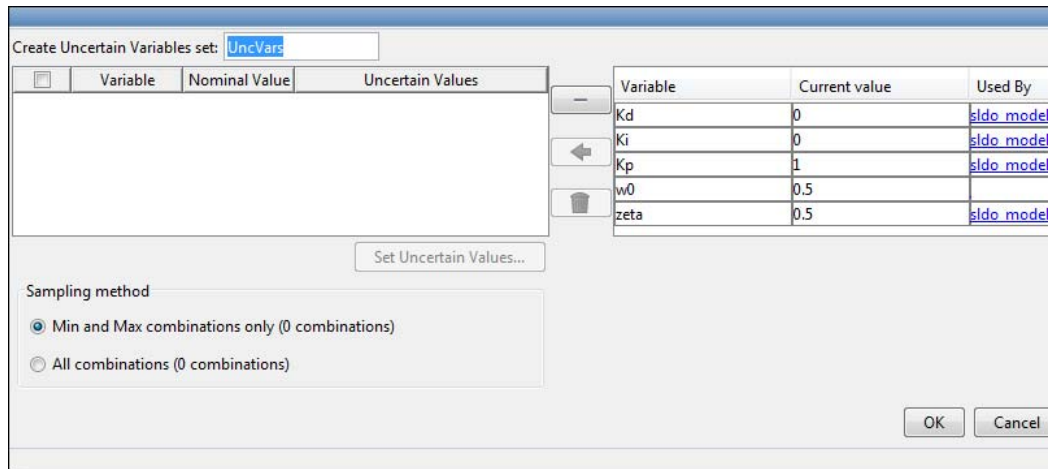


2 Specify parameter uncertainty.


- a In the **Uncertain Variables Set** drop-down list, select **New**.

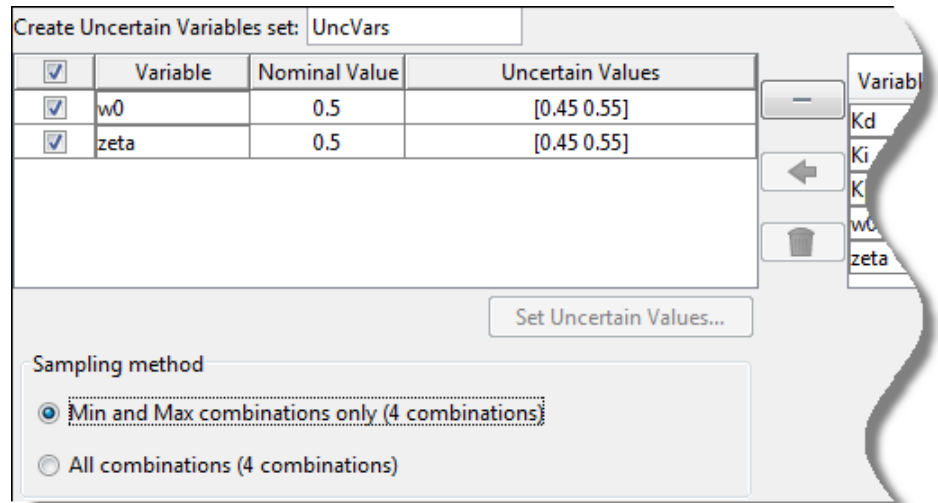


A window opens where you specify uncertain variables.



- b Click w0 and zeta to select them.

- c Click  to add the selected parameters to an uncertain variables set.



The software displays the following parameter settings:

- **Variable** — Parameter name
- **Nominal Value** — Nominal value of the parameters as specified in the Simulink model
- **Uncertain Values** — Values that the uncertain parameter can take. By default, the maximum and minimum values vary by 10% of the nominal value.

The total number of sample values to use during optimization is a combination of the maximum and minimum values of the uncertain parameters.

The check-box indicates that the parameter is included in the uncertain variable set. The default uncertain variable set name is `UncVars`.

Click **OK**. A new variable `UncVars` appears in **Design Optimization Workspace** of the Design Optimization tool.

### Specify Random Values

Instead of specifying sample values, you can auto-generate random values in a specific range. Select a parameter and click **Set Uncertain Values**.

Create Uncertain Variables set:

<input checked="" type="checkbox"/>	Variable	Nomi...	Uncertain Values
<input checked="" type="checkbox"/>	w0	0.5	[0.45 0.55]

A window opens where you specify the range and the number of samples.

Generate uncertain values for 'w0'

Nominal value: 0.5

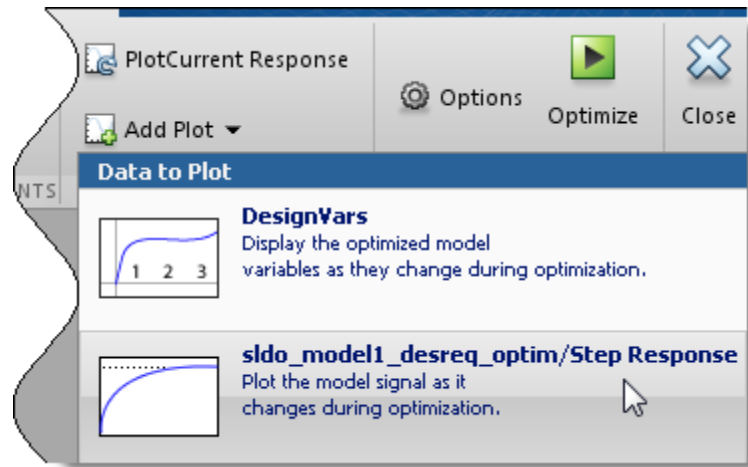
Distribution: Uniform

Minimum:

Maximum:

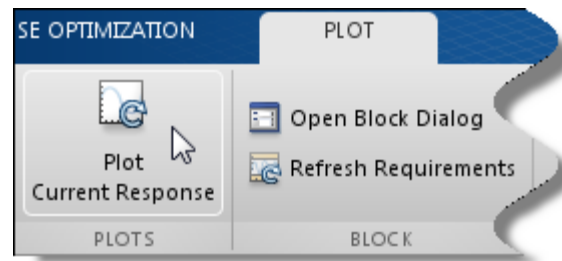
Number of samples:

- 3** Test the model robustness to the uncertain parameters.
  - a** In the **Add Plot** drop-down list, select **sldo\_model1\_desreq\_optim\_Step Response**.

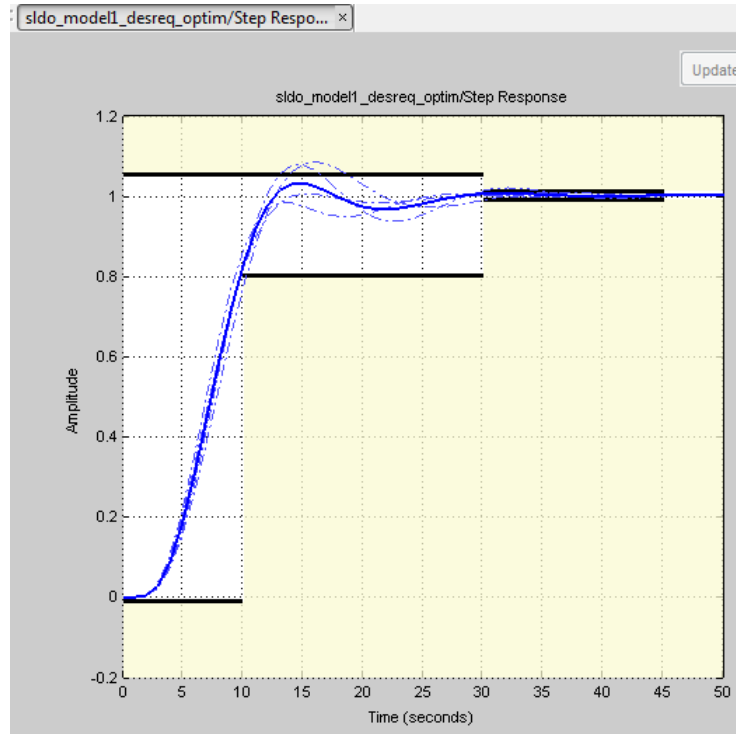


A plot window displaying the requirements specified in the Step Response block in the model opens.

- b** Click **Plot Current Response**.



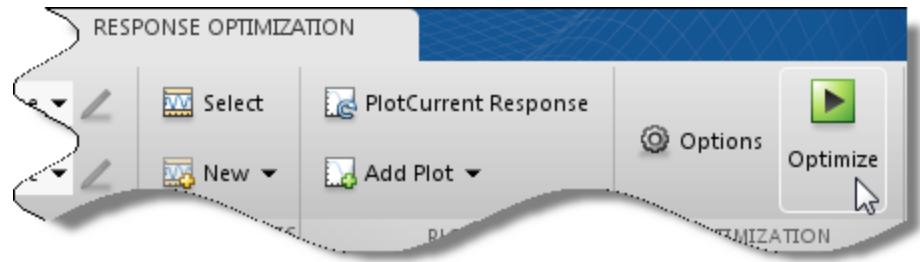
The plot updates.



- The solid curve corresponds to the model response computed using the optimized parameters and nominal values of the uncertain parameter.
- The four dashed curves correspond to the model response with the minimum and maximum values of the uncertain parameters.

The dashed plot lines show that the response during the period of 10 to 20 seconds violates the design requirements.

**4** Optimize the parameters for model robustness. Click **Optimize**.



The Optimization Progress window opens which displays the optimization iterations.

After the optimization completes, the message **Optimization converged** indicates that the final model response computed by varying the uncertain parameters meets the specified design requirements.

 A screenshot of the 'Optimization Progress' window. The window title is 'sldo\_model1\_desreq\_optim/Step Respo...' and 'Optimization Progress'. It contains a table with the following data:
 

Iter	F-count	f(x)	max constraint
0	5	0	0.0317
1	11	0	0.0038
2	18	0	2.4458e-04

 Below the table, there is a log of messages:
 

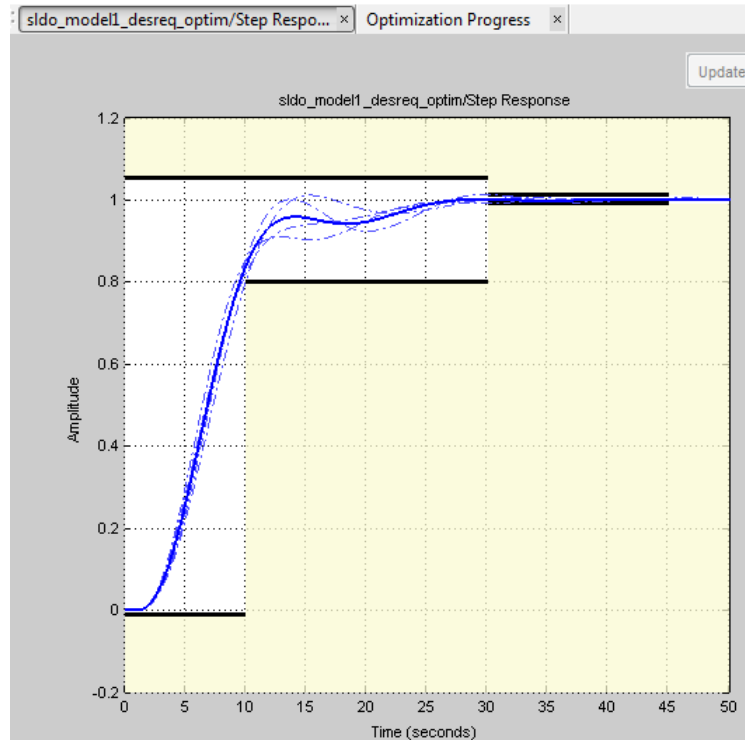
```

  Optimization started 06-Jun-2011 16:01:53
  Optimization converged, 06-Jun-2011 16:15:54

  Optimized variable values written to 'DesignVars' in SDOTOOL workspace
  'sldo_model1_desreq_optim' updated with optimized values
  
```

## 5 Examine the responses.

**Tip** To view only the final responses of the model, right-click the white area in the plot and uncheck **Responses > Show Iteration Responses**.



The final responses appear as the thick solid and dashed curves. The nominal and uncertain responses with parameter variations now meet the design requirements.

#### More About

- “What Is Robustness?” on page 3-157
- “Sampling Methods for Uncertain Parameters” on page 3-158



# Accelerating Model Simulations During Optimization

## In this section...

“About Accelerating Optimization” on page 3-171

“Limitations” on page 3-171

“Setting Accelerator Mode for Response Optimization” on page 3-171

## 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

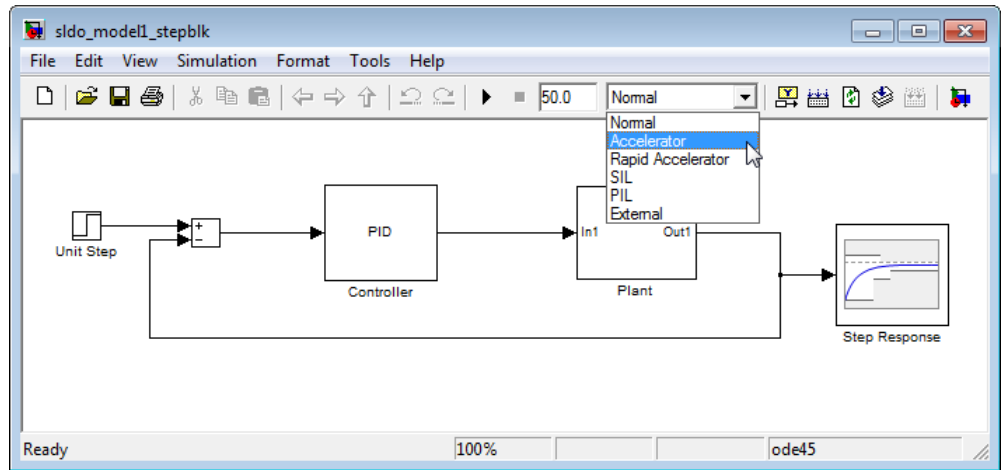
You cannot use the `Accelerator` mode if your model contains algebraic loops. If the model contains MATLAB function blocks, you must either remove them or replace them with `Fcn` blocks.

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.

---

## Speedup Using Parallel Computing

**In this section...**

“When to Use Parallel Computing for Response Optimization” on page 3-173

“How Parallel Computing Speeds Up Optimization” on page 3-174

### 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 3-174.

See “How to Use Parallel Computing” on page 3-178 for more information on how to configure your system, and use parallel computing.

## 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 topics describe which simulations are distributed and the expected speedup using parallel computing:

- “Parallel Computing with the Gradient descent Method” on page 3-174
- “Parallel Computing with the Pattern search Method” on page 3-175

### 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:

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

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 3-191 in “Troubleshooting Optimization Results” on page 3-186.

---

For a demo on the performance improvement achieved with the **Pattern search** method, see the **Improving Optimization Performance Using Parallel Computing** demo.

## How to Use Parallel Computing

In this section...
“Configure Your System for Parallel Computing” on page 3-178
“Specify Model Dependencies” on page 3-179
“Optimize Design Using Parallel Computing (GUI)” on page 3-180
“Optimize Design Using Parallel Computing (Code)” on page 3-184

### Configure Your System for Parallel Computing

To use parallel computing, you must first configure your system as described in the following topics:

- “Configuring Parallel Computing on Multicore Processors” on page 3-178
- “Configuring Parallel Computing on Multiprocessor Networks” on page 3-178

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 “Define Cluster Profiles” 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 “Apply Cluster Profiles 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.

## Specify 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 all model dependencies are included and the remote workers can access them. You can add additional dependencies and make the dependencies accessible to remote workers, as described in the following topics:

- “Specifying Additional Path and File Dependencies” on page 3-179
- “Making Model Dependencies Accessible to Remote Workers” on page 3-180

## 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 “Optimize Design Using Parallel Computing (GUI)” on page 3-180, and “Optimize Design Using Parallel Computing (Code)” on page 3-184.
- 2 Add the file dependencies, as described in “Configuring Parallel Computing on Multiprocessor Networks” on page 3-178.

### **Making Model Dependencies Accessible to Remote Workers**

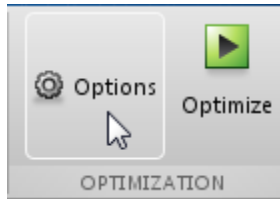
If your model has dependencies that the remote MATLAB workers cannot access directly, modify the list of dependencies to make them accessible. For example, remote workers cannot access model dependencies on your local drive. Update the list of paths so that the workers can access them, as described in “Optimize Design Using Parallel Computing (GUI)” on page 3-180, and “Optimize Design Using Parallel Computing (Code)” on page 3-184.

### **Optimize Design Using Parallel Computing (GUI)**

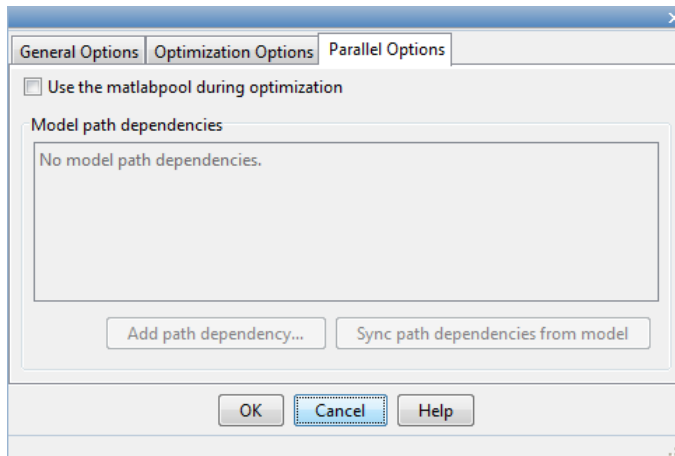
After you configure your system for parallel computing, as described in “Configure Your System for Parallel Computing” on page 3-178, you can use the GUI to optimize the model response:

- 1 Open the Design Optimization tool for a model.
- 2 Configure the design requirements, design variables and optionally optimization settings. See “Specifying Time Domain Design Requirements (GUI)” on page 3-16, “Design Variables (GUI)” on page 3-54 and “Optimization Options” on page 3-64.

**3** In the Design Optimization tool, , click **Options**.



Select the **Parallel Options** tab.



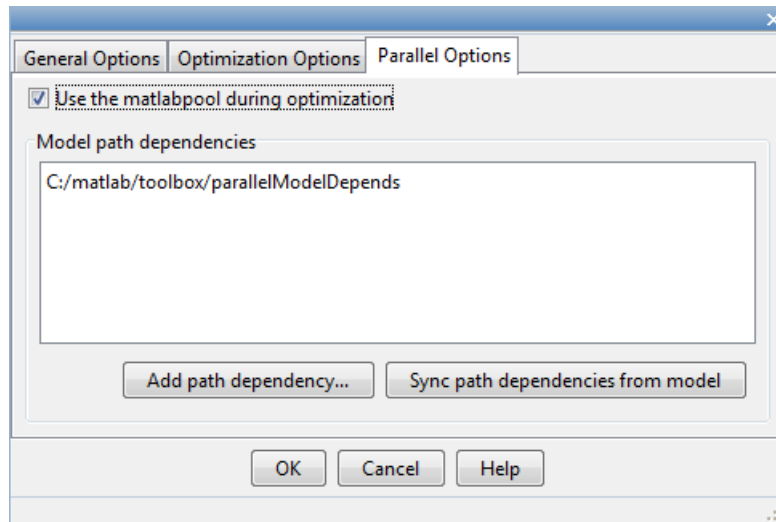
**4** Select **Use the matlabpool during optimization**.

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 “Specify Model Dependencies” on page 3-179, the automatic path dependencies check may not detect all the path dependencies in your model.

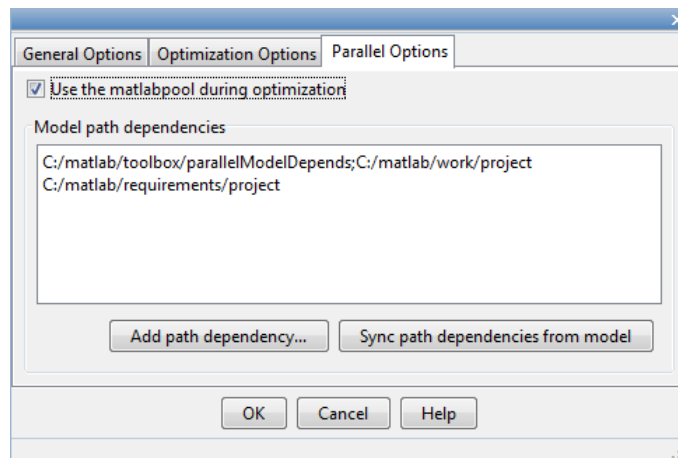
---



5 (Optional) Add the path dependencies that the automatic check does not detect.

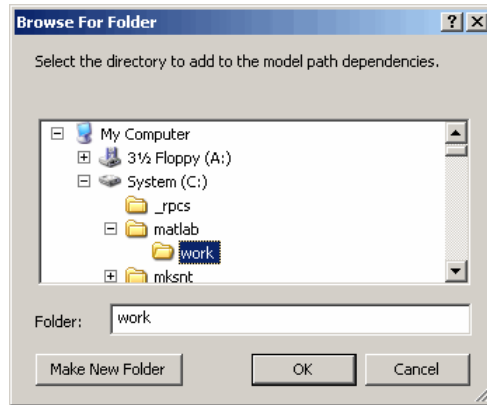
- a Specify the paths in the **Model path dependencies** list box.

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) 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 Design Optimization tool, click **Optimize** 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 3-190.

## Optimize Design Using Parallel Computing (Code)

After you configure your system for parallel computing, as described in “Configure Your System for Parallel Computing” on page 3-178, you can optimize your model’s response using the command-line functions. To learn more about response optimization using parallel computing, see “Speedup Using Parallel Computing” on page 3-173.

To use parallel computing for response optimization at the command line:

- 1 Open the model you want to optimize.
- 2 Specify design requirements and design variables.
- 3 Enable parallel computing.

```
sim_opt = sdo.OptimizeOptions;  
sim_opt.UseParallel = 'always';
```

- 4 Find the model path dependencies.

```
dirs = sdo.getModelDependencies(modelname)
```

This command returns the model path dependencies in your Simulink model.

---

**Note** As described in “Specify Model Dependencies” on page 3-179, the `sdo.getModelDependencies` command may not detect all the path dependencies in your model.

---

- 5 (Optional) Modify `dirs` to include path dependencies that `sdo.getModelDependencies` 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 path dependencies for optimization.

```
sim_opt.ParallelPathDependencies = dirs;
```

**8** Run the optimization.

For more information on how to troubleshoot problems that occur during optimization using parallel computing, see “Optimization Speed and Parallel Computing” on page 3-190 in “Troubleshooting Optimization Results” on page 3-186.

## Refining and Troubleshooting Optimization Results

### Troubleshooting Optimization Results

When optimizing model parameters, the optimization solver 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 3-186
- “Optimization Convergence” on page 3-187
- “Optimization Speed and Parallel Computing” on page 3-190
- “Undesirable Parameter Values” on page 3-192
- “Reverting to Initial Parameter Values” on page 3-194

### Optimization Does Not Make Progress

- “Should I worry about the scale of my responses and how constraints and design requirements are discretized?” on page 3-186
- “Why don’t the responses and parameter values change at all?” on page 3-187
- “Why does the optimization stall?” on page 3-187

**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.



**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 optimizing the response of a Simulink model, you could also try removing nonlinear blocks such as Quantizer or Dead Zone.

**Why does the optimization stall?** When optimizing 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 3-188
- “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 3-188
- “What to do if the optimization takes a long time to converge even though it is close to a solution?” on page 3-189

- “What to do if the response becomes unstable and does not recover?” on page 3-189

### **What to do if the optimization does not get close to an acceptable solution?.**

- 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 design variables 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 Design Optimization tool, click **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 3-190
- “Why are the optimization results with and without using parallel computing different?” on page 3-191
- “Why do I not see the optimization speedup I expected using parallel computing?” on page 3-191
- “Why does the optimization using parallel computing not make any progress?” on page 3-192
- “Why does the optimization using parallel computing not stop when I click the **Stop optimization** button?” on page 3-192

### How can I speed up the optimization?.

- The optimization time is dominated by the time it takes to simulate the model. When optimizing 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 3-171.

---

- 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 3-175.

**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 3-175.

**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 3-180.

**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 `Stop`, 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 3-193
- “What to do if the optimization violates bounds on parameter values?” on page 3-193

### **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:
  - Dialog box to select design variables (in Design optimization tool)
  - **Compensators** pane (in a SISO Design Task)

This information helps guide the optimization method towards a reasonable solution.

