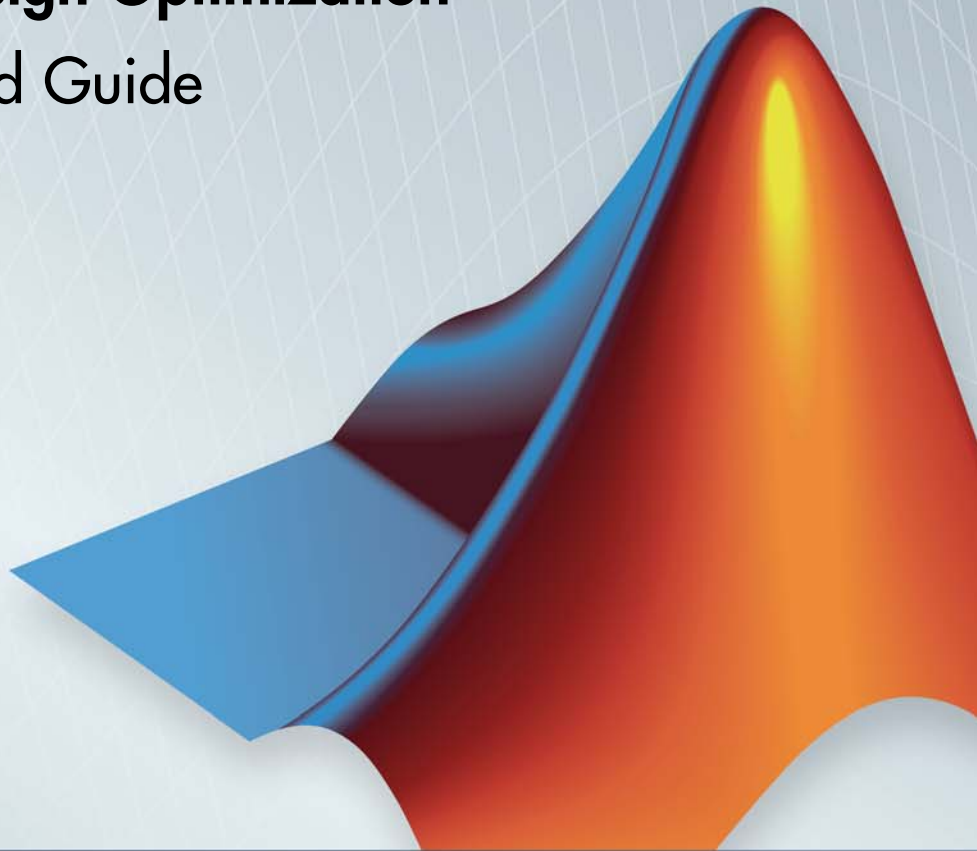


Simulink® Design Optimization™

Getting Started Guide

R2014a



MATLAB® & SIMULINK®



How to Contact MathWorks



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



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



508-647-7000 (Phone)



508-647-7001 (Fax)



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

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

Simulink® Design Optimization™ Getting Started Guide

© COPYRIGHT 1993–2014 by The MathWorks, Inc.

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

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

Trademarks

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See www.mathworks.com/trademarks for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

Patents

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

Revision History

March 2009	Online only	New for Version 1 (Release 2009a)
September 2009	Online only	Revised for Version 1.1 (Release 2009b)
March 2010	Online only	Revised for Version 1.1.1 (Release 2010a)
September 2010	Online only	Revised for Version 1.2 (Release 2010b)
April 2011	Online only	Revised for Version 1.2.1 (Release 2011a)
September 2011	Online only	Revised for Version 2.0 (Release 2011b)
March 2012	Online only	Revised for Version 2.1 (Release 2012a)
September 2012	Online only	Revised for Version 2.2 (Release 2012b)
March 2013	Online only	Revised for Version 2.3 (Release 2013a)
September 2013	Online only	Revised for Version 2.4 (Release 2013b)
March 2014	Online only	Revised for Version 2.5 (Release 2014a)

Product Overview

1

Simulink Design Optimization Product Description ..	1-2
Key Features	1-2
Optimization Support for Simulink Models Using Third-Party Applications	1-3
Speeding Up Design Optimization Using Parallel Computing	1-4
Required and Related Products	1-5