- 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 clicking **Optimize** in the Design Optimization tool. 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?.

- Before running an optimization, do one of the following:
  - In the Design Optimization tool, click **Options**. Uncheck **Update model at end of optimization** in the **General Options** tab.
  - In the Design Optimization tool, click **Options**. Select **Save optimized variable values as new design variable set** in the **General Options** tab.
  - Make a copy of the design variable set in the **Design Optimization Workspace**.

If you want to revert to the initial parameter values after the optimization terminates or you stop the optimization by clicking **Stop**, select the design variable that contains the initial values in the **Design Variable Set**

drop-down list and click  adjacent to **Design variables Set**. Select the design variables in the dialog box and click **Update model variable values** to revert the model parameters to their original values.



- 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.

## Managing Design Optimization Tool Session

### In this section...

“Saving a Session” on page 3-196

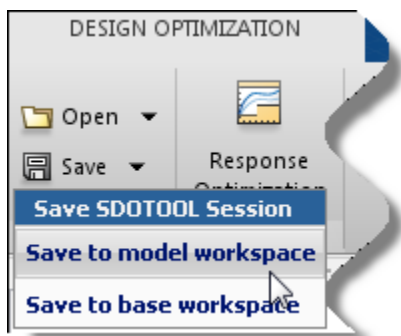
“Loading a Session” on page 3-196

### Saving a Session

Saving a session lets you reuse your settings and optimization results later. These settings include design requirements, design and uncertain variables, plots and optimization settings. Each Design Optimization tool session is associated with a Simulink model.

You can save the session as a MAT-file or workspace variable:

- To save the session as a MAT-file, click **Save** in the **Design Optimization** tab. A window opens where you specify the MAT-file name.
- To save the session as a model or MATLAB workspace variable, select **Save to model workspace** or **Save to base workspace** in the **Save** drop-down list.



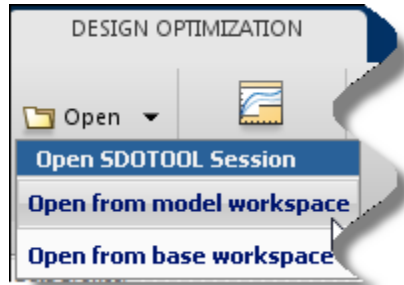
### Loading a Session

You can load a saved MAT-file or workspace session:

- 1 Open a Design Optimization tool for the model.

- 2** To load a MAT-file, click **Open** in the **Design Optimization** tab. A window opens where you select the MAT-file to load.

To load a workspace variable, select **Open from model workspace** or **Open from base workspace** in the **Open** drop-down list.





# Optimization-Based Control Design

---

- “Overview of Optimization-Based Compensator Design” on page 4-2
- “Time Domain Design Requirements in Simulink” on page 4-4
- “Frequency-Domain Design Requirements in Simulink” on page 4-20
- “Time- and Frequency-Domain Requirements in SISO Design Tool” on page 4-42
- “Time-Domain Simulations in SISO Design Tool” on page 4-47
- “Designing Optimization-Based Controllers for LTI Systems” on page 4-48
- “Designing Linear Controllers for Simulink Models” on page 4-70

# Overview of Optimization-Based Compensator Design

You can design optimization-based controllers for Simulink models to meet time and frequency-domain design requirements, as described in Chapter 3, “Response Optimization”.

If you have Control System Toolbox software installed, you can also 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 to meet time- and frequency-domain requirements.

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. For an example, see “Example — Frequency-Domain Optimization for LTI System” on page 4-49.
- 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. For an example, see “Design an Optimization-Based PID Controller for a 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.

## Time Domain Design Requirements in Simulink

### In this section...

“Specifying Piecewise-Linear Lower and Upper Bounds” on page 4-4

“Specifying Step Response Characteristics” on page 4-35


“Track Reference Signals” on page 4-13

“Specifying Custom Requirements” on page 4-38

“Editing Design Requirements” on page 4-17

### Specifying Piecewise-Linear Lower and Upper Bounds

To specify upper and lower bounds on a signal:

- 1 In the Design Optimization tool, select **Signal Bound** in the **New** drop-down list. A window opens where you specify upper or lower bounds on a signal.
- 2 Specify a requirement name in the **Name** box.
- 3 Select the requirement type using the **Type** list.
- 4 Specify the edge start and end times and corresponding amplitude in the **Time (s)** and **Amplitude** columns.
- 5 Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

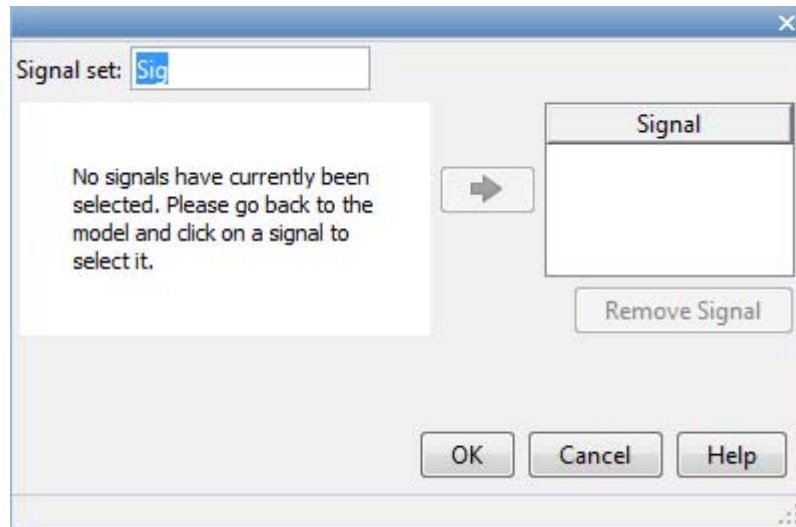
- 6 In the **Select Signals to Bound** area, select a logged signal to apply the requirement to.

If you have already selected signals, as described in “Specify Signals to Log (GUI)” on page 3-14, they appear in the list. Select the corresponding check-box.


If you haven’t selected a signal to log:



- a Click . A window opens where you specify the logged signal.
- b In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.

Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- 7 Click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the requirement also appears in the Design Optimization tool window.

- 8 (Optional) In the graphical display, you can:
  - “Moving Constraints” on page 4-6

- “Split and Join Constraints” on page 4-8

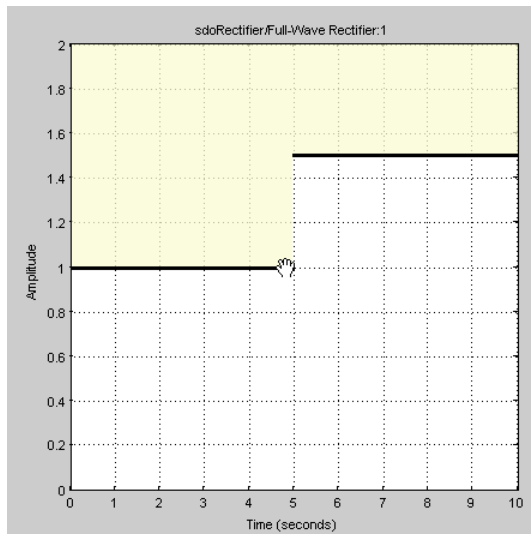
Alternatively, you can add a Check Custom Bounds block to your model to specify piecewise-linear bounds.

### Moving Constraints

- “Moving Constraints Graphically” on page 4-6
- “Positioning Constraints Exactly” on page 4-7

Constraint-bound edges define time-domain constraints you would like to place on a particular signal in your model. You can position these edges, which appear as a yellow shaded region bordered by a black line, graphically or exactly.

**Moving Constraints Graphically.** Use the mouse to click and drag edges in the amplitude versus time plot, as shown in the following figure.



- To move a constraint edge boundary or to change the slope of a constraint edge, position the pointer over a constraint edge 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 edges on either side of the boundary might not maintain their slopes.

- To move an entire constraint edge up, down, left, or right, position the mouse pointer over the edge 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 edges on either side of the boundary might not maintain their slopes.

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

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

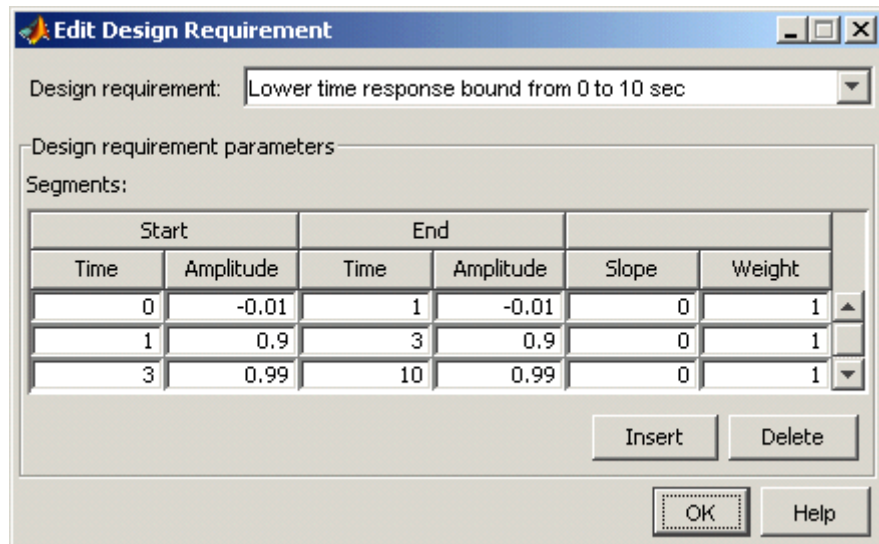
---

**Note** You can move a lower bound constraint edge above an upper bound constraint edge, or vice versa, but this produces an error when you attempt to run the optimization.

---

**Positioning Constraints Exactly.** To position a constraint edge exactly:

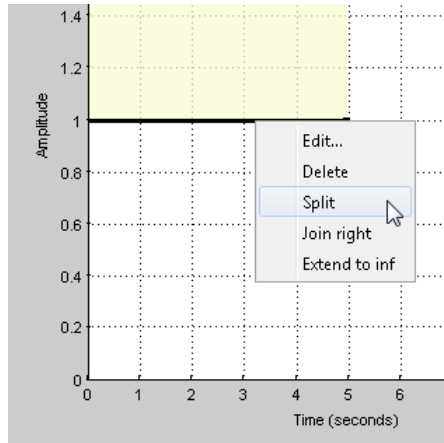
- 1 Position the pointer over the edge you want to move and right-click. Select **Edit** to open the Edit Design Requirement dialog box.



- 2 Specify the position of each constraint edge in the **Time** and **Amplitude** columns.

### Split and Join 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 segment too the left or right respectively.

## Specifying Step Response Characteristics

To specify step response characteristics:

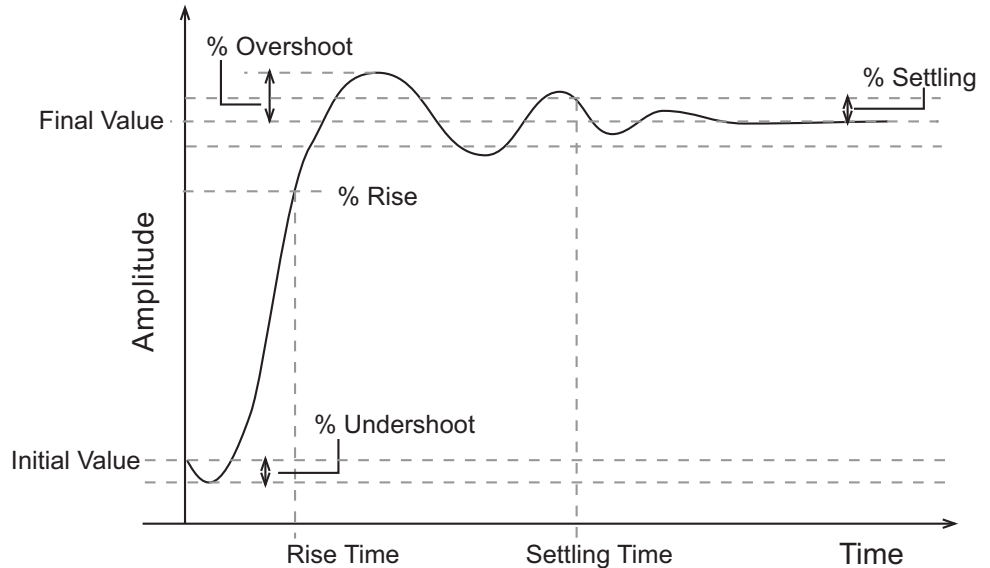
- 1 You can apply this requirement to either a signal or a linearization of your model.

In the Design Optimization Tool, click **New**. To apply this requirement to a signal, select the **Step Response Envelope** entry in the **New Time Domain Requirement** section of the **New** list. To apply this requirement to a linearization of your model, select the **Step Response Envelope** entry in the **New Frequency Domain Requirement** section of the **New** list. The latter option requires Simulink Control Design software.

A window opens where you specify the step response requirements on a signal, or system.

- 2 Specify a requirement name in the **Name** box.

3 Specify the step response characteristics:



- **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
- **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.

#### 4 Specify the signals or systems to be bound.


You can apply this requirement to a model signal or to a linearization of your Simulink model (requires Simulink Control Design software).

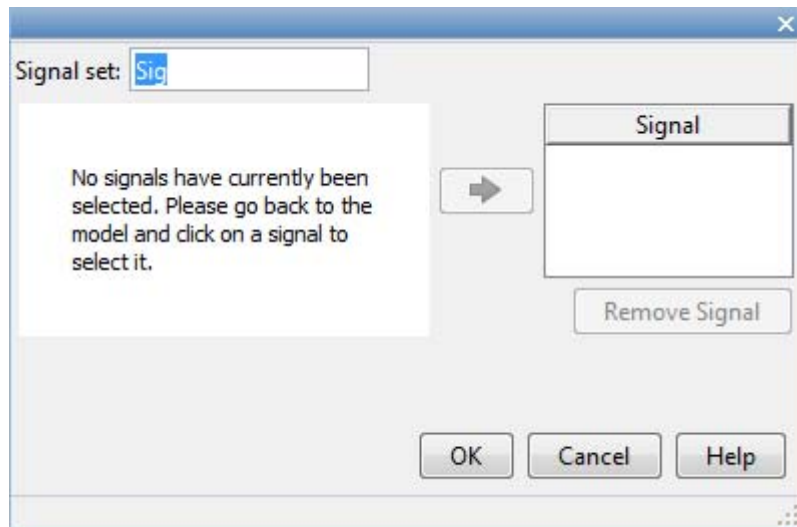
- Apply this requirement to a model signal:

In the **Select Signals to Bound** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, it appears in the list. Select the corresponding check-box.

If you haven't selected a signal to log:

- Click . A window opens where you specify the logged signal.
- In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.


Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- Apply this requirement to a linear system.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5 Click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the requirement also appears in the Design Optimization tool window.

Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.



## See Also

“Optimize Controller Parameters to Meet Step Response Requirements (GUI)”

## Track Reference Signals

Use reference tracking to force a model signal to match a desired signal.

To track a reference signal:


- 1** In the Design Optimization tool, select **Signal Tracking** in the **New** drop-down list. A window opens where you specify the reference signal to track.
- 2** Specify a requirement name in the **Name** box.
- 3** Define the reference signal by entering vectors, or variables from the workspace, in the **Time vector** and **Amplitude** fields.

Click **Update reference signal data** to use the new amplitude and time vector as the reference signal.

- 4** Specify how the optimization solver minimizes the error between the reference and model signals using the **Tracking Method** list:
  - SSE — Reduces the sum of squared errors
  - SAE — Reduces the sum of absolute errors
- 5** In the **Specify Signal to Track Reference Signal** area, select a logged signal to apply the requirement to.


If you already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, they appear in the list. Select the corresponding check-box.

If you haven't selected a signal to log:

- a** Click . A window opens where you specify the logged signal.
- b** In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.

Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- e Select the check-box corresponding to the signal and click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the signal bound also appears in the Design Optimization tool window.

---

**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 3-66.

---


Alternatively, you can use the Check Against Reference block to specify a reference signal to track.


### See Also

“Optimize Controller Parameters to Track Reference Signal (GUI)”

## Specifying Custom Requirements

To specify custom requirements such as minimizing system energy:

- 1 In the Design Optimization tool, select **Custom Requirement** in the **New** drop-down list. A window opens where you specify the custom requirement.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the requirement type using the **Type** list.
- 4 Specify the name of the function that contains the custom requirement in the **Function** box. The field must be specified as a function handle using `@`. The function must be on the MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is `myCustomRequirement`.

- 5 (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design software), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

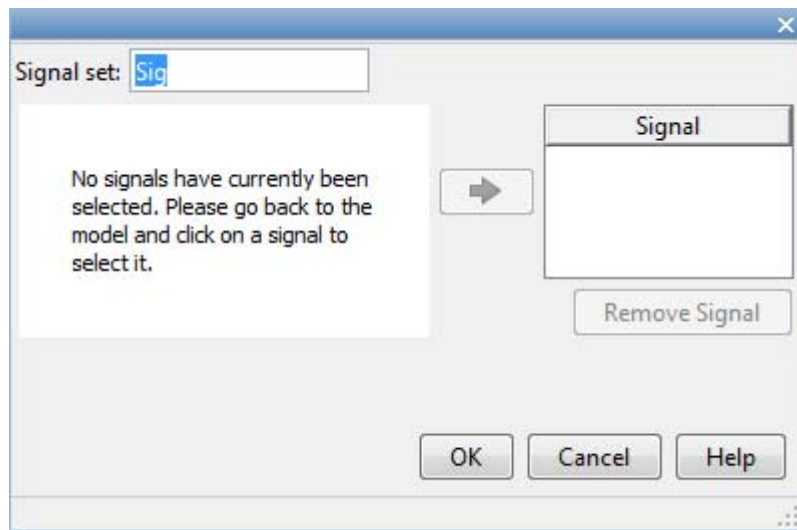
- Apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, it appears in the list. Select the corresponding check-box.

If you haven't selected a signal to log:

- a Click . A window opens where you specify the logged signal.
- b In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.


Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- Apply this requirement to a linear system.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

## 6 Click OK.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.

## See Also

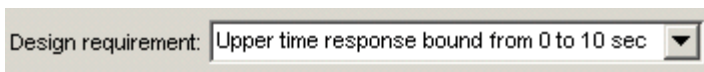
- “Design Optimization to Meet a Custom Objective Using the GUI” on page 3-103
- “Design Optimization to Meet Custom Signal Requirements (GUI)” on page 3-133

## Editing Design Requirements

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.

### 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.

**Edit Design Requirement Dialog Box Parameters (Continued)**

<b>Parameter</b>	<b>Found in</b>	<b>Description</b>
<b>Slope (1/s)</b>	Upper and lower time response bounds	Defines the slope, in 1/s, of a constraint segment. It is an alternative method of specifying the magnitude values. Entering a new <b>Slope</b> value changes any previously defined magnitude values.
<b>Final value</b>	Step response bounds	Defines the input level after the step occurs.
<b>Rise time</b>	Step response bounds	Defines a constraint segment for a particular rise time.
<b>% Rise</b>	Step response bounds	The percentage of the step's range used to describe the rise time.
<b>Settling time</b>	Step response bounds	Defines a constraint segment for a particular settling time.
<b>% Settling</b>	Step response bounds	The percentage of the final value that defines the settling region used to describe the settling time.
<b>% Overshoot</b>	Step response bounds	
<b>% Undershoot</b>	Step response bounds	Defines the constraint segments for a particular percent undershoot.

## Frequency-Domain Design Requirements in Simulink

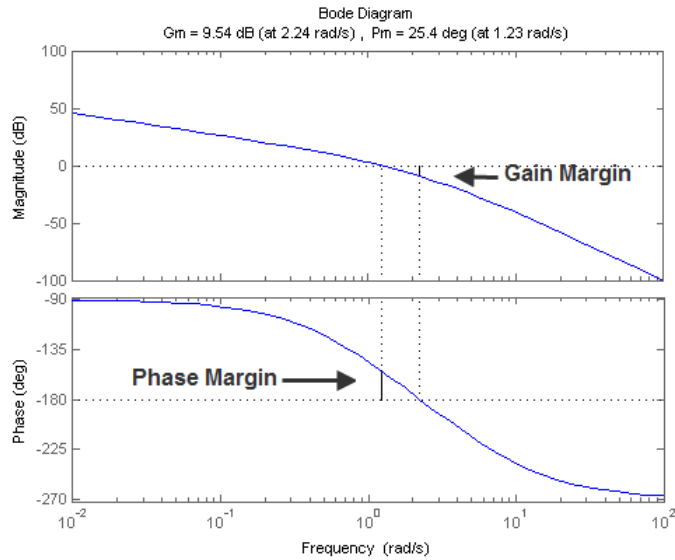
In this section...
“Specifying Lower Bounds on Gain and Phase Margin” on page 4-20
“Specifying Piecewise-Linear Lower and Upper Bounds on Frequency Response” on page 4-22
“Specifying Bound on Closed-Loop Peak Gain” on page 4-24
“Specifying Lower Bound on Damping Ratio” on page 4-26
“Specifying Upper and Lower Bounds on Natural Frequency” on page 4-28
“Specifying Upper Bound on Approximate Settling Time” on page 4-30
“Specifying Piecewise-Linear Upper and Lower Bounds on Singular Values” on page 4-32
“Specifying Step Response Characteristics” on page 4-35
“Specifying Custom Requirements” on page 4-38

### Specifying Lower Bounds on Gain and Phase Margin

To specify lower bounds on the gain and phase margin of a linear system:

- 1** In the Design Optimization tool, select **Gain and Phase Margin** in the **New** list. A window opens where you specify lower bounds on the gain and phase margin of your linear system.
- 2** Specify a requirement name in **Name**.
- 3** Specify bounds on the gain margin or phase margin, or both.





- **Gain margin** — Amount of gain increase or decrease required to make the loop gain unity at the frequency where the phase angle is  $-180^\circ$ .
- **Phase margin** — Amount of phase increase or decrease required to make the phase angle  $-180^\circ$  when the loop gain is 1.0


To specify a lower bound on the gain margin or phase margin, or both, select the corresponding check box and enter the lower bound value.

- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

### 5 Click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the requirement also appears in the Design Optimization tool window.

### 6 (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8


Alternatively, you can use the Check Gain and Phase Margins block to specify bounds on the gain and phase margin. (Requires Simulink Control Design.)

## Specifying Piecewise-Linear Lower and Upper Bounds on Frequency Response

To specify upper or lower bounds on the magnitude of a system response:

- 1 In the Design Optimization tool, select **Bode Magnitude** in the **New** list. A window opens where you specify the lower or upper bounds on the magnitude of the system response.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the requirement type using the **Type** list.
- 4 Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns.

**5** Insert or delete bound edges.

Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

**6** In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a** Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

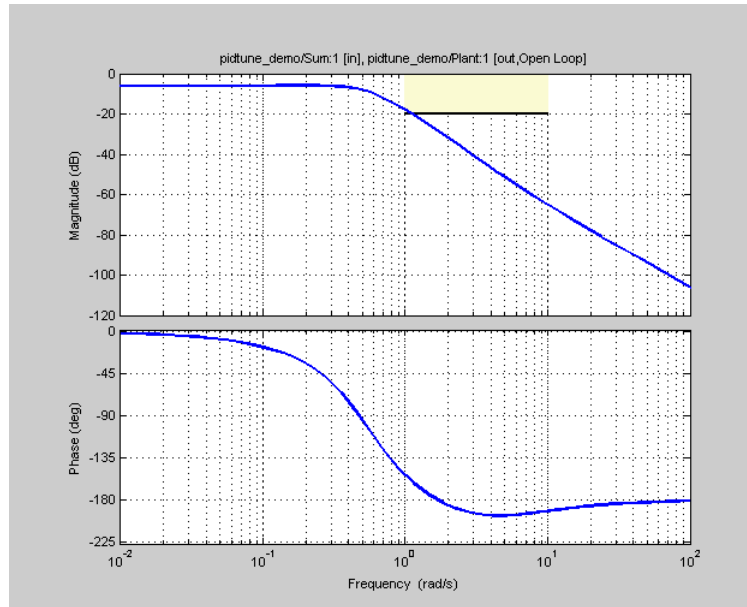
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

**7** Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window. A graphical display of the requirement also appears in the Design Optimization tool window.



**8** (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Bode Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

### Specifying Bound on Closed-Loop Peak Gain

To specify an upper bound on the closed-loop peak response of a system:


- 1** In the Design Optimization tool, select **Closed-Loop Peak Gain** in the **New** list. A window opens where you specify an upper bound on the closed-loop peak gain of the system.
- 2** Specify a requirement name in the **Name** box.

- 3** Specify the upper bound on the closed-loop peak gain in the **Closed-Loop peak gain** box.
- 4** In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a** Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

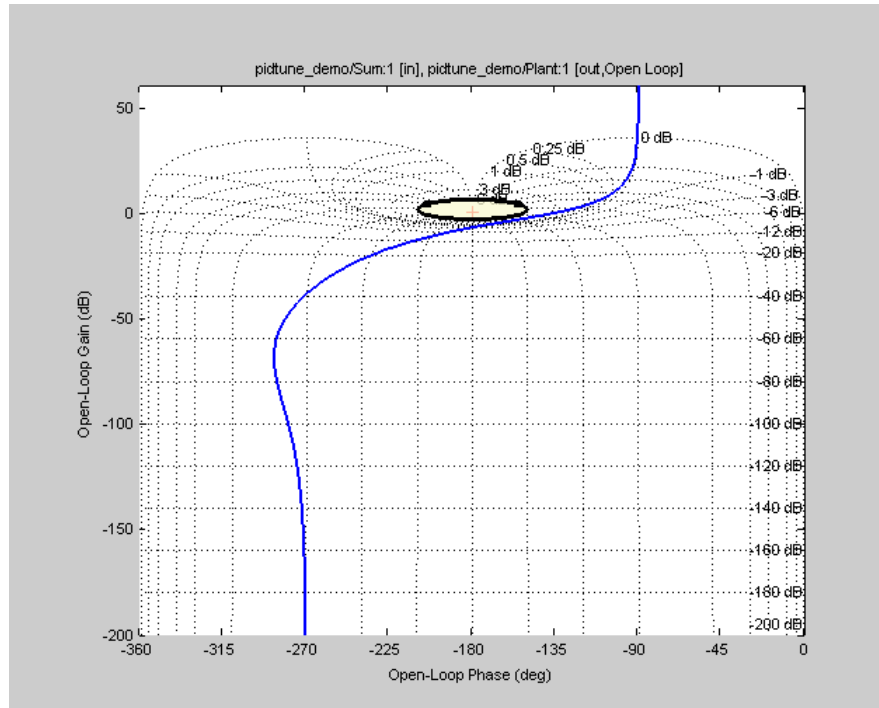
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5** Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window. A graphical display of the requirement also appears in the Design Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Nichols Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

### Specifying Lower Bound on Damping Ratio

To specify a lower bound on the damping ratio of the system:


- 1 In the Design Optimization tool, select **Damping Ratio** in the **New** list. A window opens where you specify an upper bound on the damping ratio of the system.

- 2** Specify a requirement name in the **Name** box.
- 3** Specify the lower bound on the damping ratio in the **Damping ratio** box.
- 4** In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a** Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

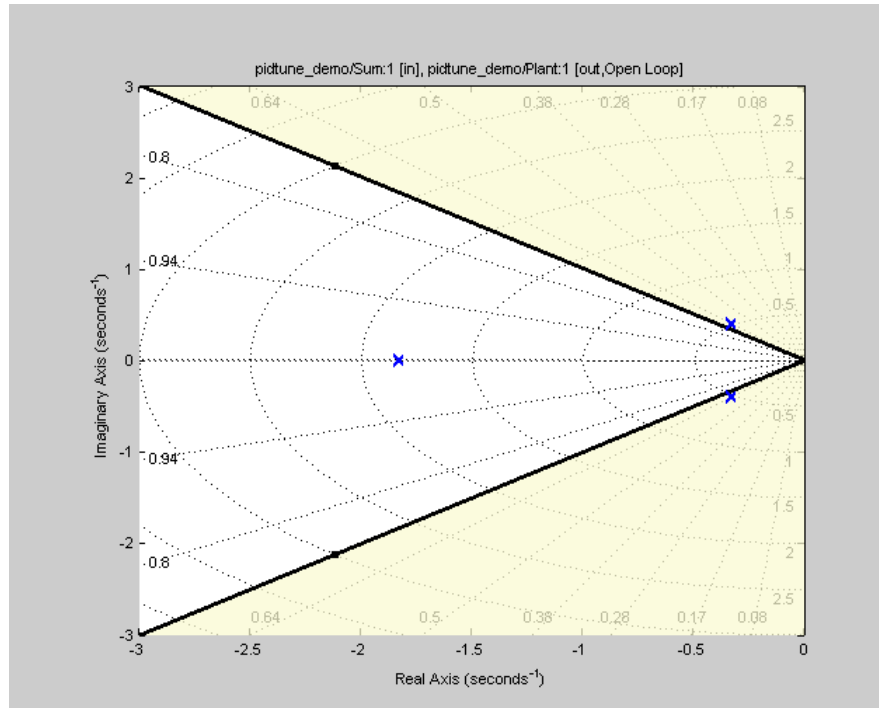
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5** Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the damping ratio. (Requires Simulink Control Design.)

### Specifying Upper and Lower Bounds on Natural Frequency

To specify a bound on the natural frequency of the system:

- 1 In the Design Optimization tool, select **Natural Frequency** in the **New** list. A window opens where you specify a bound on the natural frequency of the system.




- 2 Specify a requirement name in the **Name** box.
- 3 Specify a lower or upper bound on the natural frequency in the **Natural frequency** box.
- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

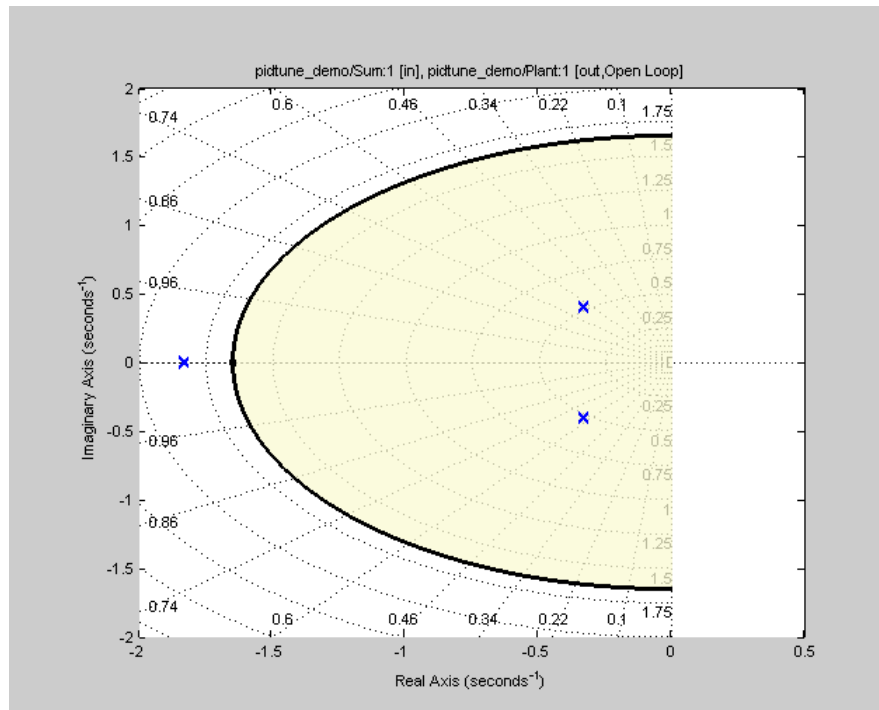
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5 Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the natural frequency. (Requires Simulink Control Design.)

### Specifying Upper Bound on Approximate Settling Time

To specify an upper bound on the approximate settling time of the system:


- 1 In the Design Optimization tool, select **Settling Time** in the **New** list. A window opens where you specify an upper bound on the approximate settling time of the system.

- 2 Specify a requirement name in the **Name** box.
- 3 Specify the upper bound on the approximate settling time in the **Settling time** box.
- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

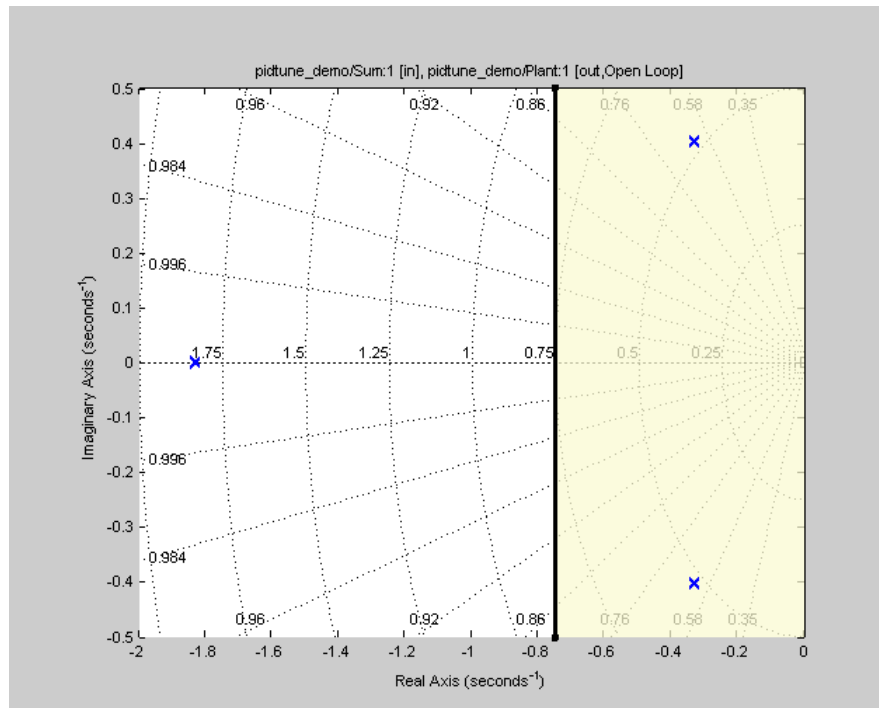
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

- 5 Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.



6 (Optional) In the graphical display, you can:


- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Pole-Zero Characteristics block to specify the approximate settling time. (Requires Simulink Control Design.)

### Specifying Piecewise-Linear Upper and Lower Bounds on Singular Values

To specify piecewise-linear upper and lower bounds on the singular values of a system:

- 1 In the Design Optimization tool, select **Singular Values** in the **New** list. A window opens where you specify the lower or upper bounds on the singular values of the system.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the requirement type using the **Type** list.
- 4 Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns, respectively.
- 5 Insert or delete bound edges.

Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

- 6 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

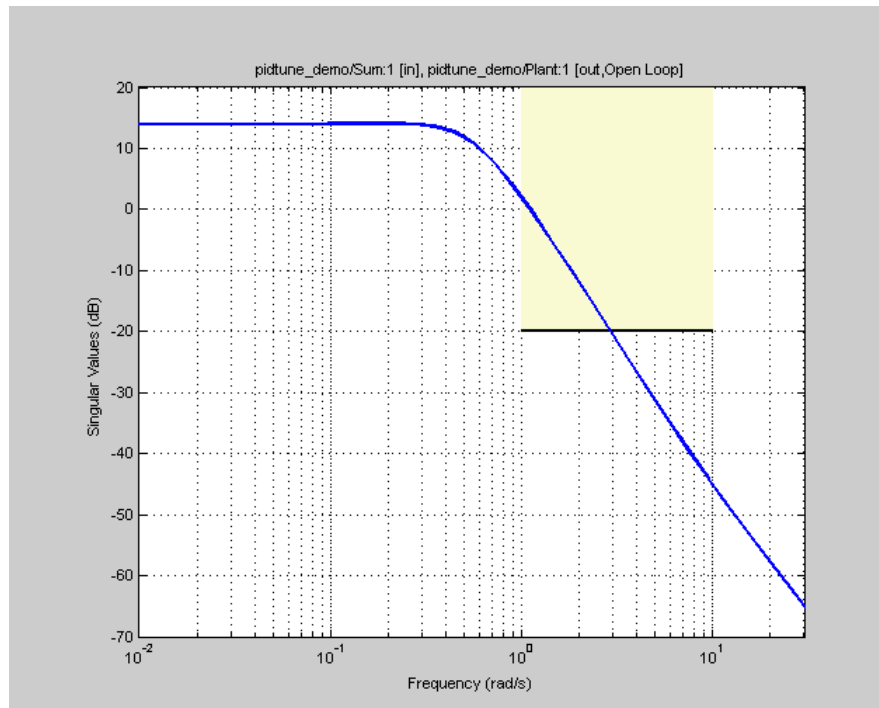
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

### 7 Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.



### 8 (Optional) In the graphical display, you can:

- “Moving Constraints” on page 4-6
- “Split and Join Constraints” on page 4-8

Alternatively, you can use the Check Singular Value Characteristics block to specify bounds on the singular value. (Requires Simulink Control Design.)

## Specifying Step Response Characteristics

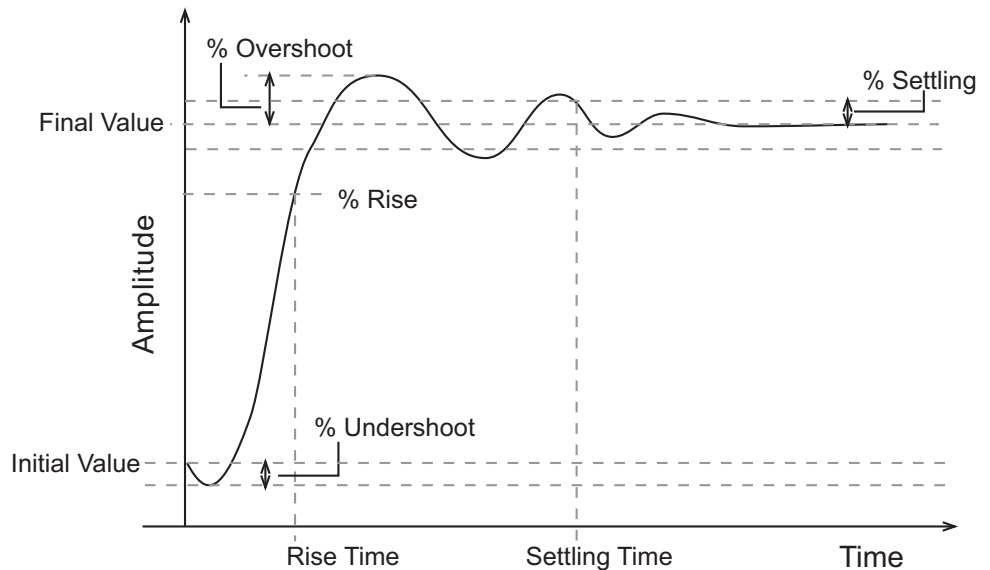
To specify step response characteristics:

- 1 You can apply this requirement to either a signal or a linearization of your model.

In the Design Optimization Tool, click **New**. To apply this requirement to a signal, select the **Step Response Envelope** entry in the **New Time Domain Requirement** section of the **New** list. To apply this requirement to a linearization of your model, select the **Step Response Envelope** entry in the **New Frequency Domain Requirement** section of the **New** list. The latter option requires Simulink Control Design software.

A window opens where you specify the step response requirements on a signal, or system.

- 2 Specify a requirement name in the **Name** box.
- 3 Specify the step response characteristics:



- **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
- **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.

#### 4 Specify the signals or systems to be bound.


You can apply this requirement to a model signal or to a linearization of your Simulink model (requires Simulink Control Design software).

- Apply this requirement to a model signal:

In the **Select Signals to Bound** area, select a logged signal to which you will apply the requirement.

If you have already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, it appears in the list. Select the corresponding check-box.

If you haven't selected a signal to log:


- Click . A window opens where you specify the logged signal.



- b In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.

Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.


- Apply this requirement to a linear system.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.

- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

### 5 Click **OK**.

A variable with the specified requirement name appears in the **Design Optimization Workspace**. A graphical display of the requirement also appears in the Design Optimization tool window.

Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.

### **See Also**


“Optimize Controller Parameters to Meet Step Response Requirements (GUI)”

## **Specifying Custom Requirements**

To specify custom requirements such as minimizing system energy:

- 1** In the Design Optimization tool, select **Custom Requirement** in the **New** drop-down list. A window opens where you specify the custom requirement.
- 2** Specify a requirement name in the **Name** box.
- 3** Specify the requirement type using the **Type** list.
- 4** Specify the name of the function that contains the custom requirement in the **Function** box. The field must be specified as a function handle using

@. The function must be on the MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is `myCustomRequirement`.

**5** (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design software), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

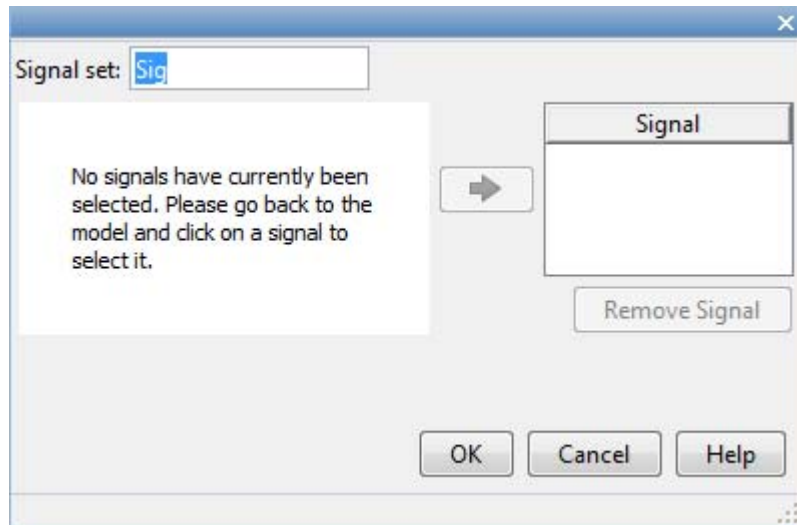
- Apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log (GUI)” on page 3-14, it appears in the list. Select the corresponding check-box.

If you haven’t selected a signal to log:

- a Click . A window opens where you specify the logged signal.
- b In the Simulink model window, click the signal where you want to add a requirement.



The window updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** box, enter a name for the selected signal set.

Click **OK**. A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool window.

- Apply this requirement to a linear system.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it will appear in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click



to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Creating Linearization I/O Sets (GUI)” on page 3-69.

For more information on linearization, see “What Is Linearization?”.

#### **6** Click **OK**.

A new variable with the specified name appears in the **Design Optimization Workspace** of the Design Optimization tool. A graphical display of the requirement also appears in the Design Optimization tool window.

#### **See Also**

- “Design Optimization to Meet a Custom Objective Using the GUI” on page 3-103
- “Design Optimization to Meet Custom Signal Requirements (GUI)” on page 3-133

# Time- and Frequency-Domain Requirements in SISO Design Tool

In this section...
“Root Locus Diagrams” on page 4-42
“Open-Loop and Prefilter Bode Diagrams” on page 4-44
“Open-Loop Nichols Plots” on page 4-45
“Step/Impulse Response Plots” on page 4-46

This topic lists the time- and frequency-domain requirements that you can specify for optimization-based control design using the SISO Design Tool.

### Examples and How To

- “Designing Optimization-Based Controllers for LTI Systems” on page 4-48
- “Design an Optimization-Based PID Controller for a Linearized Simulink Model”

### Root Locus Diagrams

- “Settling Time” on page 4-42
- “Percent Overshoot” on page 4-43
- “Damping Ratio” on page 4-43
- “Natural Frequency” on page 4-44
- “Region Constraint” on page 4-44

### 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  $\text{Re}\{pole\} < -4.6/T_{settle}$  for continuous systems and  $\log(\text{abs}(pole))/T_{discrete} < -4.6/T_{settle}$  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  $\zeta$  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  $abs(pole)$  for continuous systems and on

$$\begin{aligned}r &= abs(pSys) \\t &= angle(pSys) \\c &= \sqrt{(\log(r))^2 + t^2} / Ts_{model}\end{aligned}$$

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” on page 4-44
- “Upper Gain Limit” on page 4-44
- “Lower Gain Limit” on page 4-44

### 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” on page 4-45
- “Gain Margin” on page 4-45
- “Closed-Loop Peak Gain” on page 4-45
- “Gain-Phase Requirement” on page 4-45

### 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” on page 4-46
- “Lower Time Response Bound” on page 4-46

### **Upper Time Response Bound**

You can specify an upper time response bound.

### **Lower Time Response Bound**

You can specify a lower time response bound.

## Time-Domain Simulations in SISO Design Tool

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.

# Designing Optimization-Based Controllers for LTI Systems

In this section...
“How to Design Optimization-Based Controllers for LTI Systems” on page 4-48
“Example — Frequency-Domain Optimization for LTI System” on page 4-49

## 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 4-50.
- 2 Create a SISO Design Task with design and analysis plots, as described in “Creating Design and Analysis Plots” on page 4-51.

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 4-54 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 4-56 for more information.

---

**Note** Compensator elements or parameters cannot have uncertainty when used with frequency-domain based response optimization.

---

- 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 4-57 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 4-65 for more information.

## **Example — Frequency-Domain Optimization for LTI System**

- “Introduction” on page 4-49
- “Design Requirements” on page 4-50
- “Creating an LTI Plant Model” on page 4-50
- “Creating Design and Analysis Plots” on page 4-51
- “Creating a Response Optimization Task” on page 4-54
- “Selecting Tunable Compensator Elements” on page 4-56
- “Adding Design Requirements” on page 4-57
- “Optimizing the System’s Response” on page 4-65
- “Creating and Displaying the Closed-Loop System” on page 4-68