Parameter Estimation

2

Supported Data	2-2
Prepare Data for Parameter Estimation (GUI)	2-3
About This Tutorial	2-3
Configuring a Project for Parameter Estimation	2-4
Importing Data into the GUI	2-6
Analyzing Data	2-15
Selecting Data for Estimation	2-17
Removing Outliers	2-25
Filtering Data	2-29
Interpolating Missing Data	2-34
Saving the Project	2-36
Estimate Parameters from Measured Data (GUI)	2-39
About This Tutorial	2-39

Estimating Model Parameters Using Default Estimation	
Settings	2-43
Improving Estimation Results Using Parameter Bounds ..	2-56
Validating Estimated Model Parameters	2-61

Response Optimization

3

Supported Design Requirements	3-2
--	------------

Design Optimization to Meet Step Response

Requirements (GUI)	3-4
Model Structure	3-4
Design Requirements	3-5
Specify Step Response Requirements	3-5
Specify Design Variables	3-8
Optimize Model Response	3-12
Save the Session	3-15

Design Optimization to Meet Step Response

Requirements (Code)	3-18
Model Structure	3-18
Design Requirements	3-19
Specify Step Response Requirements	3-19
Specify Design Variables	3-20
Optimize Model Response	3-20

Design Optimization to Track Reference Signal

(GUI)	3-25
Model Structure	3-25
Design Requirements	3-26
Specify Reference Signal	3-26
Specify Design Variables	3-33
Optimize Model Response	3-36

Design Optimization Using Frequency-Domain Check

Blocks (GUI)	3-40
Model Structure	3-40
Design Requirements	3-41

Specify Design Requirements	3-41
Specify Design Variables	3-44
Optimize Design	3-46
Time-Domain Model Verification	3-52

Optimization-Based Linear Control Design

4

When to Use Optimization-Based Linear Control Design	4-2
Types of Time- and Frequency-Domain Design Requirements for Optimization-Based Control Design	4-3
Quick Start — Optimization-Based Linear Control Design	4-4
Design Optimization-Based PID Controller for Linearized Simulink Model (GUI)	4-15
About This Tutorial	4-15
Configuring a Project for Optimization-Based Control Design	4-17
Designing an Initial PID Controller to Meet Bode Magnitude and Phase Margins Requirements	4-22
Refining the Controller Design to Meet Controller Output Bounds	4-43
Saving the Project	4-59

Product Overview

- “Simulink® Design Optimization™ Product Description” on page 1-2
- “Optimization Support for Simulink Models Using Third-Party Applications” on page 1-3
- “Speeding Up Design Optimization Using Parallel Computing” on page 1-4
- “Required and Related Products” on page 1-5

Simulink Design Optimization Product Description

Estimate and optimize Simulink® model parameters

Simulink Design Optimization™ provides interactive tools, functions, and Simulink blocks for estimating and tuning Simulink model parameters using numerical optimization. An interactive tool lets you automatically estimate model parameters such as friction and aerodynamic coefficients from test data to increase model accuracy. You can preprocess test data, select model parameters to estimate, start an optimization, and validate estimation results.

You can also automatically tune design parameters in a Simulink model to meet objectives such as improved system performance and minimized energy consumption. Using design optimization techniques, you can meet both time- and frequency-domain constraints such as overshoot and phase margin. You can also jointly optimize physical plant parameters and algorithmic or controller gains to maximize overall system performance.

Key Features

- Model parameter estimation from test data
- Simultaneous optimization of time- and frequency-domain responses of Simulink models (with Simulink Control Design™)
- Graphical specification of response requirements and visual monitoring of the optimization progress
- Optimization of parameters to meet requirements specified by Model Verification blocks
- Custom constraints and cost functions for response optimization
- Scripting interface for programmatic specification of design optimization problems
- Robust design optimization, accounting for parameter variation or uncertainty

Optimization Support for Simulink Models Using Third-Party Applications

You can use Simulink Design Optimization software to optimize Simulink models that invoke third-party simulation tools or contain legacy simulation code. To do so, use the S-function block in Simulink. When using the command-line functions, use MATLAB® MEX functions.

References

Cherian, V., Shenoy, R., Stothert, A., Shriver, J. et al., "Model-Based Design of a SUV Anti-rollover Control System" SAE Technical Paper 2008-01-0579, 2008, doi:10.4271/2008-01-0579.