### **Introduction**

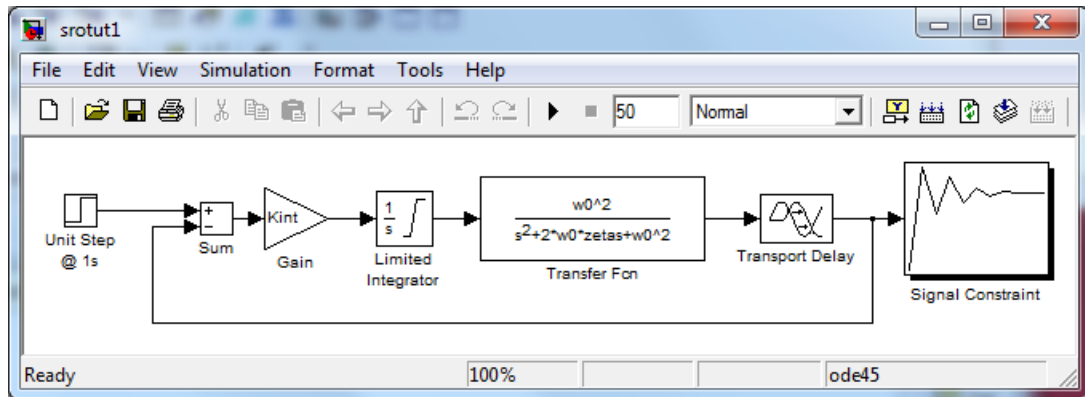
When you have Control System Toolbox software, you can place Simulink Design Optimization design requirements or constraints on plots in the SISO Design Tool graphical tuning editor and analysis plots that are part of a SISO Design Task. This allows you to include design requirements for response optimization in the frequency-domain in addition to the time-domain. This topic 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  $\pm 0.7$  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;
```

If the plant model is an array of LTI models, the controller is designed for a nominal model only but you can analyze the control design for the remaining models in the array. For more information, see “Control Design Analysis of Multiple Models” in the Control System Toolbox documentation.

---

**Tip** You can directly linearize the Simulink model using Simulink Control Design software.

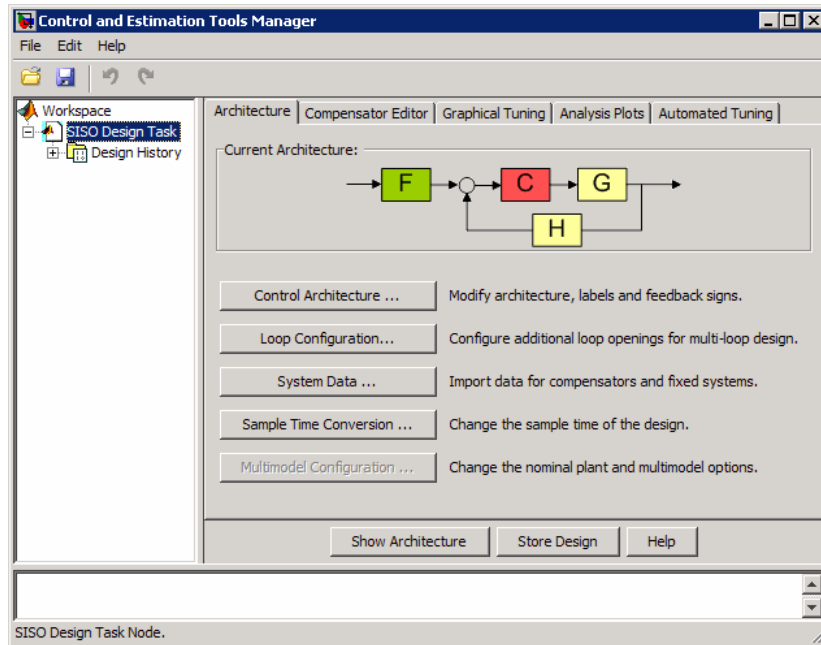
---

## 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.

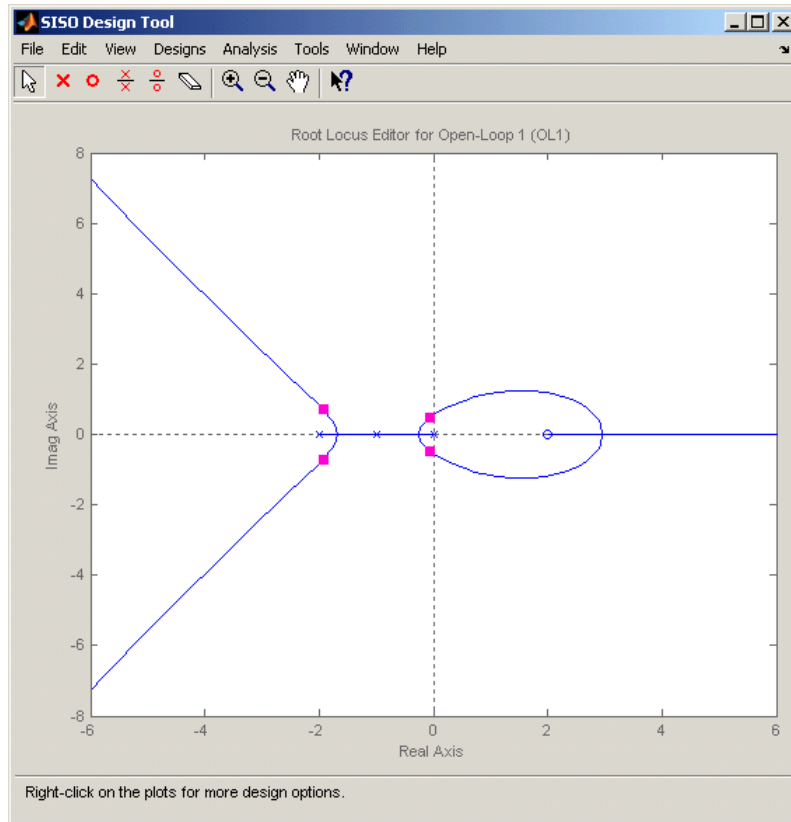


The Control and Estimation Tools Manager is a graphical environment for managing and performing tasks such as designing SISO systems. The SISO Design Task node contains five panels that perform actions related to designing SISO control systems. For more information, see “Using the SISO Design Task in 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.

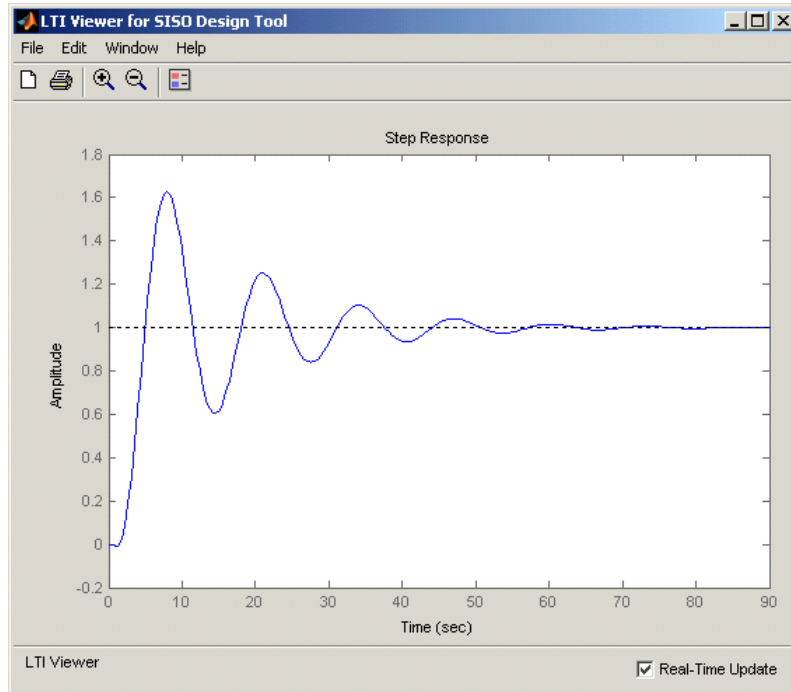




In addition to the root-locus diagram, it is helpful to visualize the response of the system with a step response plot. To add a step response:

- 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

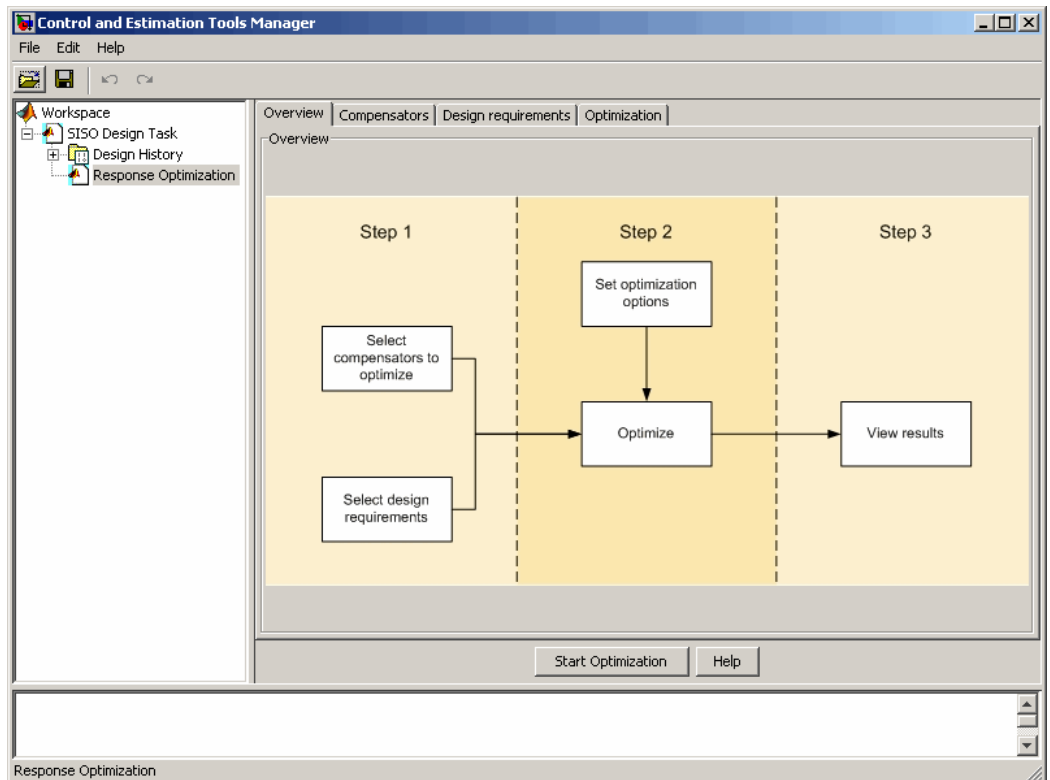
There are several possible methods for designing a SISO system; this example uses an automated approach involving response optimization methods. After creating the design and analysis plots as discussed in “Creating Design and Analysis Plots” on page 4-51, you are ready to start a response optimization 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 4-56.
- **Design requirements:** Select the design requirements that you want the system to meet after tuning the compensator elements. See “Adding Design Requirements” on page 4-57.
- **Optimization:** Configure optimization options and view the progress of the response optimization. See “Optimizing the System’s Response” on page 4-65.

---

**Note** When optimizing responses in a SISO Design Task, you cannot add uncertainty to parameters or compensator elements.

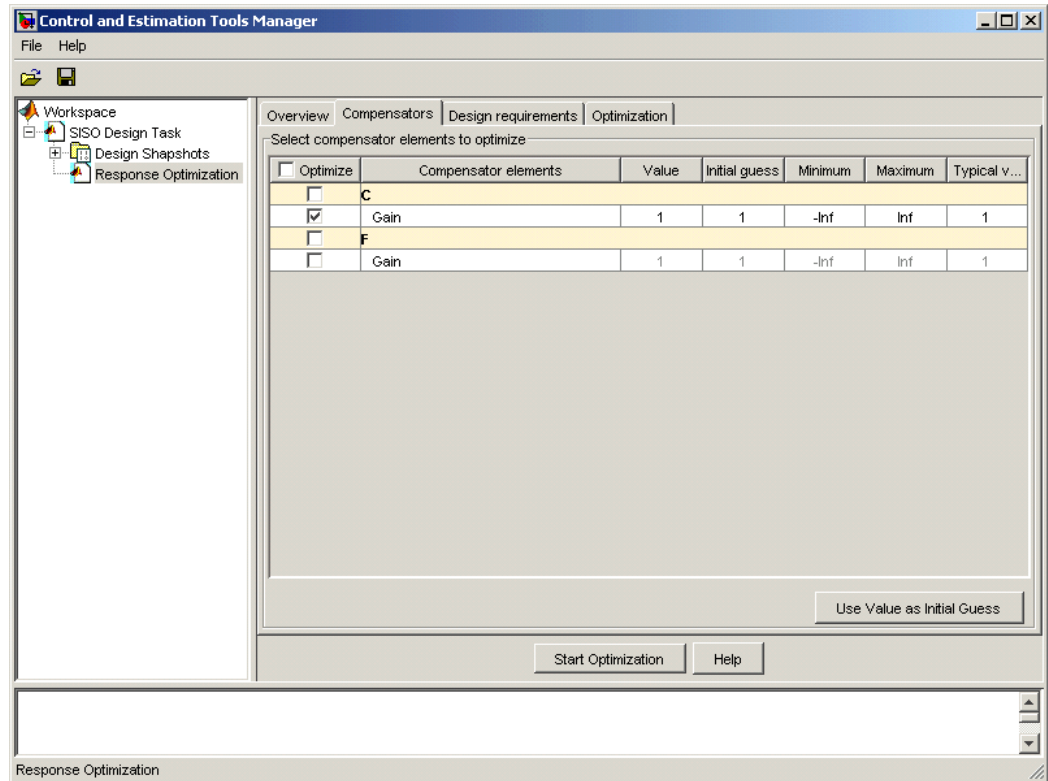
---

### 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 4-50. The following sections each create a new design requirement to meet these specifications:

- “Settling Time Design Requirement” on page 4-58
- “Overshoot Design Requirement” on page 4-59
- “Rise Time Design Requirement” on page 4-61
- “Actuator Limit Design Requirement” on page 4-62

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 4-65.

**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.

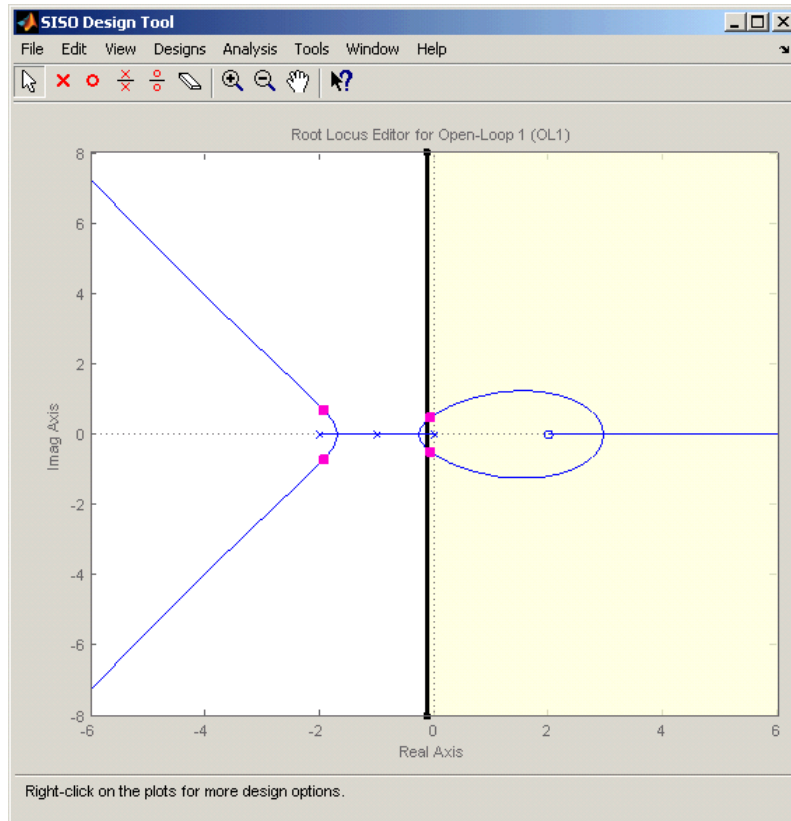
To add this design requirement:

- 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**.

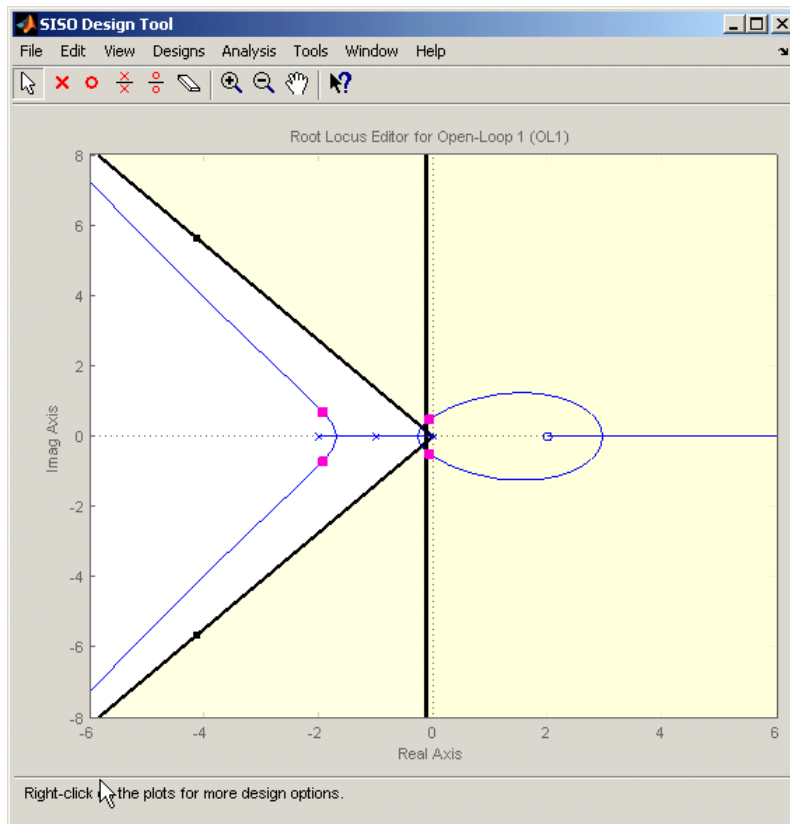
A vertical line should appear on the root-locus diagram, as shown in the following figure.



**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.



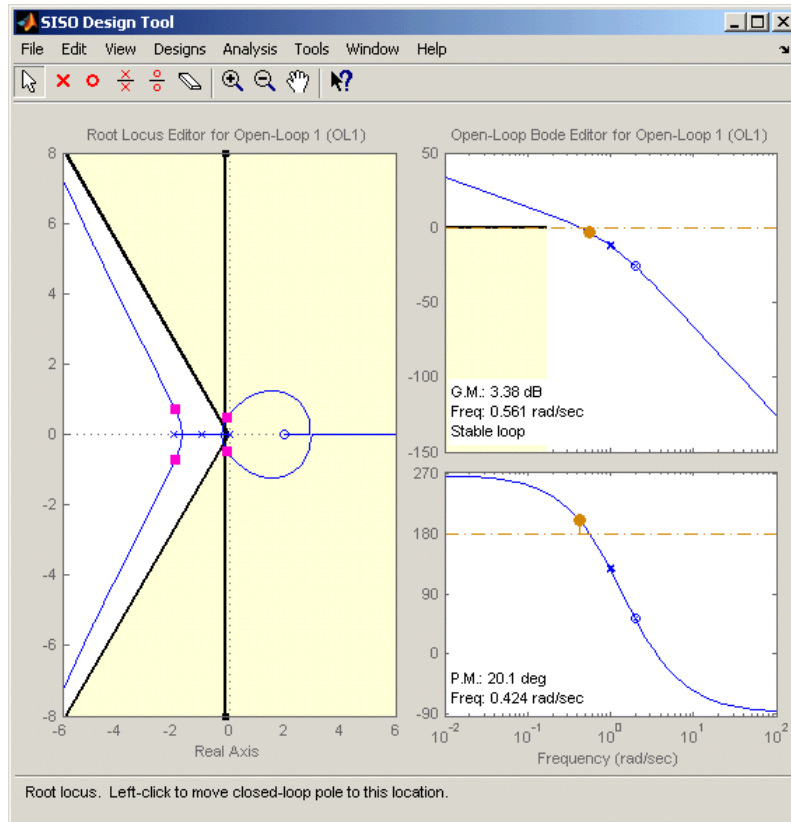


**Rise Time Design Requirement.** The third design specification for this example is to have a rise time of 10 seconds or less. This specification is related to a lower limit on a Bode Magnitude diagram.

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**.

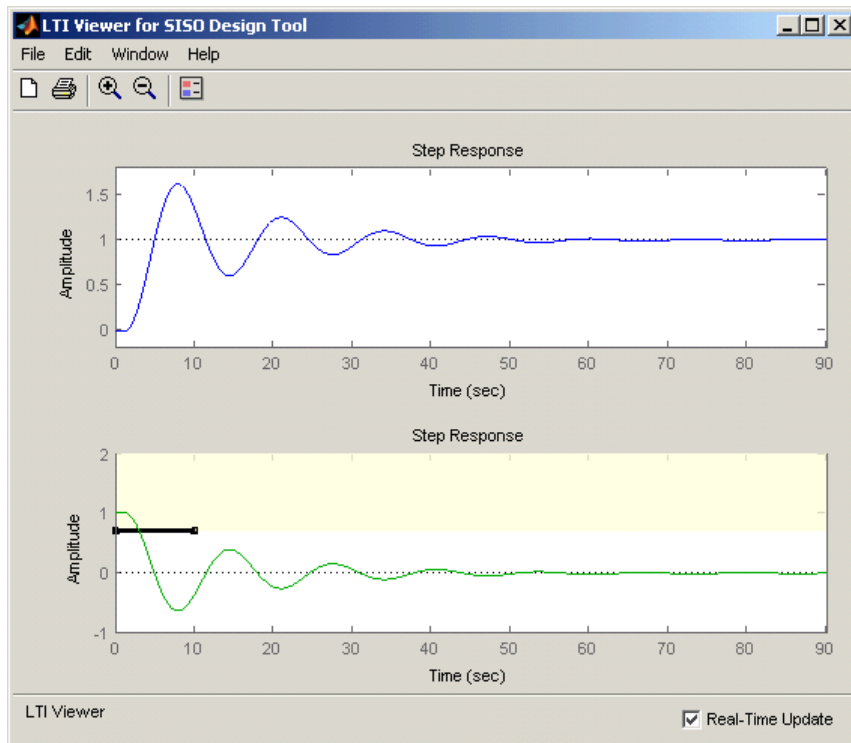
A Bode diagram appears within the SISO Design Tool window. The magnitude plot of the Bode diagram includes a horizontal line representing the design requirement, as shown in the following figure.



**Actuator Limit Design Requirement.** The fourth design specification for this example is to limit the actuator signal to within  $\pm 0.7$ . To add this design requirement:

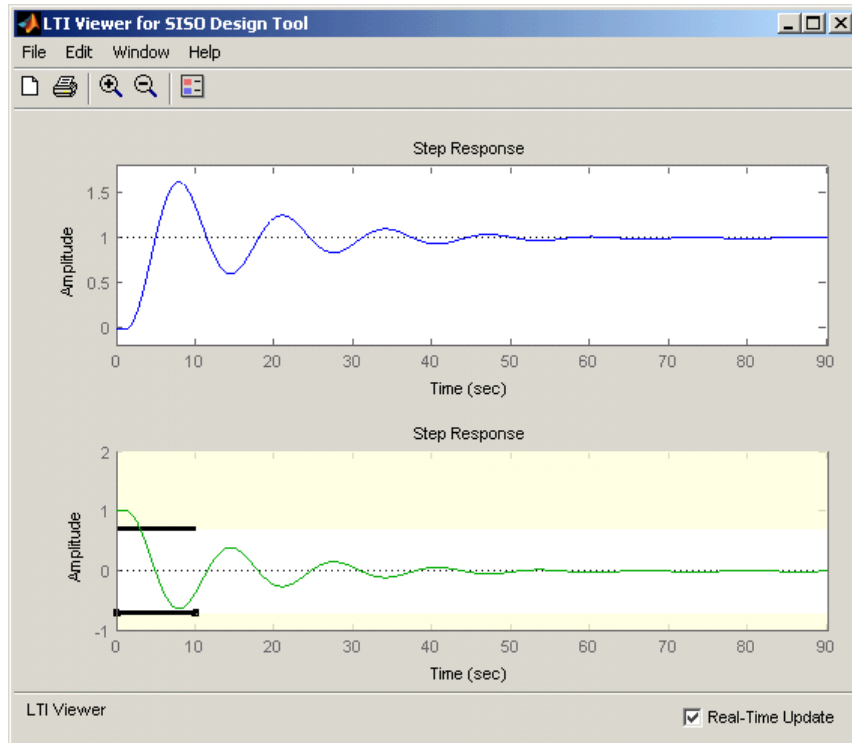
- 1 Select the **Design requirements** pane in the **Response Optimization** node of the Control and Estimation Tools Manager.
- 2 Click the **Add new design requirement** button to open the New Design Requirement dialog box.
- 3 Create a time-domain design requirement to represent the upper limit on 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

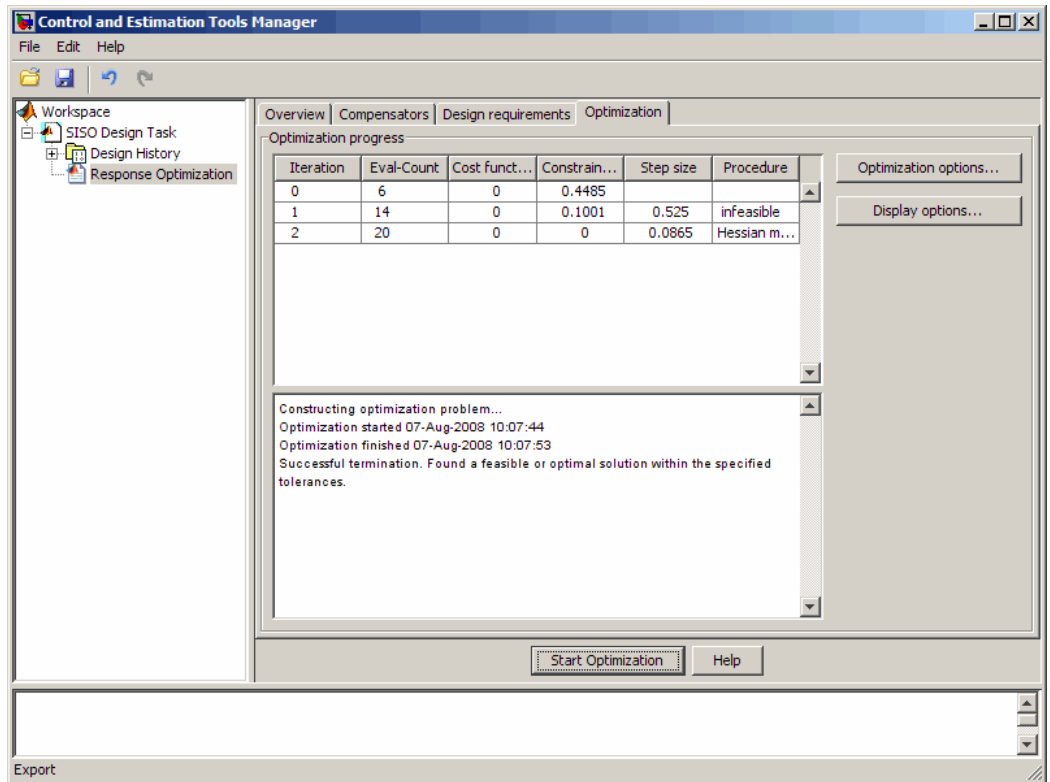
After selecting the compensator elements to tune and adding design requirements for the response signals to satisfy, you are ready to begin the response optimization.

The **Optimization** pane within the **Response Optimization** node of the Control and Estimation Tools Manager displays the progress of the response

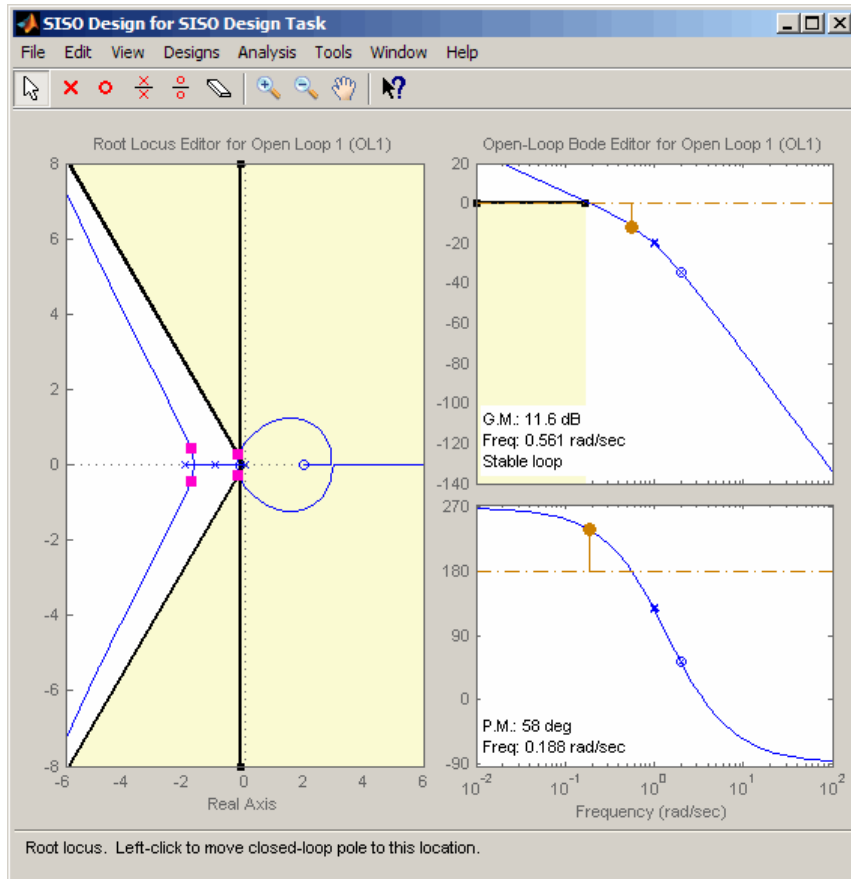
optimization. The pane also contains options to configure the types of progress information displayed during the optimization and options to configure the optimization methods and algorithms.

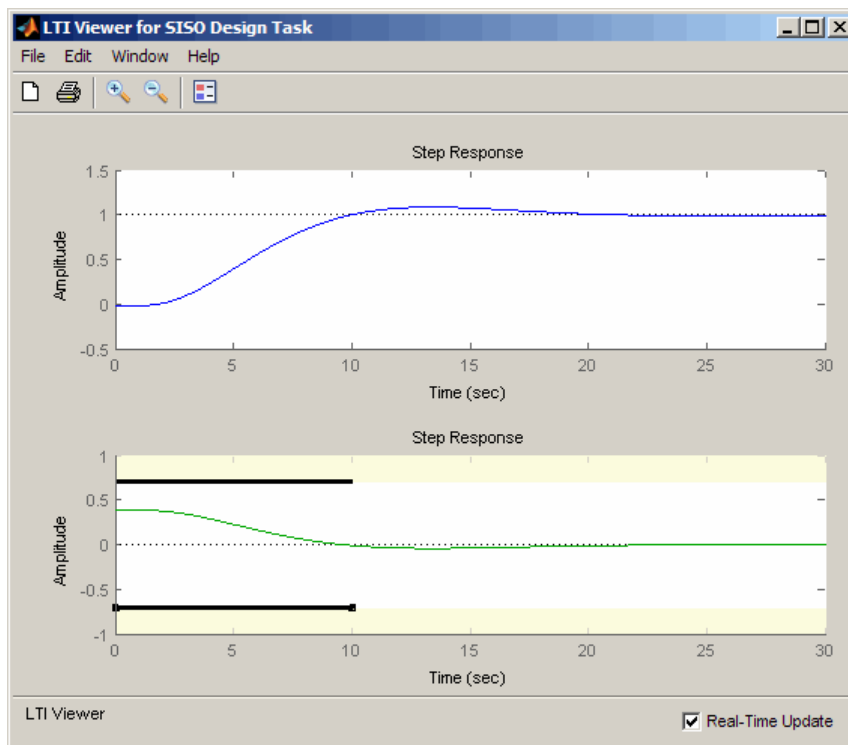
To optimize the response of the system in this example, click the **Start Optimization** button.

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:

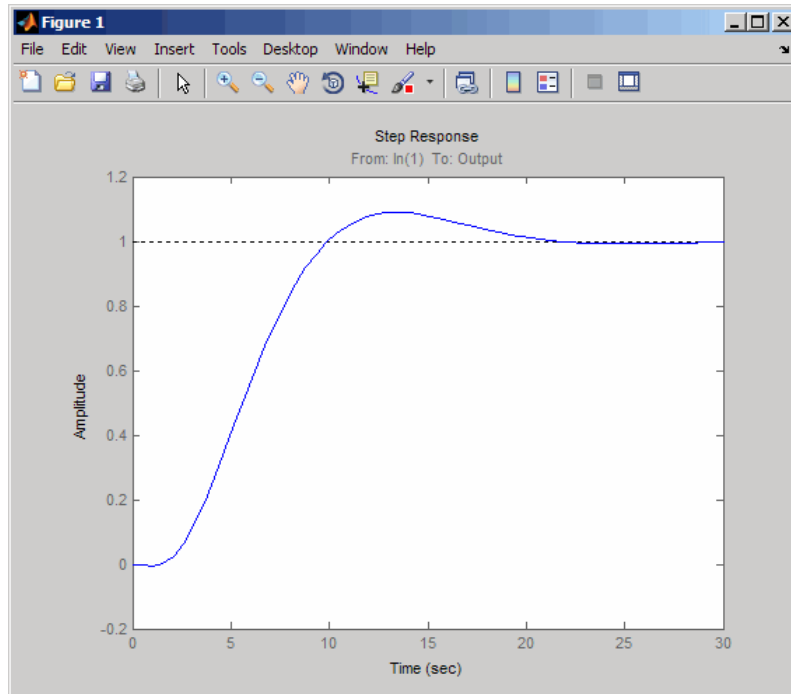
```
Zero/pole/gain from input to output "Output":  
      -0.19414 (s-2)
```