Concepts

- "Introducing MEX-Files"

Speeding Up Design Optimization Using Parallel Computing

When you have the Parallel Computing Toolbox™ software, you can use parallel computing to speed up parameter estimation and response optimization. When you use parallel computing, the software distributes the independent simulations on multiple MATLAB sessions. Thus, the simulations run in parallel which reduces the optimization time.

Using parallel computing may reduce the optimization time in the following cases:

- The model contains a large number of parameters to optimize, and the `Gradient descent` or `Nonlinear least squares` method is selected.
- The `Pattern search` method is selected as the optimization method.
- The model contains a large number of uncertain parameters and uncertain parameter values.
- The model is complex and takes a long time to simulate.

Required and Related Products

Simulink Design Optimization software requires MATLAB, Simulink, and Optimization Toolbox™ software.

The following table summarizes MathWorks® products that extend and complement the Simulink Design Optimization software. For current information about these and other MathWorks products, visit <http://www.mathworks.com/products/>.

Product	Description
Control System Toolbox™	Enables you to design controllers for linear time-invariant (LTI) models using optimization methods.
Global Optimization Toolbox	Provides genetic algorithms, and direct search methods to estimate and optimize model parameters.
Neural Network Toolbox™	Provides Simulink models of neural networks for optimization-based control design.
Parallel Computing Toolbox	Enables parallel computing on multicore processors and multiprocessor networks to speed up estimation and optimization.
Simulink Control Design	Lets you linearize Simulink models. Use Simulink Design Optimization software to design controllers for linearized models using optimization methods.
System Identification Toolbox™	Lets you estimate linear and nonlinear models from measured data. Import the estimated model into Simulink software, and use Simulink Design Optimization software for optimization-based control design.

Parameter Estimation

- “Supported Data” on page 2-2
- “Prepare Data for Parameter Estimation (GUI)” on page 2-3
- “Estimate Parameters from Measured Data (GUI)” on page 2-39

Supported Data

You can estimate model parameters and initial conditions of single or multiple input and output Simulink models from *transient data*. You measure *transient data* when the system is not in steady-state to capture the system dynamics expected under normal operating conditions. For example, the response of a system to step or impulse inputs is transient data.

Simulink Design Optimization software lets you estimate model parameters from the following types of data:

- *Time-domain* data — Data with one or more input variables $u(t)$ and one or more output variables $y(t)$, sampled as a function of time. See “Import Data (GUI)”.
- *Time-series* data — Data stored in time-series objects. For more information, see “Time Series Objects” in the MATLAB documentation. See “Importing Time-Series Data into the GUI”.

Using complex data for parameter estimation is not directly supported. See “Importing Complex Data into the GUI”.

Simulink Design Optimization software estimates model parameters by comparing the transient data with simulation data generated from the Simulink model. Using optimization techniques, the software estimates the parameters and initial conditions of states to minimize a user-selected cost function. The cost function typically calculates a least-square error between the measured and simulated data. To learn more, see “Estimate Parameters from Measured Data (GUI)” on page 2-39.

Prepare Data for Parameter Estimation (GUI)

In this section...

- “About This Tutorial” on page 2-3
- “Configuring a Project for Parameter Estimation” on page 2-4
- “Importing Data into the GUI” on page 2-6
- “Analyzing Data” on page 2-15
- “Selecting Data for Estimation” on page 2-17
- “Removing Outliers” on page 2-25
- “Filtering Data” on page 2-29
- “Interpolating Missing Data” on page 2-34
- “Saving the Project” on page 2-36

About This Tutorial

- “Objectives” on page 2-3
- “About the Sample Data” on page 2-4

Objectives

In this tutorial, you learn how to import, analyze, and prepare measured input and output (I/O) data for estimating parameters of a Simulink model.

Note Simulink Design Optimization software estimates parameters from real, time-domain data only.

You learn to perform the following tasks using the GUI:

- Import data from the MATLAB workspace.
- Analyze data quality using a time plot.
- Select a subset of data for estimation.

- Remove outliers.
- Filter high-frequency noise.
- Compute missing data using interpolation.

About the Sample Data

In this tutorial, you use `spe_engine_throttle1.mat`, which contains I/O data measured from an engine throttle system. The MAT-file includes the following variables:

- `input1` — Input data samples
- `position1` — Output data samples
- `time1` — Time vector

Note The number of input and output data samples must be equal to the length of the corresponding time vector.