```
-----  
(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:



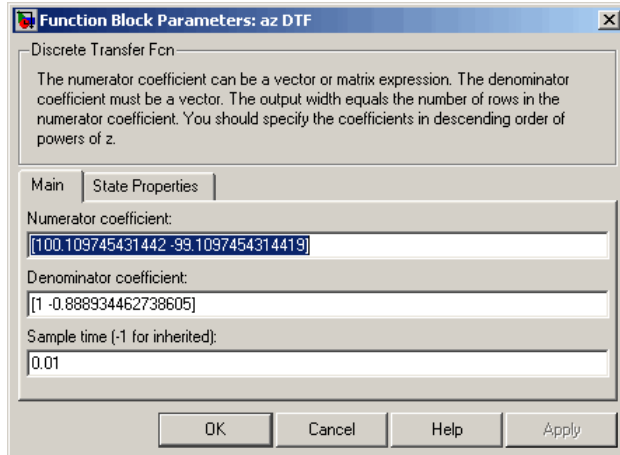
# Designing Linear Controllers for Simulink Models

When you have Control System Toolbox and Simulink Control Design software, you can perform frequency-domain optimization of Simulink models.

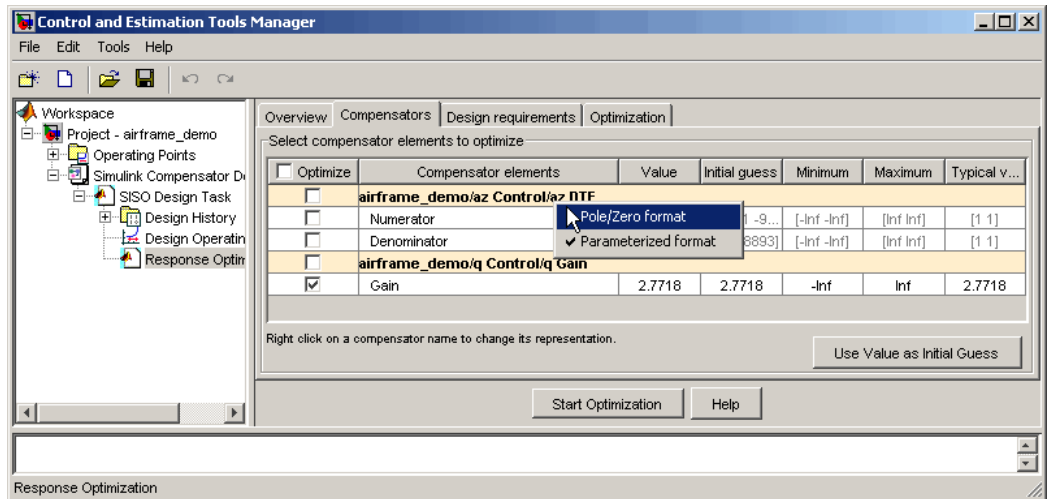
You can use Simulink Control Design software to configure SISO Design Tool with compensators, inputs, outputs, and loops computed from a Simulink model. For more information, see “Creating a SISO Design Task” in Simulink Control Design documentation.

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 “Design an Optimization-Based PID Controller for a Linearized Simulink Model” 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.

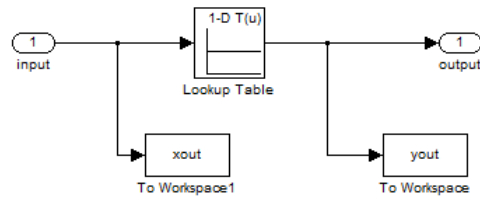
### **Configuring a Project for Parameter Estimation**

To estimate the lookup table values, you must first configure a Control and Estimation Tools Manager project.

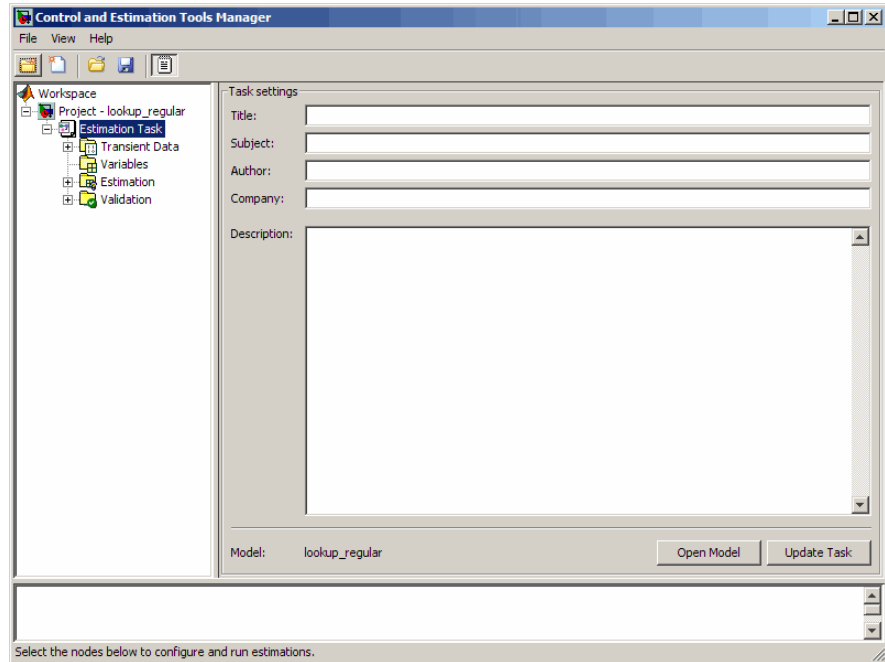
- 1 Open the lookup table model by typing the following command at the MATLAB prompt:

```
lookup_regular
```

This command opens the Simulink model, and loads the estimation data into the MATLAB workspace.



- 2 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

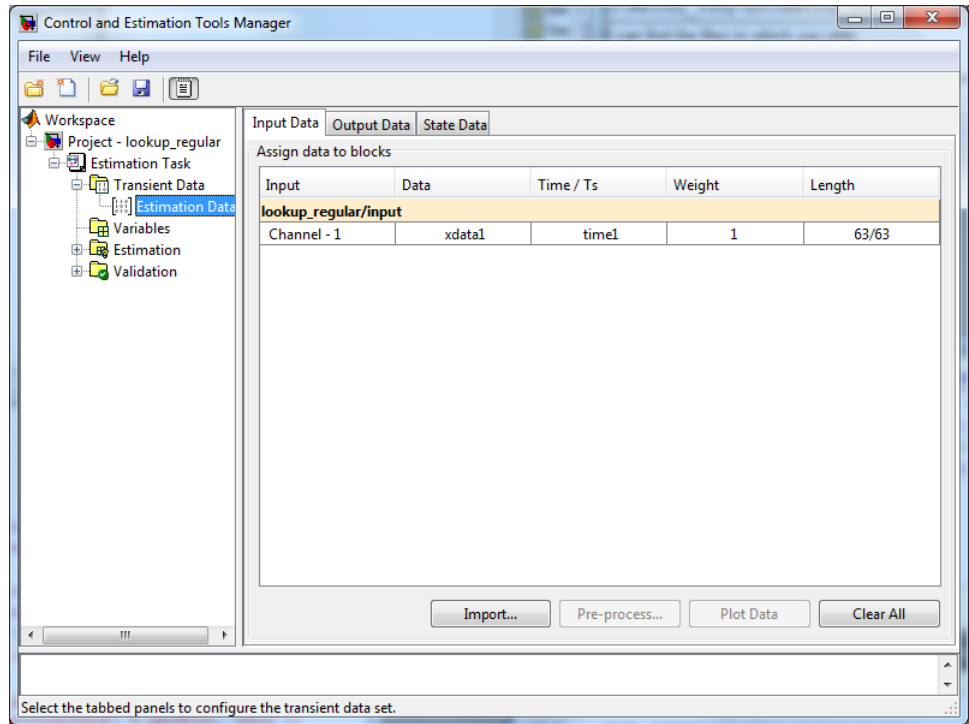
After you configure a project for parameter estimation, as described in “Configuring a Project for Parameter Estimation” on page 5-6, use the following steps to estimate the lookup table values.

- 1 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 “Import Data (GUI)” on page 1-4.

You can also load a preconfigured project that already contains the imported data. To do so, type the following command at the MATLAB prompt:

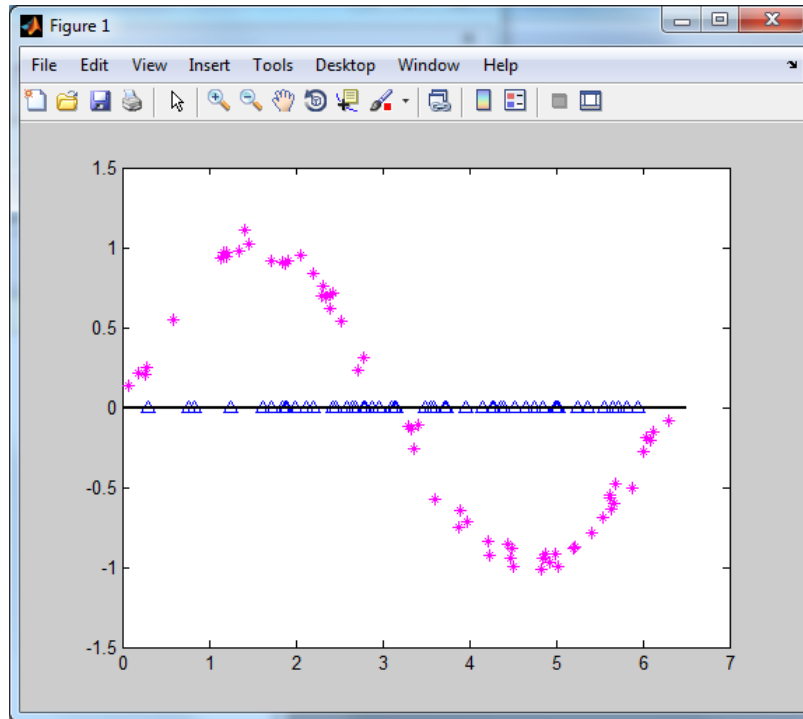
```
lookup_regular;  
explorer.loadProject('lookup_regular_import', ...
```

```
'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)
```

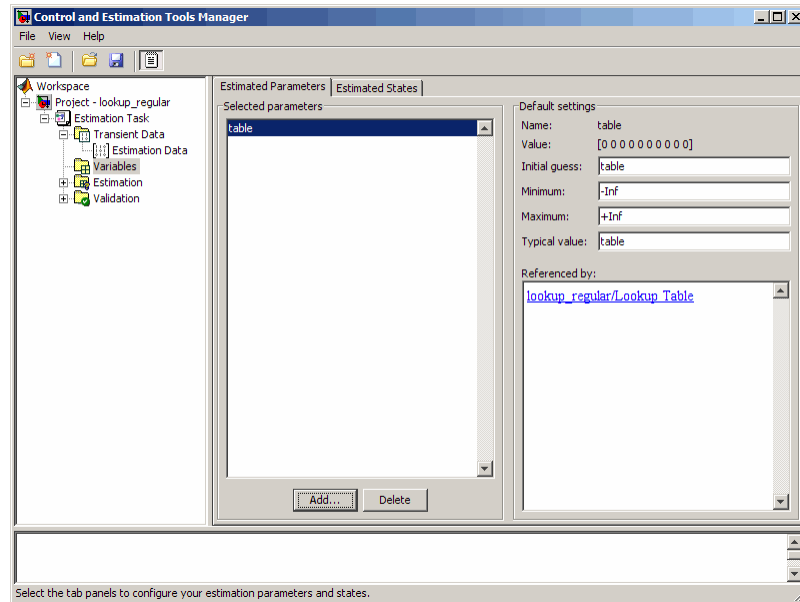


The x- and y-axes of the figure represent the input and output data, respectively. The figure shows the following plots:

- Measured data — Represented by the magenta stars (\*).
- Initial table values — Represented by the black line.
- Initial simulation data — Represented by the blue deltas ( $\Delta$ ).

- 3 Select the table values to estimate.

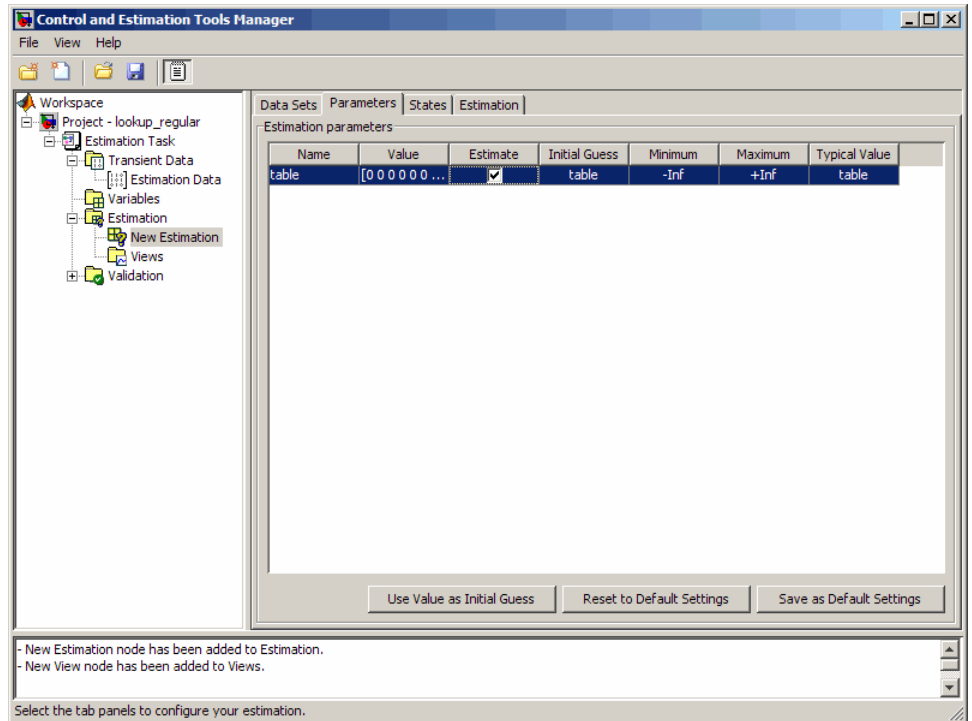
- a In the Control and Estimation Tools Manager GUI, select the **Variables** node under the **Estimation Task** node.
- b Click **Add** to open the Select Parameters dialog box, which shows the Simulink model parameters.
- c Select **table**, and click **OK** to add the table values to the **Estimated 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.

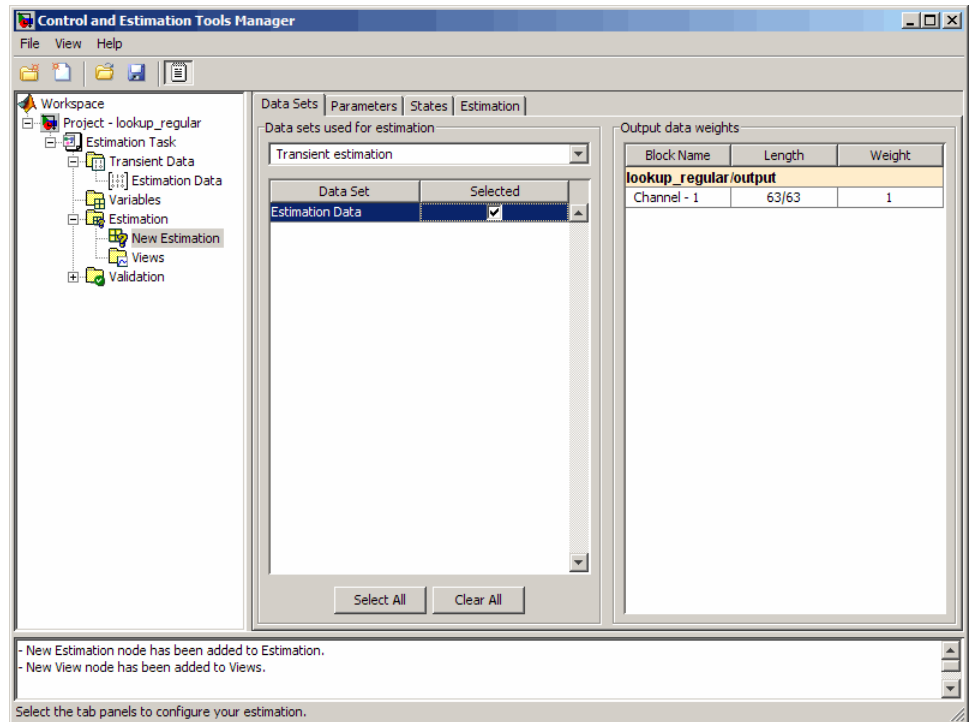
- d Select the **Estimation** node, and click **New** to add a **New Estimation** node.

- e Select the **New Estimation** node. In the **Parameters** tab, select the **Estimate** check box to specify the lookup table values, table, for estimation.





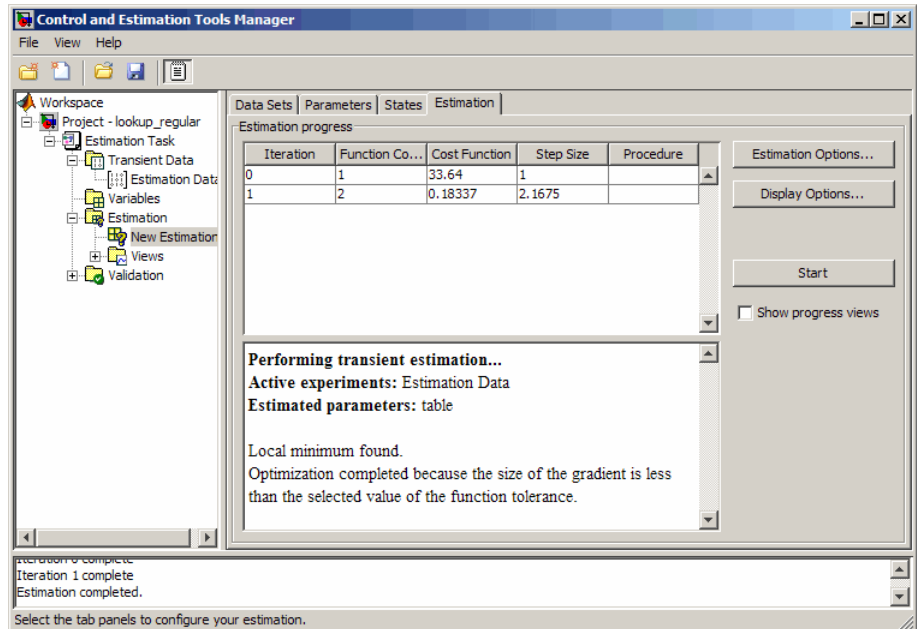
- 4 In the **Data Sets** tab of the **New Estimation** node, select the **Selected** check box to specify the estimation data set.