The engine throttle system controls the flow of air and fuel mixture to the engine cylinders. The throttle body contains a butterfly valve which opens when a driver presses the accelerator pedal. Opening this valve increases the amount of fuel mixture entering the cylinders, which increases the engine speed. A DC motor controls the opening angle of the butterfly valve in the throttle system. The input to the throttle system is the motor current (in amperes), and the output is the angular position of the butterfly valve (in degrees).

`spe_engine_throttle1` contains the Simulink model of the engine throttle system.

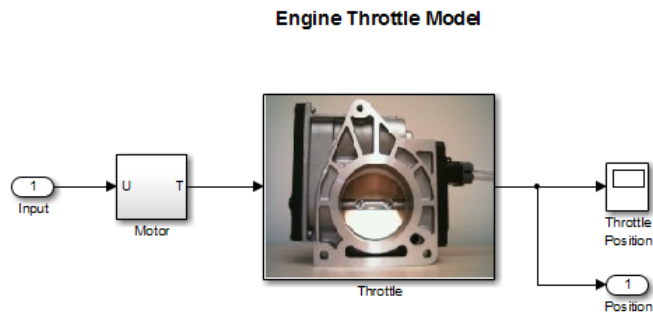
Configuring a Project for Parameter Estimation

To perform parameter estimation, you must first configure a Control and Estimation Tools Manager project.

- 1 Open the engine throttle system model by typing the following at the MATLAB prompt:

```
spe_engine_throttle1
```

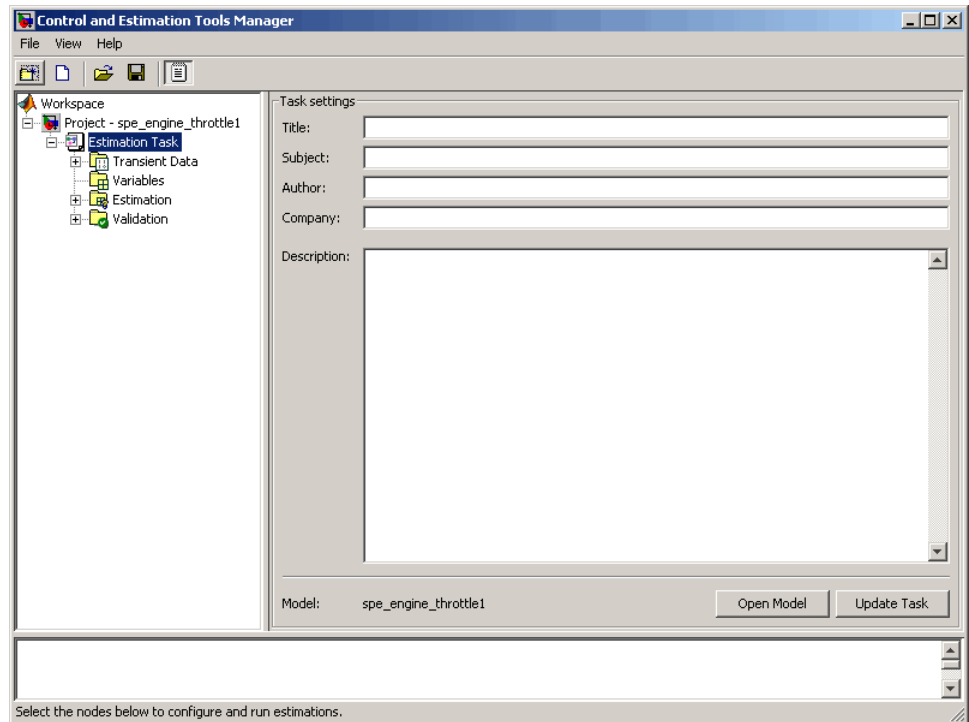
This command opens the Simulink model, and loads the data into the MATLAB workspace.



Simulink® Model of Engine Throttle System

- 2 In the Simulink model window, select **Analysis > Parameter Estimation**.

This action opens a new project named **Project - spe_engine_throttle1** in the Control and Estimation Tools Manager GUI. This project contains the **Estimation Task**, as shown in the next figure.



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

Importing Data into the GUI

- “Importing Input Data and Time Vector” on page 2-7

- “Importing Output Data and Time Vector” on page 2-12

Importing Input Data and Time Vector

In this portion of the tutorial, you import measured I/O data into the Control and Estimation Tools Manager GUI. Importing data assigns the data to the corresponding model input and output signals.

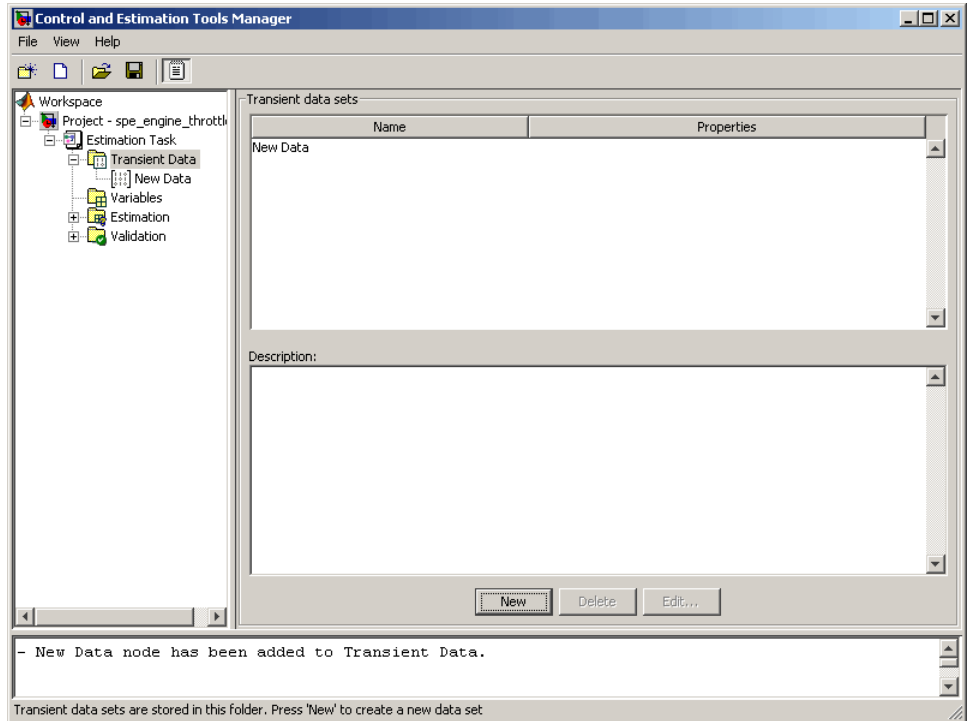
The model input and output signals are designated with the Inport Input and Outport Position blocks respectively, as shown in the figure Simulink® Model of Engine Throttle System on page 2-5. These blocks let you import I/O data into the GUI. To learn more about the blocks, see the Inport and Outport block reference pages in the Simulink documentation.

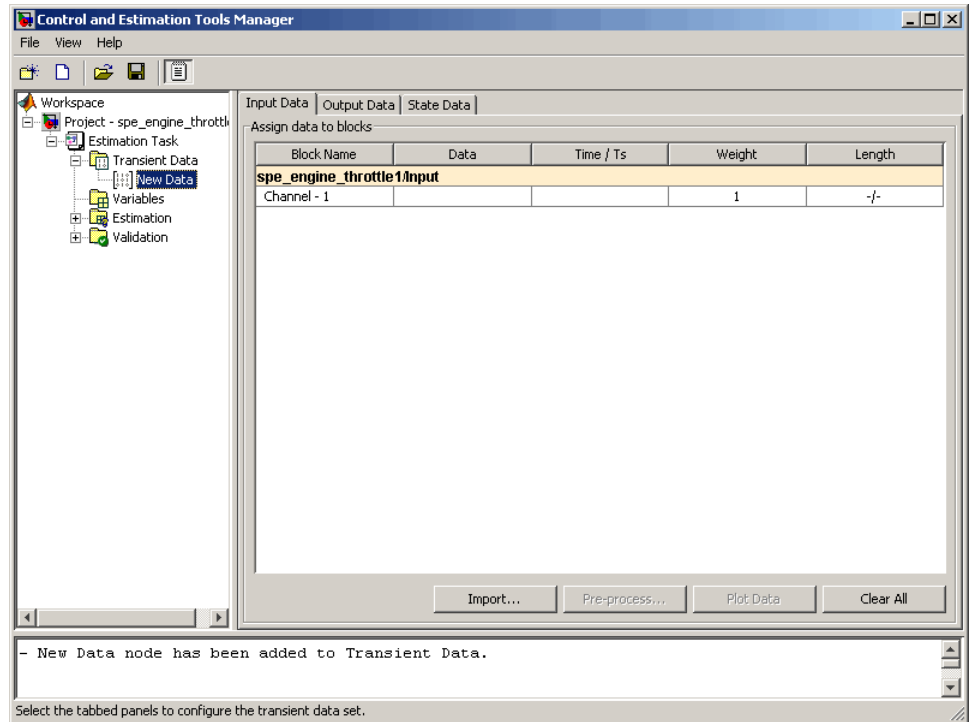
You must have already configured the parameter estimation project, as described in “Configuring a Project for Parameter Estimation” on page 2-4.