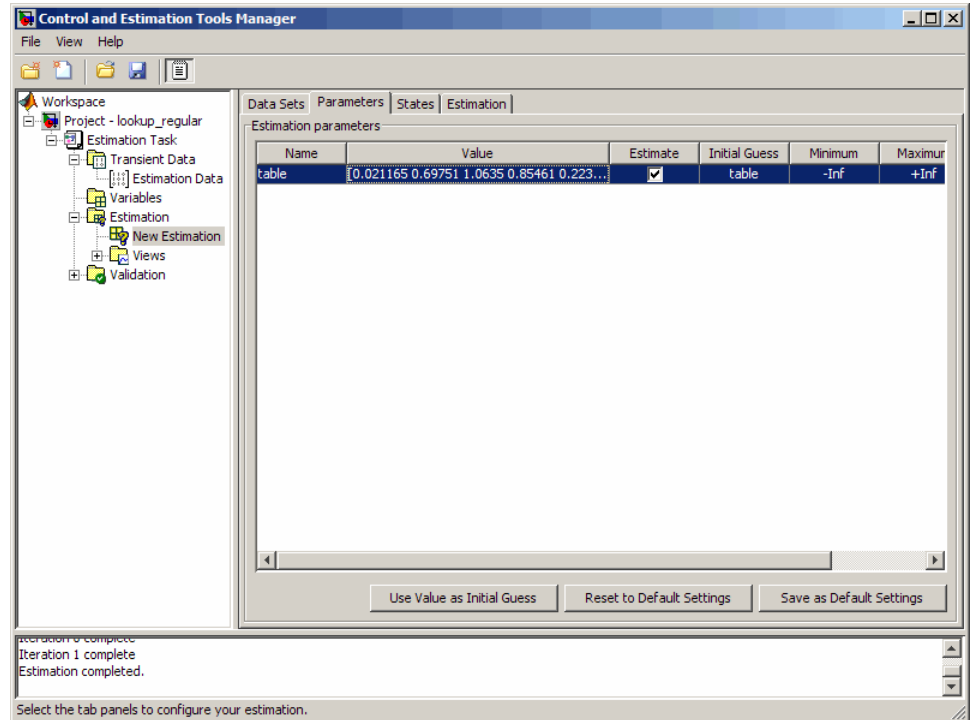
- 5 Estimate the table values using the default settings.

- 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:

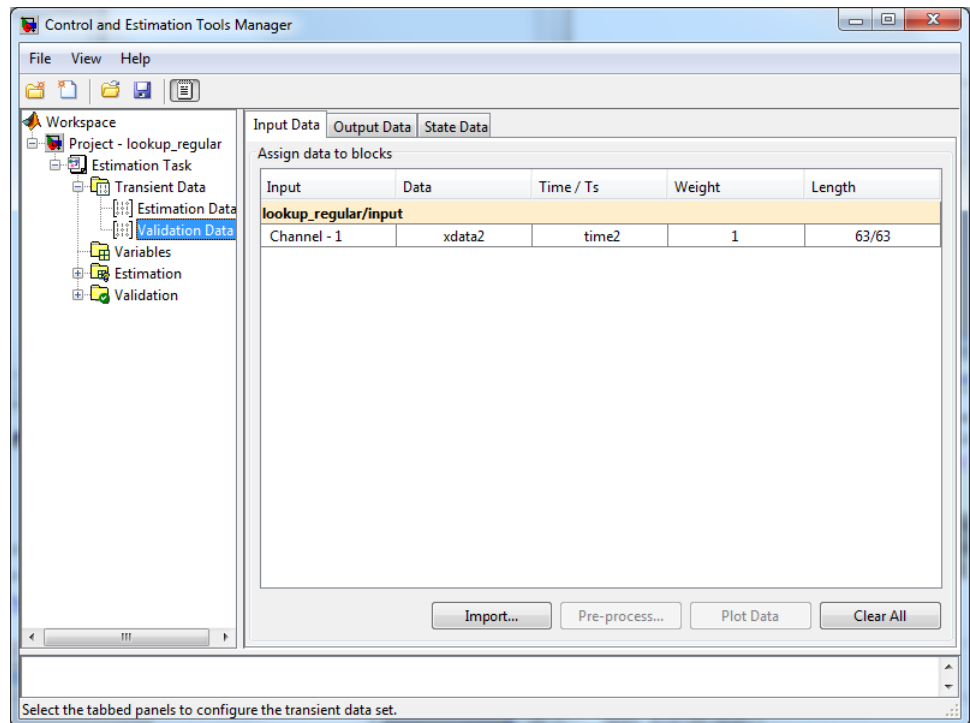
- Residuals plot
- Measured and simulated data plots

To validate the estimation results:

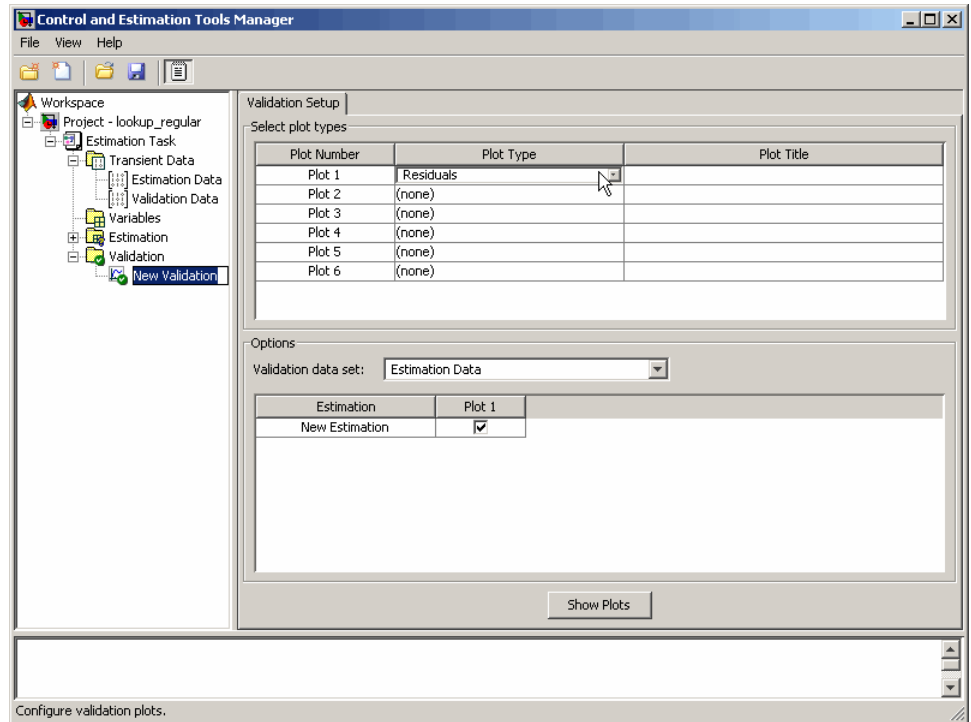
- 1 Import the validation I/O data, `xdata2` and `ydata2`, and time vector, `time2`, in the Control and Estimation Tools Manager GUI.

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');
```



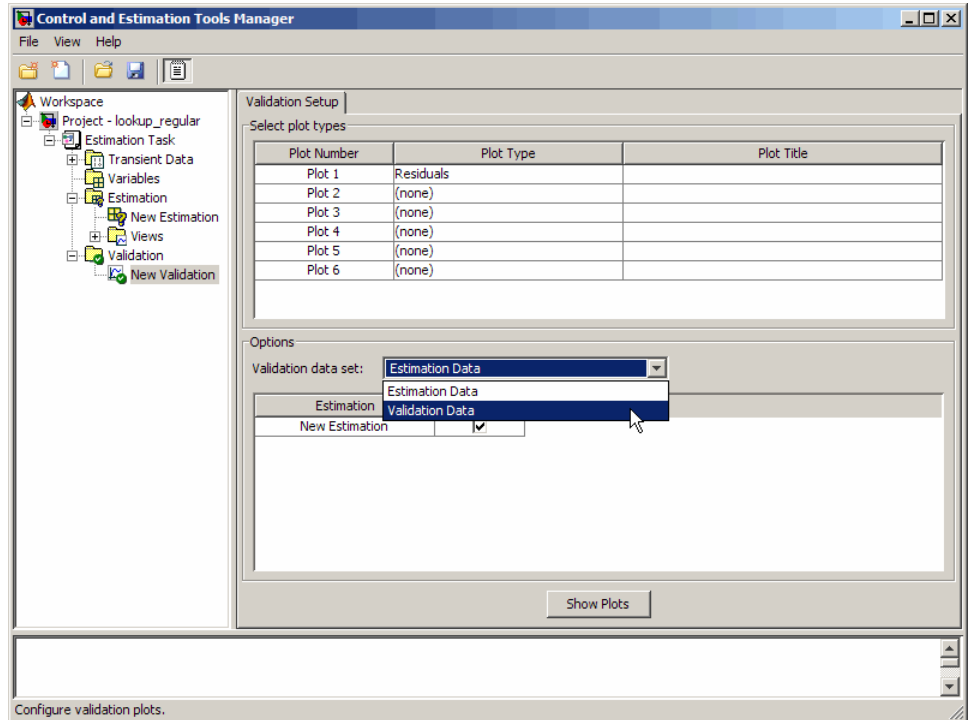
This project also contains the **Residuals** plot already configured in the **Select plot types** area of the GUI, as shown in the next figure. For more information on how to configure this plot, see “Compare Measured and Simulated Responses (GUI)” on page 2-46.



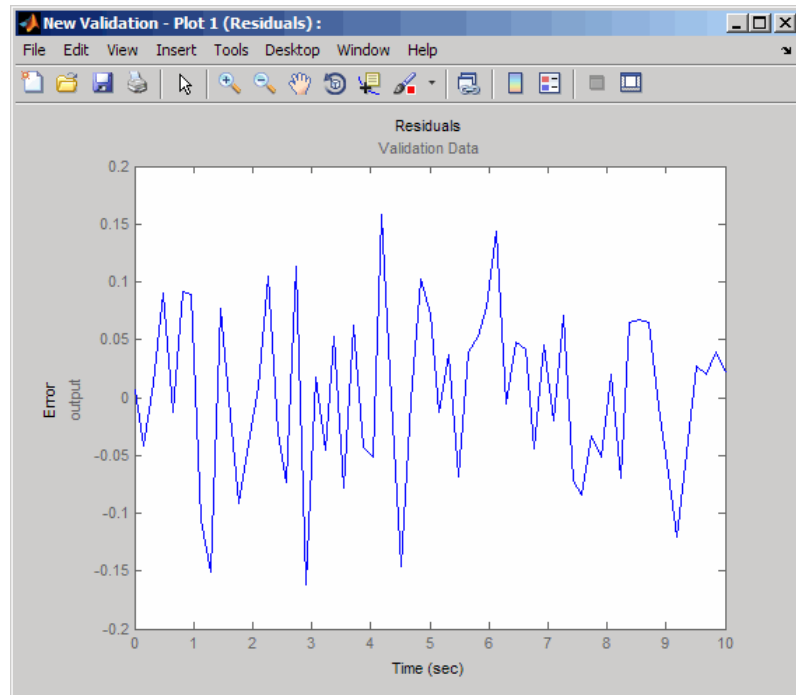
## 2 Plot and examine the residuals:

- 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.



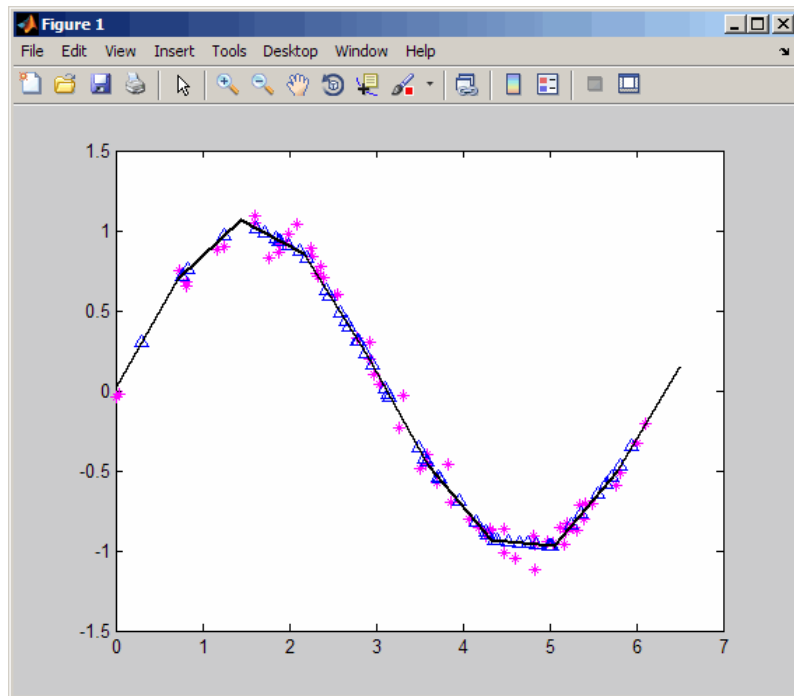
- c Click **Show Plots** to open the residuals plot.



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.

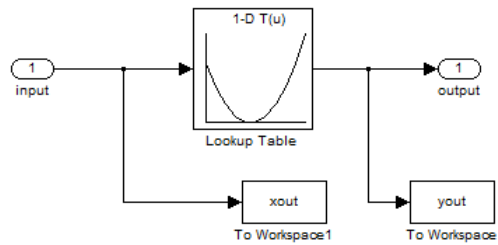
## Configuring a Project for Parameter Estimation

To estimate the monotonically increasing lookup table values, you must first configure a Control and Estimation Tools Manager project.

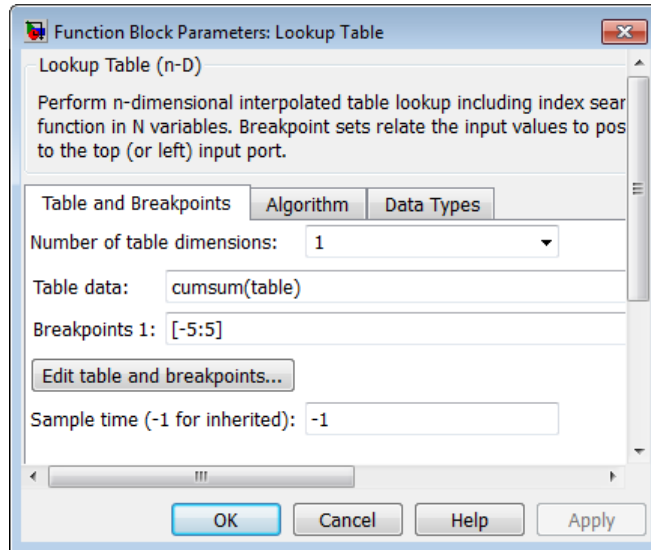
- 1 Open the lookup table model by typing the following command at the MATLAB prompt:

```
lookup_increasing
```

This command opens the Simulink model, and loads the estimation data in the MATLAB workspace.

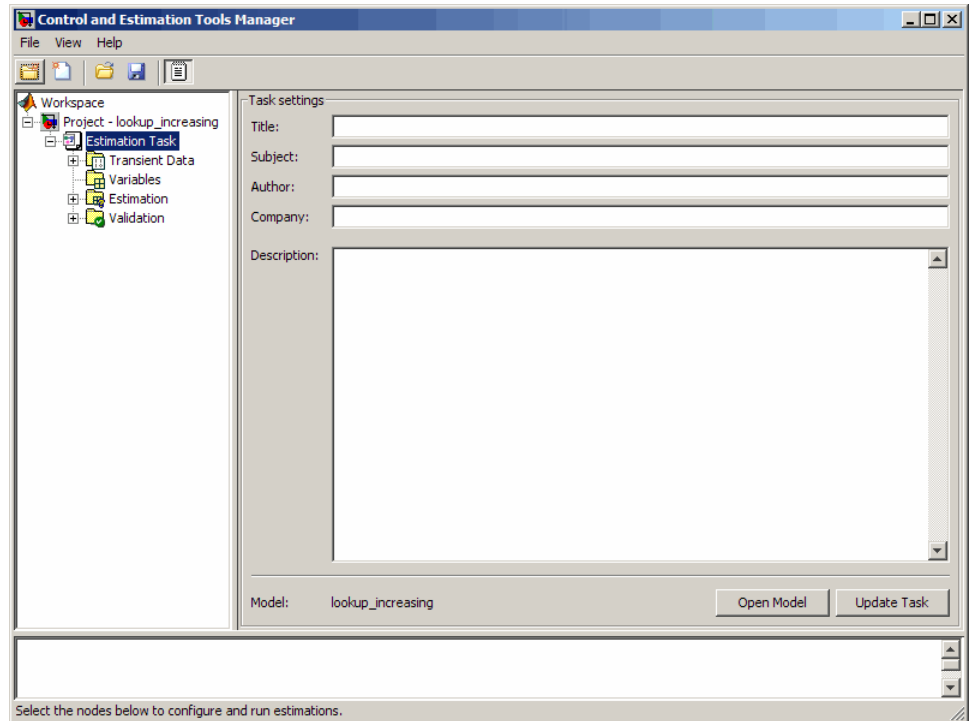


- 2 Double-click the Lookup Table block to view the monotonically increasing constraint applied to the table output values.



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.



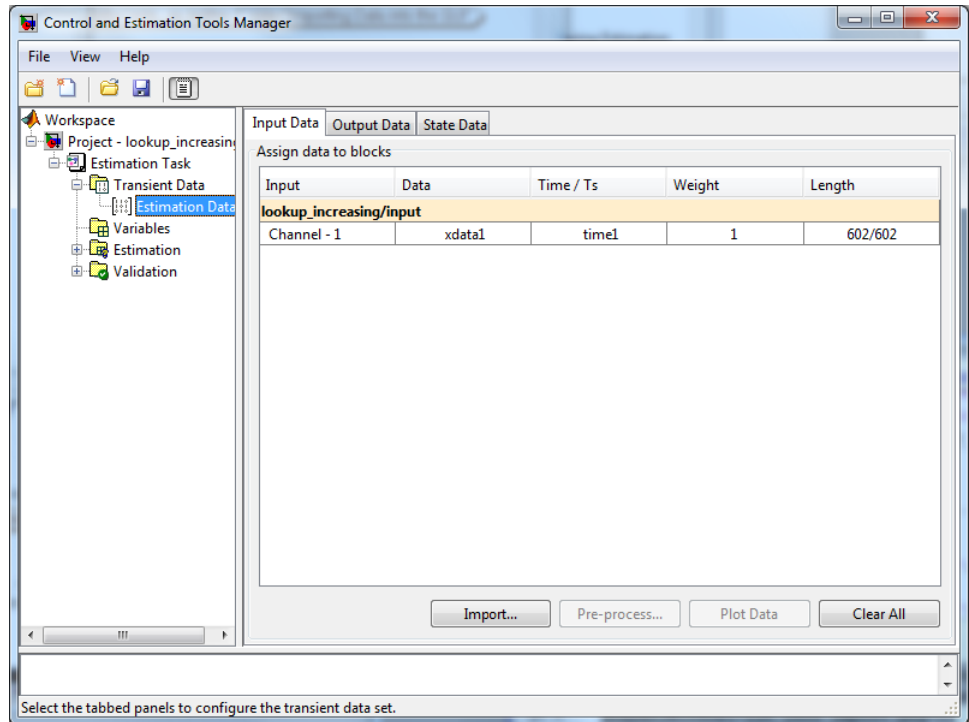
### Estimating the Monotonically Increasing Table Values Using Default Settings

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 I/O data, as described in the “Importing Data into the GUI” section of “Prepare Data for Parameter Estimation Using the GUI”.

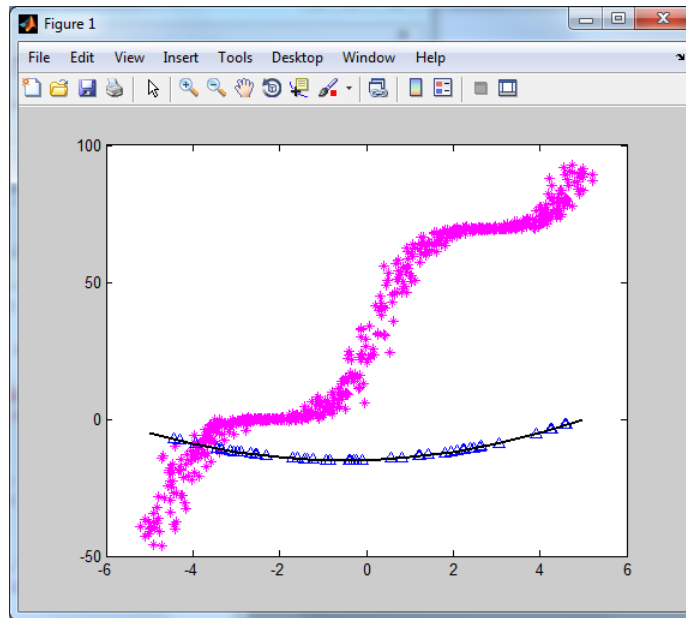
You can also load a preconfigured project that already contains the imported data. To do so, type the following commands at the MATLAB prompt:

```
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 (\*).

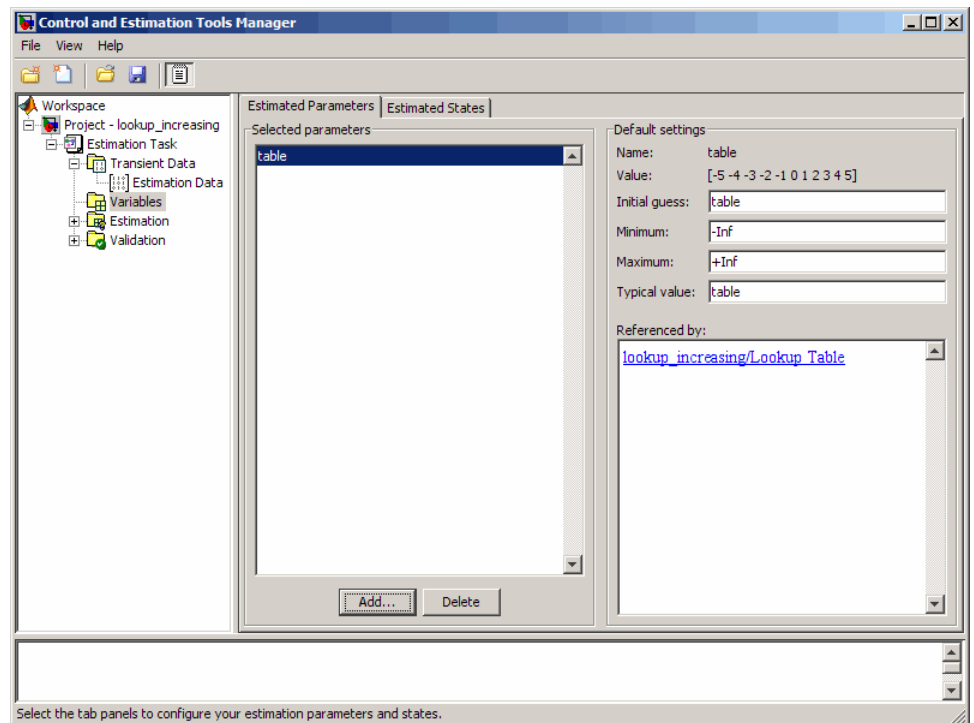
---

**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 ( $\Delta$ ).

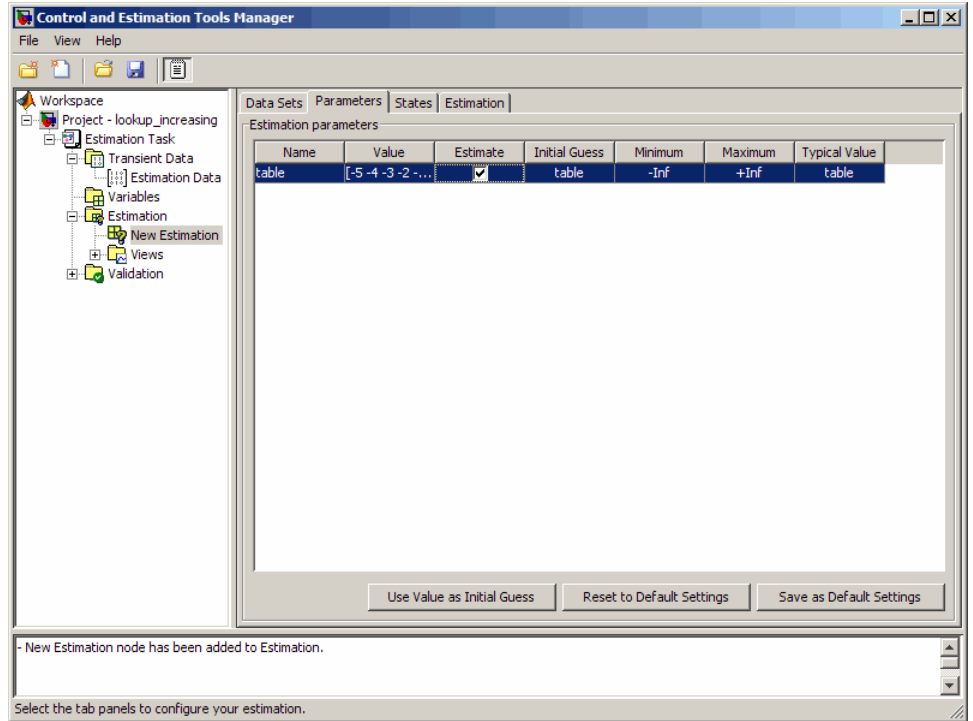
- 3 Select the table output values to estimate.
  - a In the Control and Estimation Tools Manager GUI, select the **Variables** node under the **Estimation Task** node.
  - b Click **Add** to open the Select Parameters dialog box, where you see the Simulink model parameters.
  - c Select **table**, and 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.