To import input data and time vector into the Control and Estimation Tools Manager GUI:

- 1 In the Control and Estimation Tools Manager GUI, select **Transient Data** under the **Estimation Task** node, and click **New**.

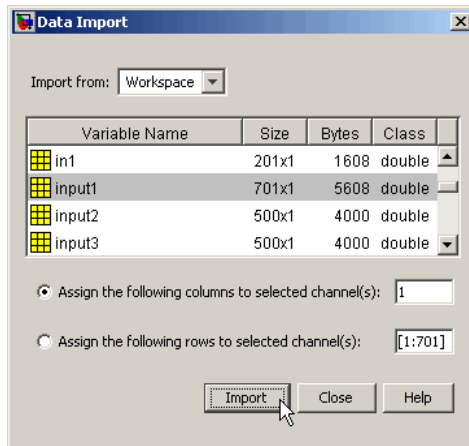
This action creates a **New Data** node under the **Transient Data** node.



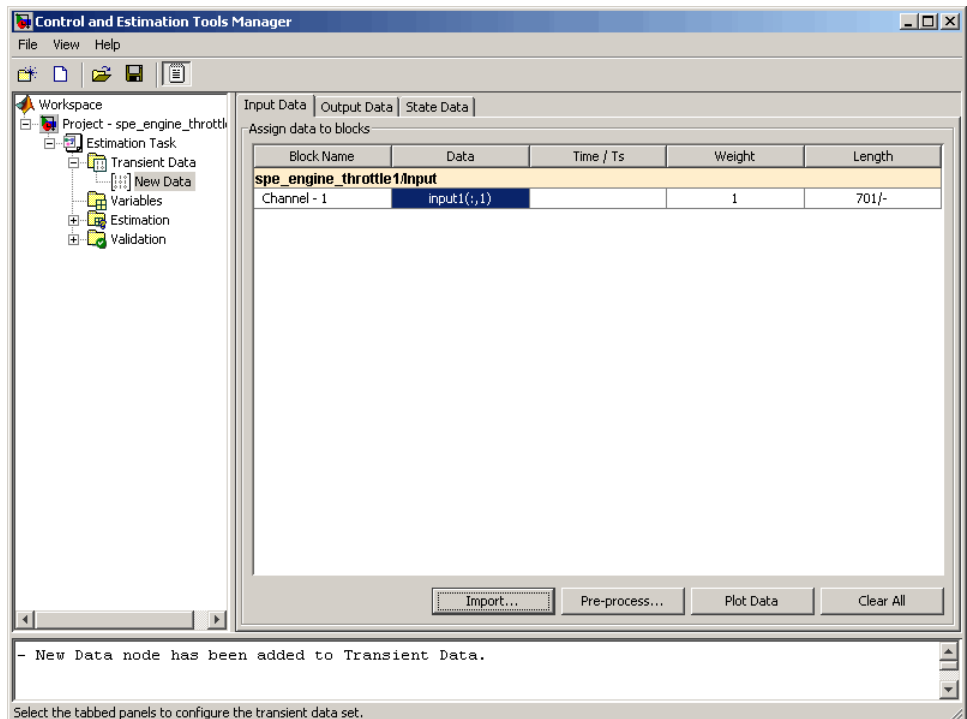
2 Select the **New Data** node.**3** In the **Input Data** tab, select the **Data** cell for **Channel - 1**, and click **Import**.

This action opens the Data Import dialog box.

- 4 In the Data Import dialog box, select input1, and click **Import**.

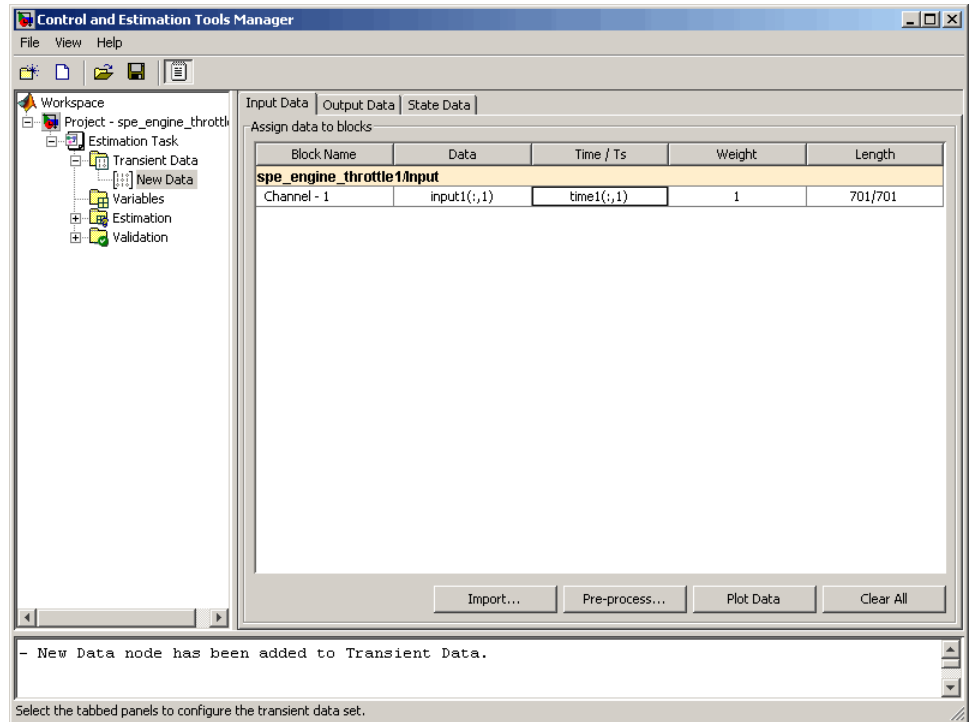


This action assigns the input data input1 to the model input signal `spe_engine_throttle1/Input`.



- 5 Select the **Time / Ts** cell for **Channel - 1**.
- 6 In the Data Import dialog box, select **time1**, and click **Import**.

This action assigns the time vector to the model input signal **spe_engine_throttle1/Input**.