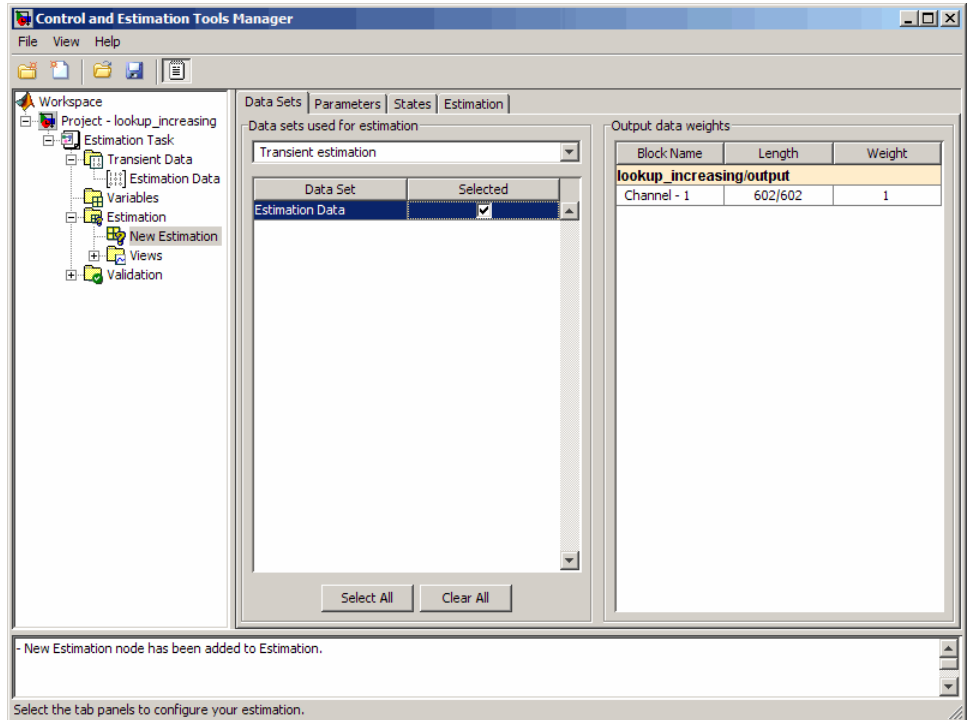
- d Select the **Estimation** node, and click **New** to add a **New Estimation** node.

- e Select the **New Estimation** node. In the **Parameters** tab, select the **Estimate** check box to specify the lookup table values, table, for estimation.





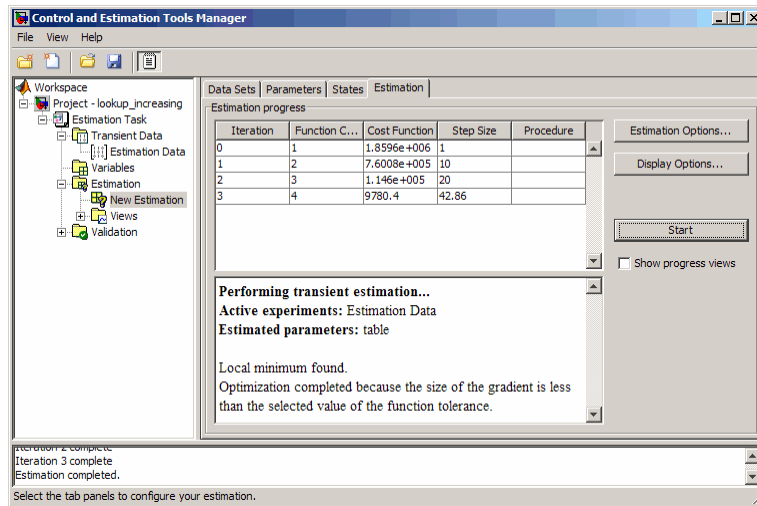
- 4 In the **Data Sets** tab of the **New Estimation** node, select the **Selected** check-box to specify the estimation data set.



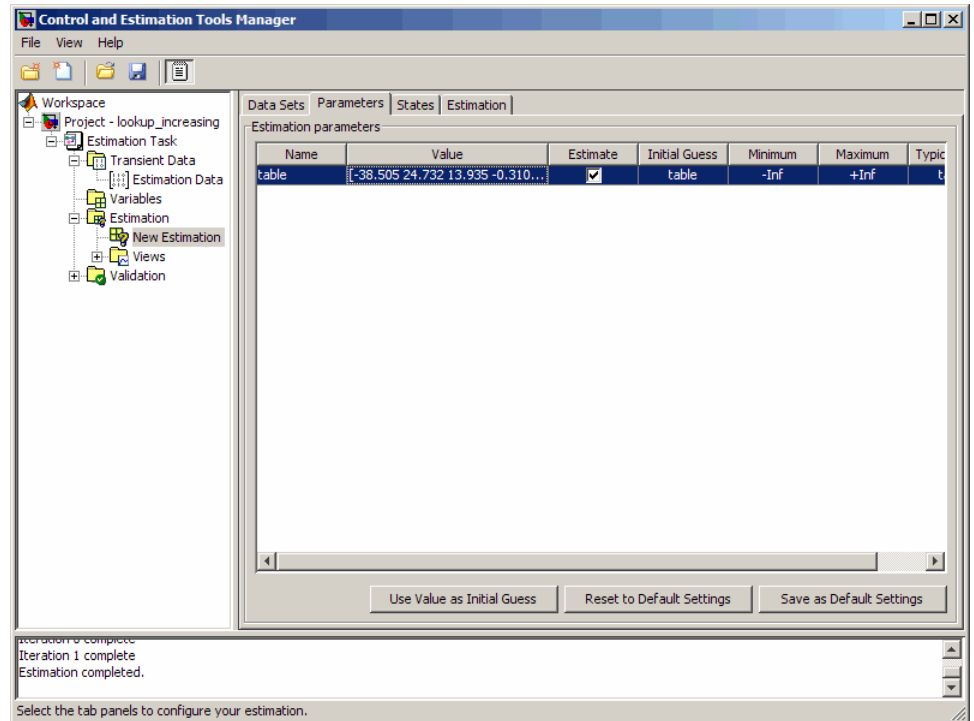
- 5 Estimate the parameters using the default settings.

- 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:

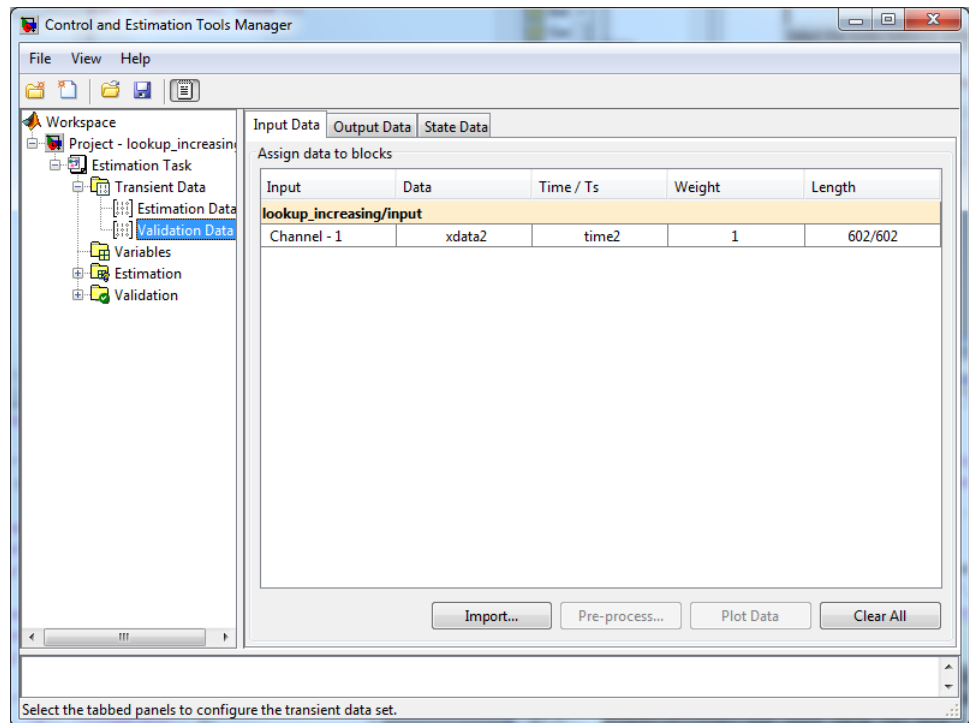
- Residuals plot
- Measured and simulated data plots

To validate the estimation results:

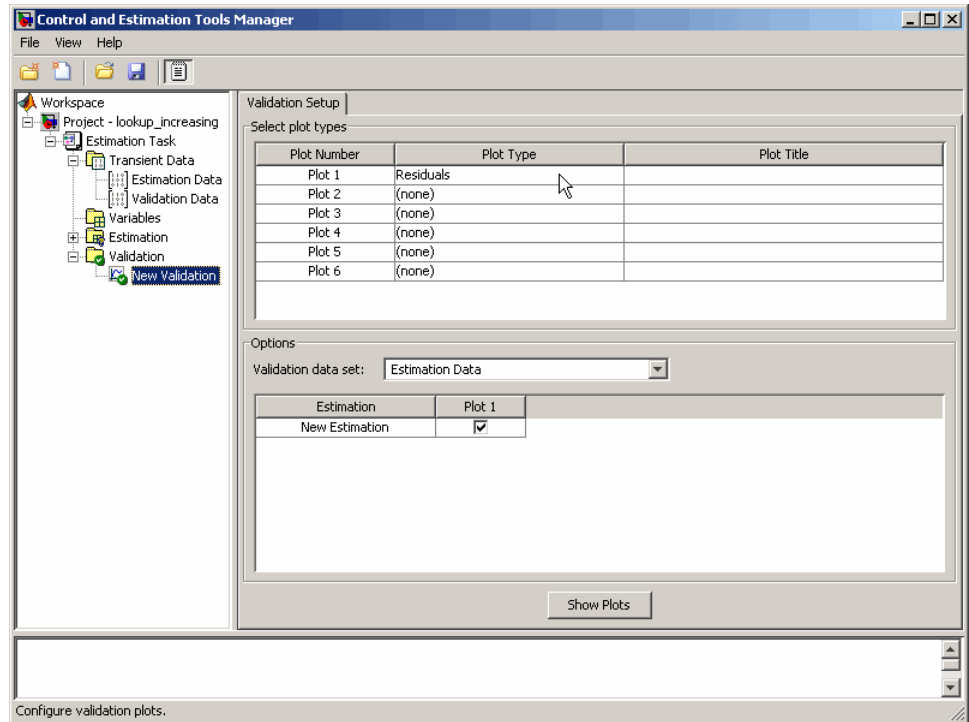
- 1 Import the validation I/O data, `xdata2` and `ydata2`, and time vector, `time2`, in the Control and Estimation Tools Manager GUI.

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')
```

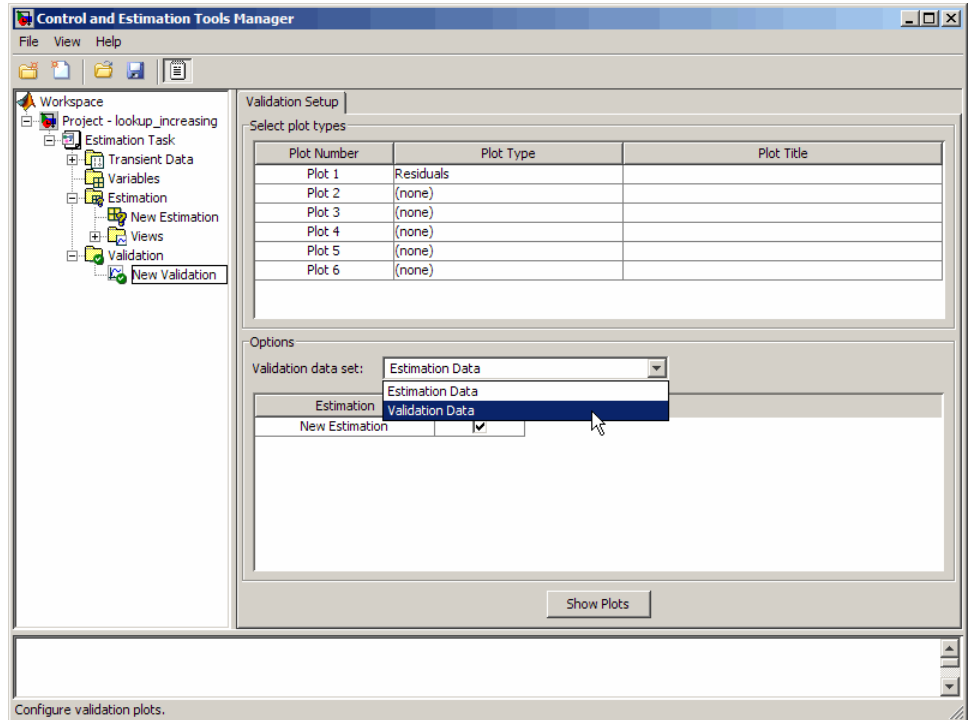


This project also contains the Residuals plot already configured in the **Select plot types** area of the GUI, as shown in the next figure. For more information on how to configure this plot, see “Compare Measured and Simulated Responses (GUI)” on page 2-46.

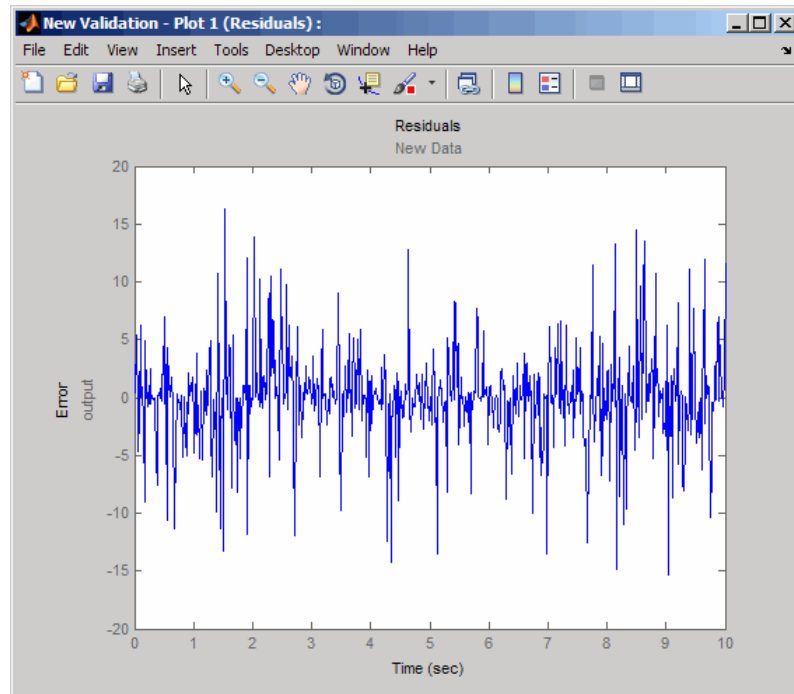


- 2 Plot and examine the residuals.
  - a Select the **New Validation** node under the **Validation** node.

- b In the **Options** area, select **Validation Data** from the **Validation data set** drop-down list.



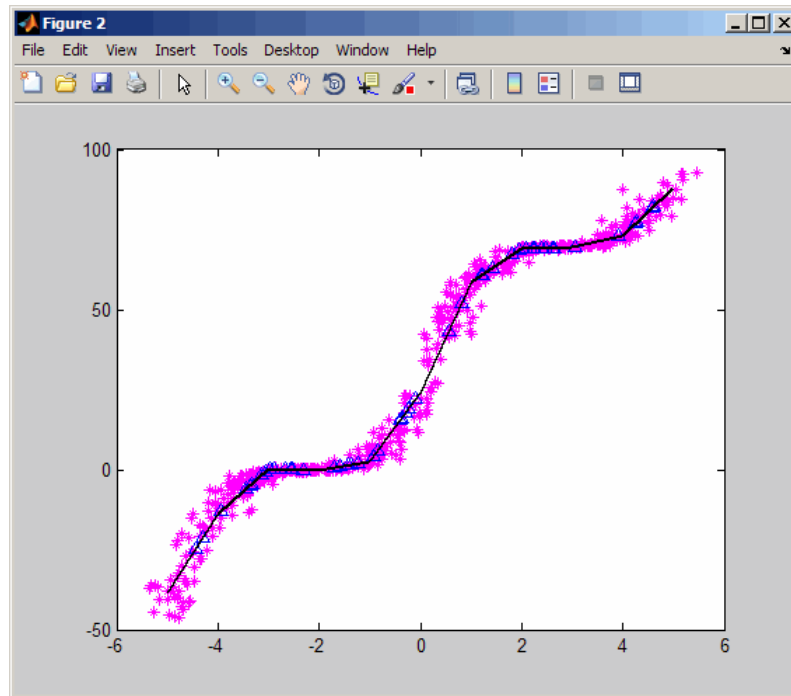
- c Click **Show Plots** to open the residuals plot.



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 5-59

## 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 (2D Stair-Fit) — Multidimensional adaptive lookup table

---

**Note** Use the Adaptive Lookup Table (2D Stair-Fit) block to create lookup tables of three or more dimensions.

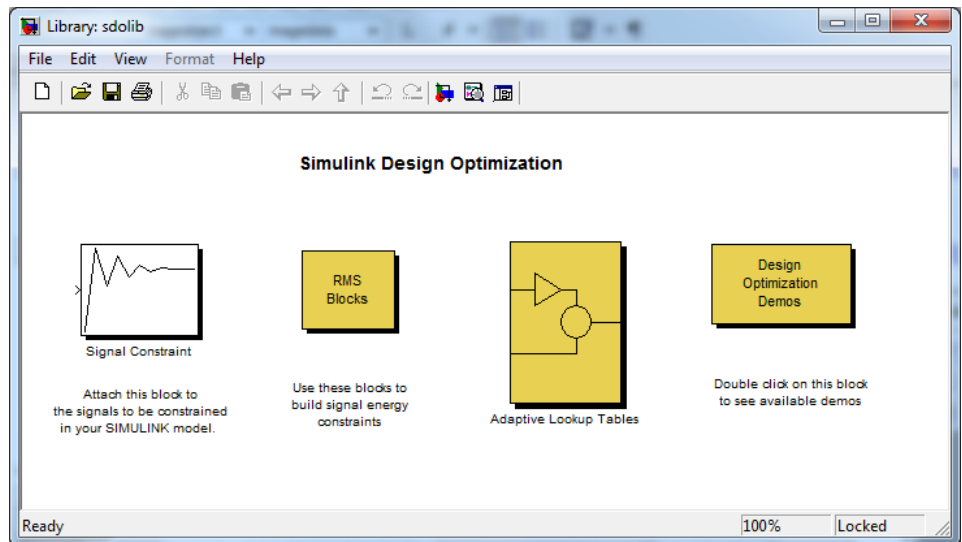
---

To access the Adaptive Lookup Tables library:

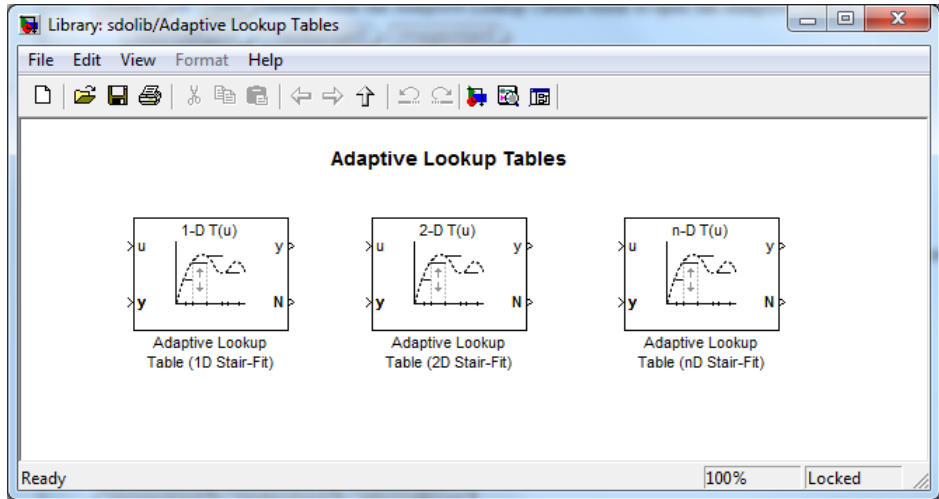
- 1 Type the following command at the MATLAB prompt:

```
sdolib
```

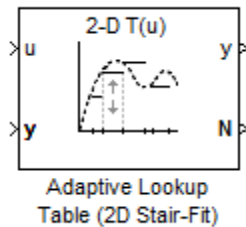
The Simulink Design Optimization library opens as shown in the next figure.



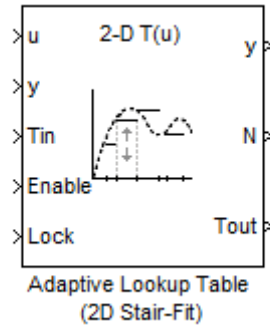
- 2 Double-click the Adaptive Lookup Tables block to open the Adaptive Lookup Tables library, as shown in the next figure.



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 Block Reference.



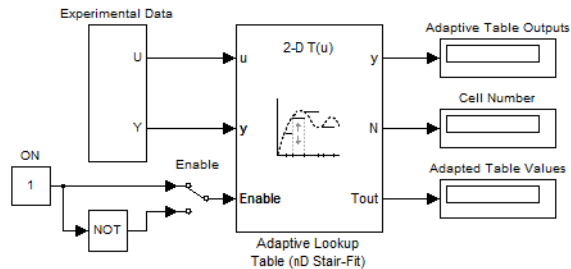
### Adaptive Lookup Table Block Showing Inputs and Outputs

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

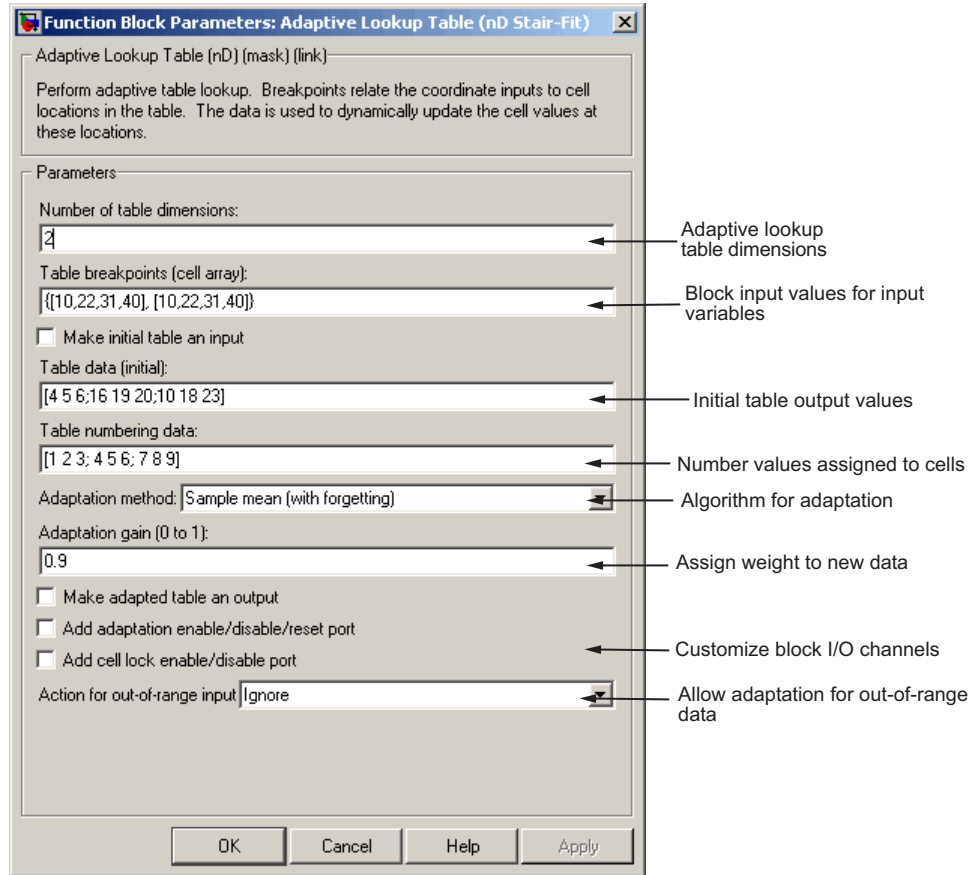
In this figure, the Experiment Data block imports a set of experimental data into Simulink through MATLAB workspace variables. The initial table is specified in the block mask parameters. When the simulation runs, the initial table begins to adapt to new data inputs and the resulting table is copied 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 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  $i^{\text{th}}$  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  $n^{\text{th}}$  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 5-42 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  $\lambda$  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 5-56

### 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.

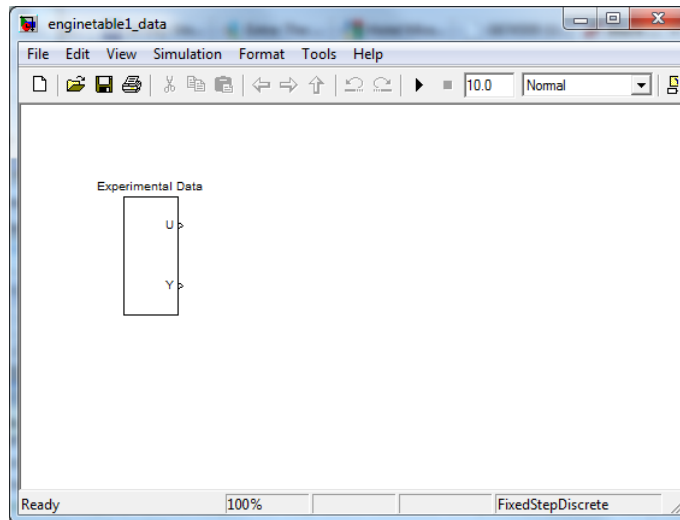
### Building a Model Using Adaptive Lookup Table Blocks

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.

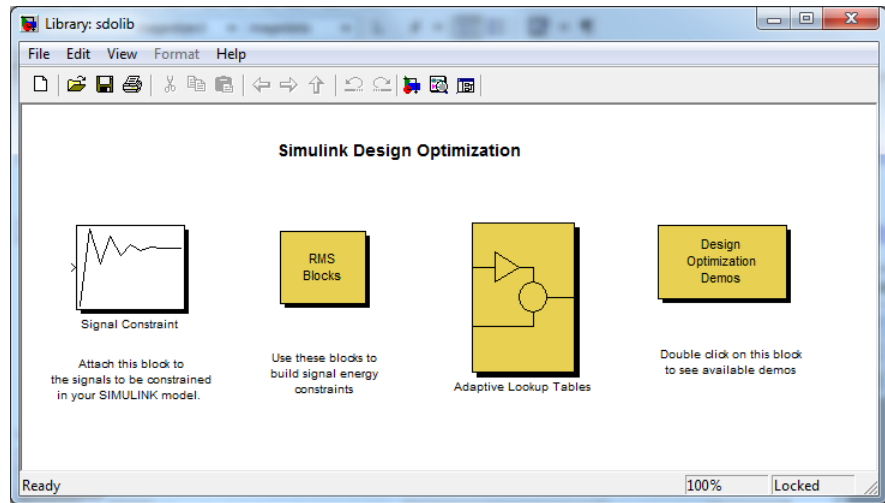


This command also loads the variables  $X$ ,  $Y$  and  $Z$  into the MATLAB workspace. To learn more about this data, see “About the Data” on page 5-45.

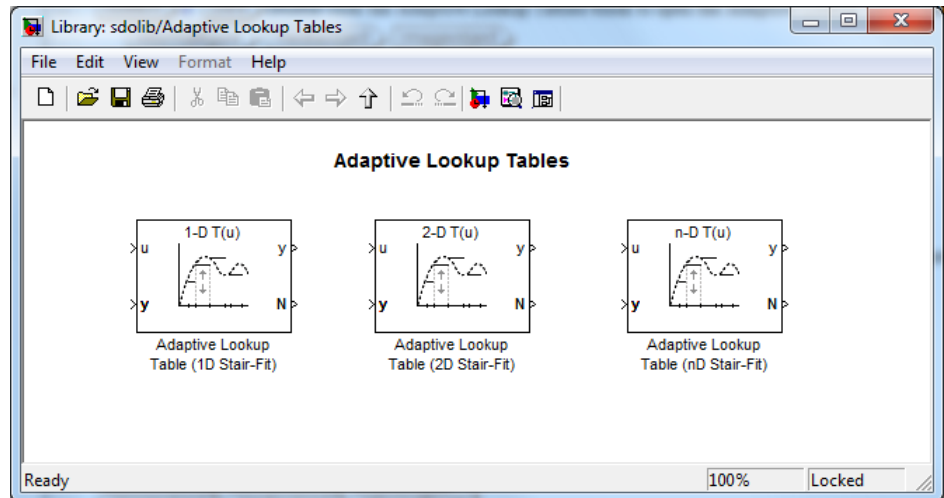
**2** Add an Adaptive Lookup Table block to the Simulink model.

- Open the Simulink Design Optimization library by typing the following command at the MATLAB prompt:

```
sdolib
```

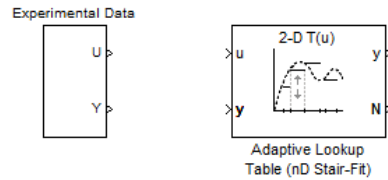


- b** Double-click the Adaptive Lookup Tables block to open the Adaptive Lookup Tables library.

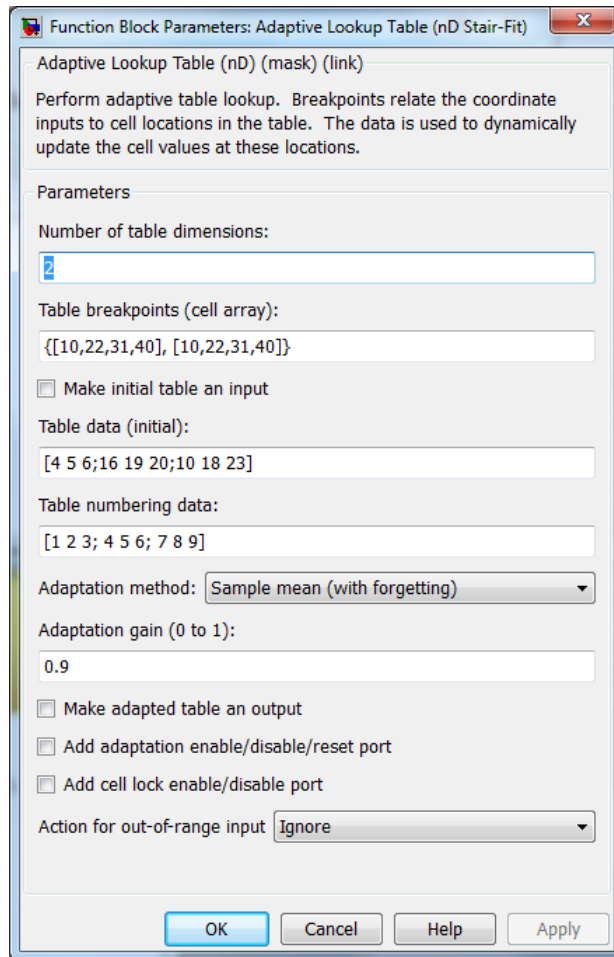


Simulink Design Optimization software provides three Adaptive Lookup Table blocks. In this example, you use the Adaptive Lookup Table (nD Stair-Fit) block to model your system. To learn more about the blocks, see [Block Reference](#).

- 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.

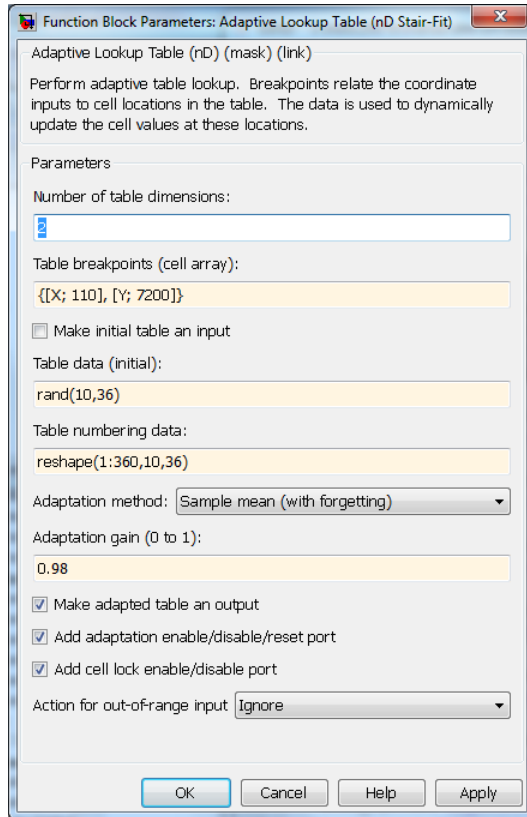
This selection specifies that the software ignores any time-varying inputs outside the range of input breakpoints during adaptation.

---

**Tip** To learn more, see Adaptive Lookup Table (nD Stair-Fit) block reference page.

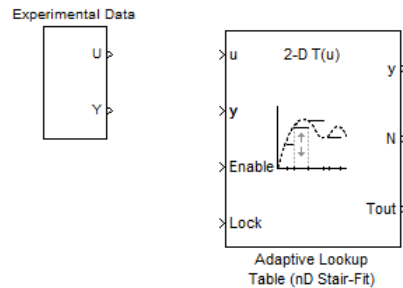
---

After you configure the parameters, the block parameters dialog box looks like the following figure.

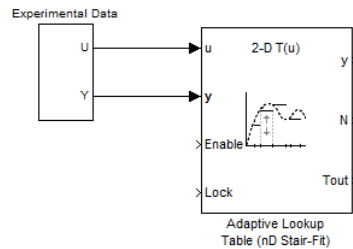


**h** Click **OK** to close the Function Block Parameters dialog box.

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.



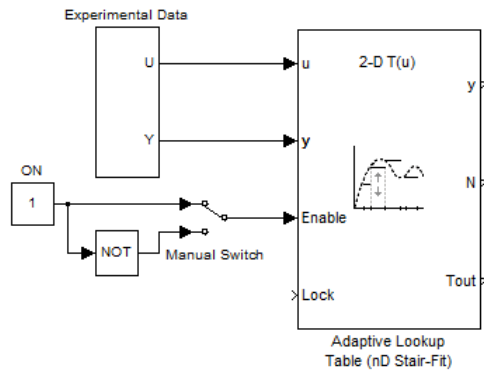
---

**Tip** To learn how to connect blocks in the Simulink model window, see “Connecting Blocks” in the Simulink documentation.

---

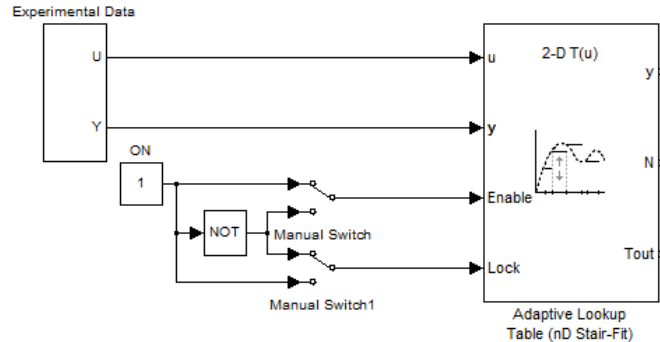
- 6 Design a logic using Simulink blocks to enable or disable the adaptation process. Connect the logic to the Adaptive Lookup Table block, as shown in the following figure.



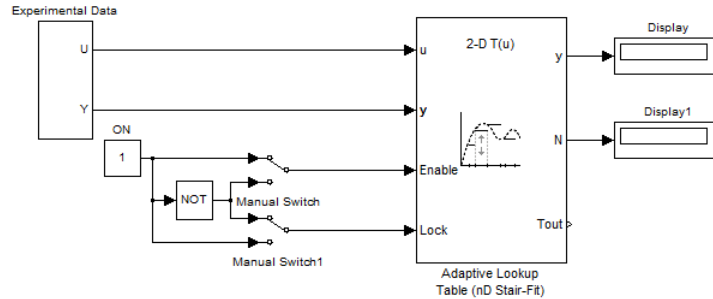


This logic outputs an initial value of 1 which enables the adaptation process.

- 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.

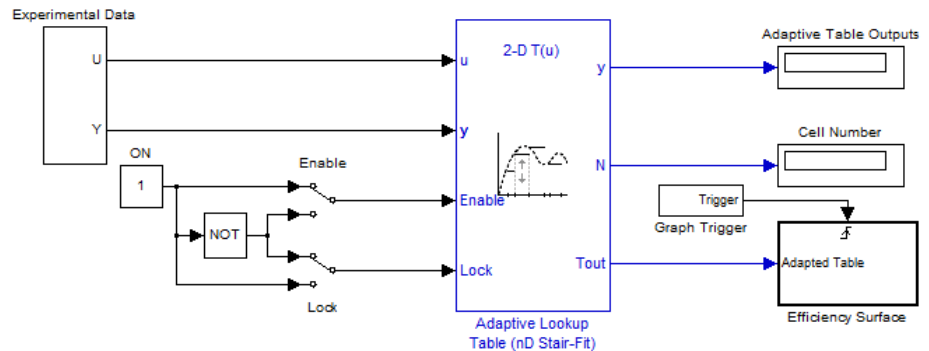


During simulation, the Display blocks show the following:

- 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, 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.

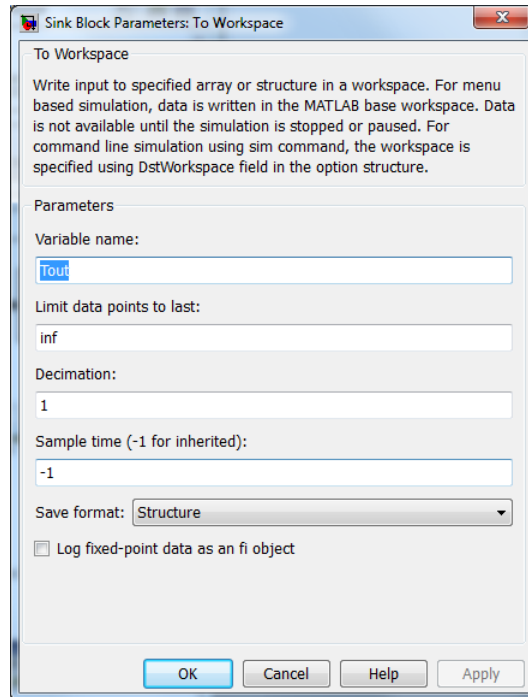


- 10 Connect a To Workspace block to export the adapted table values:

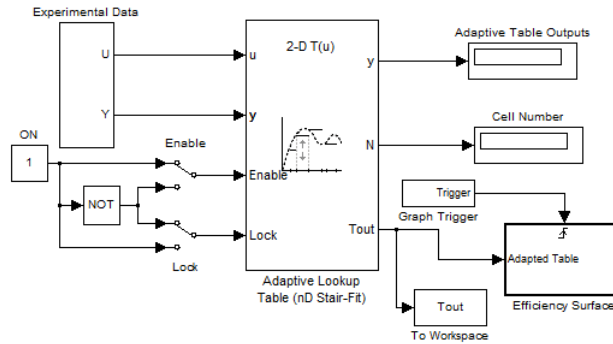
- In the Simulink Library Browser, select the **Simulink** > **Sinks** library, and drag the To Workspace block to the model window.

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 5-56.

## 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.

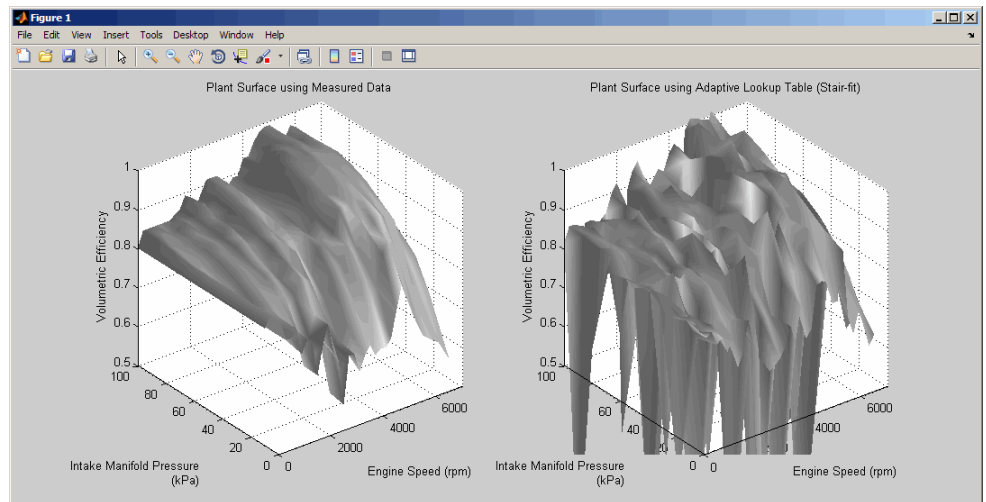


The simulation time of infinity specifies that the adaptation process continues as long as the input and output values of the engine change.

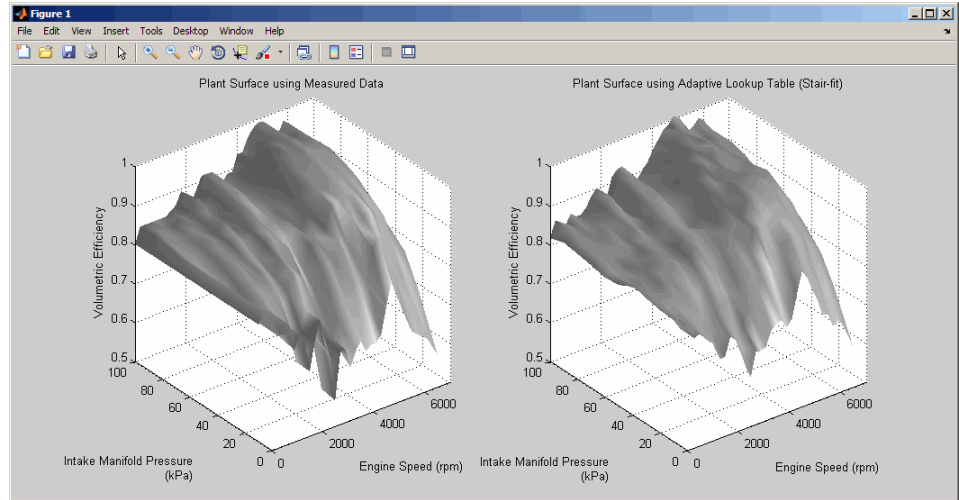
- 2 In the Simulink model window, select **Simulation > Start** to start the adaptation process.

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  $T_{out}$  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.
- 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 Simulink Coder™ 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.



# Examples

---

Use this list to find examples in the documentation.

## **Parameter Estimation**

“Estimate Initial States of a Mass-Spring-Damper System (GUI)” on page 2-71

“F14 Parameters and Initial State Estimation (Code)” on page 2-89

“RC Circuit Parameters and Initial State Estimation (Code)” on page 2-100

## **Parameter Optimization**

“Optimize Parameters for Robustness (GUI)” on page 3-162

## **Optimization-Based Linear Control Design**

“Example — Frequency-Domain Optimization for LTI System” on page 4-49

## 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



## A

- acceleration 2-53
- adaptation gain 5-43
- adaptive lookup tables 5-2
- adding data sets 1-4
- algorithms
  - response optimization 3-65

## C

- command-line estimation 2-87
- common questions 3-186
- constraint bounds 3-54
  - positioning exactly 3-19 4-7
  - splitting 3-20 4-8
- constraint segments
  - moving 3-18 4-6
- constraints
  - positioning bounds 3-54
- current responses
  - plotting 3-74

## D

- data
  - detrending 1-19
  - filtering 1-19
  - preprocessing 1-15
- Data Import dialog box 1-10
- data sets
  - adding 1-4
- design and analysis plots
  - creating 4-51
- design requirements
  - example 4-57
- desired responses
  - constraint bounds 3-54
  - reference signals 3-25 4-13
  - step responses 3-21 3-47 4-9 4-35
- detrending data 1-19

- different time lengths 2-33
- display options for estimation 2-38

## E

- Edit Design Requirement dialog box 3-19 4-7
- estimation
  - display options 2-38
  - example of command-line estimation 2-89
  - from the command line 2-87
  - running 2-39
  - selecting parameters 2-7
  - selecting states 2-68
  - setting up a project 1-4

## F

- filtering data 1-19

## H

- handling outliers 1-19

## I

- importing
  - initial conditions 2-18
  - transient data 1-6
- initial conditions
  - example of estimating 2-71
  - importing 2-18
- initial guesses 2-12
- intermediate responses
  - plotting 3-74

## L

- loading 3-196
- lookup tables
  - adaptive 5-2

**M**

multiple projects and tasks 2-83

**O**

optimization

setting options for 2-25

**P**

parallel computing

speed up 3-173

parameter estimation 2-53

parameter estimation accelerator mode 2-53

parameters

selecting for estimation 2-7

specification of 2-10

tuned 3-54

uncertain 3-157

positioning constraints

snapping to horizontal 3-19 4-7

snapping to vertical 3-19 4-7

preprocessing data 1-15

projects

definition of 2-82

saving 2-85

**R**

reference signals

specifying 3-25 4-13

tracking 3-25 4-13

response optimization

accelerating 3-171

example with control design in SISO Design

Task 4-49

gradient type 3-66

maximally feasible solutions 3-66

running 3-85

starting 3-85

task within Control and Estimation Tools

Manager 4-54

termination criteria 3-66

tolerances 3-66

response optimization accelerator mode 3-171

response optimization projects

saving 3-196

response optimization results

Optimization Progress dialog box 3-86

plots 3-85

tuning 3-64

response plots

editing properties 3-74

running an estimation 2-39

**S**

Sample mean 5-42

Sample mean with forgetting 5-43

saving 3-196

saving projects 2-85

selecting views 2-21

setting options

for optimization 2-25

for simulation 2-25

upper/lower bounds 2-12

signal responses

plotting 3-74

simulation

setting options for 2-25

simulation with 2-32

specifying parameters 2-10

states

selecting for estimation 2-68

step responses

specifications 3-21 3-47 4-9 4-35

**T**

transient data



- importing 1-6
- troubleshooting
  - response optimization 3-186
- tuned compensator elements
  - example 4-56
- tuned parameters
  - results 3-86
  - specifying 3-54
  - undoing 3-86

## **U**

- uncertain parameters 3-157

- grid 3-158
- random 3-158
- specifications 3-158
- specifying 3-157
- upper/lower bounds
  - setting 2-12

## **V**

- views
  - selecting 2-21