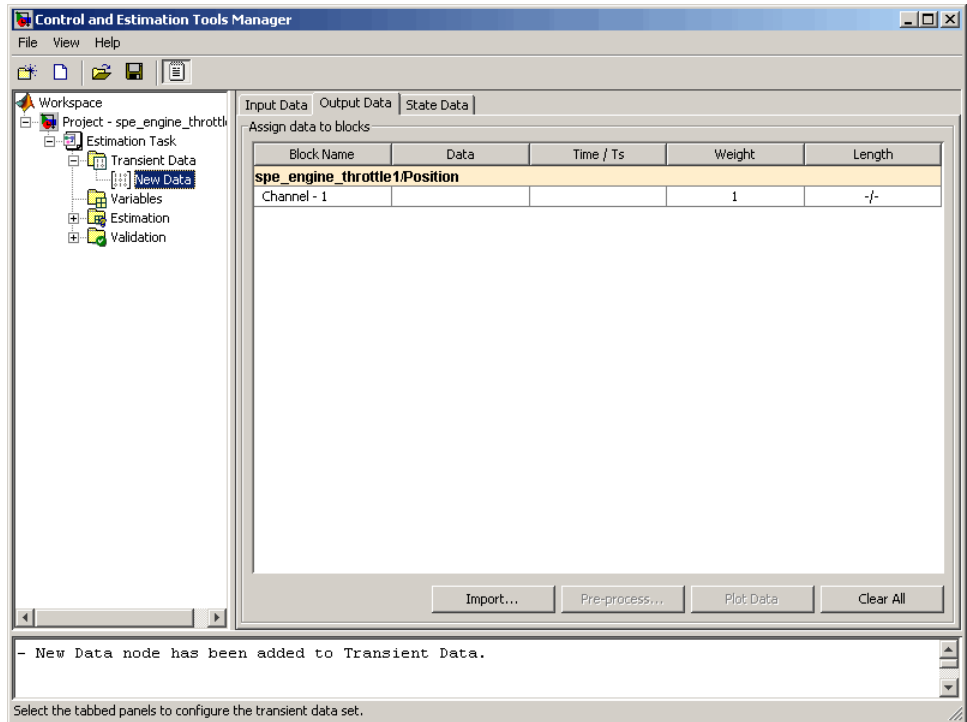


- 7 In the Data Import dialog box, click **Close**.

Importing Output Data and Time Vector

To import output data and time vector into the Control and Estimation Tools Manager GUI:

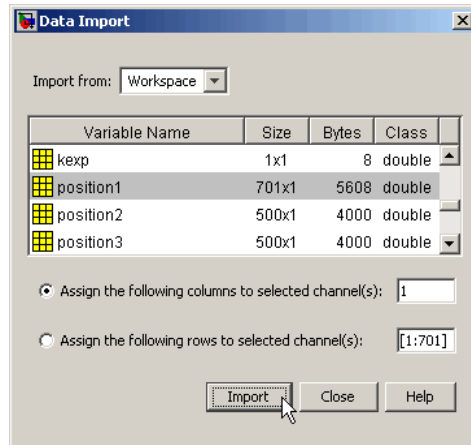
- 1 In the Control and Estimation Tools Manager GUI, select the **Output Data** tab of the **New Data** node.



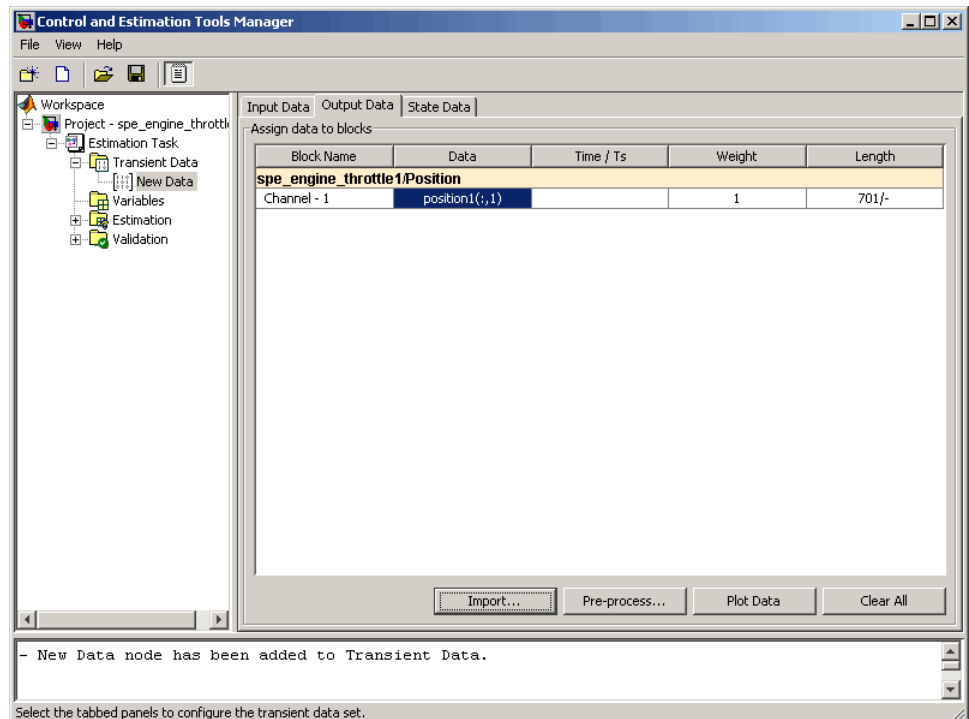
- 2 Select the **Data** cell for **Channel - 1**, and click **Import**.

This action opens the Data Import dialog box.

3 In the Data Import dialog box, select `position1`, and click **Import**.



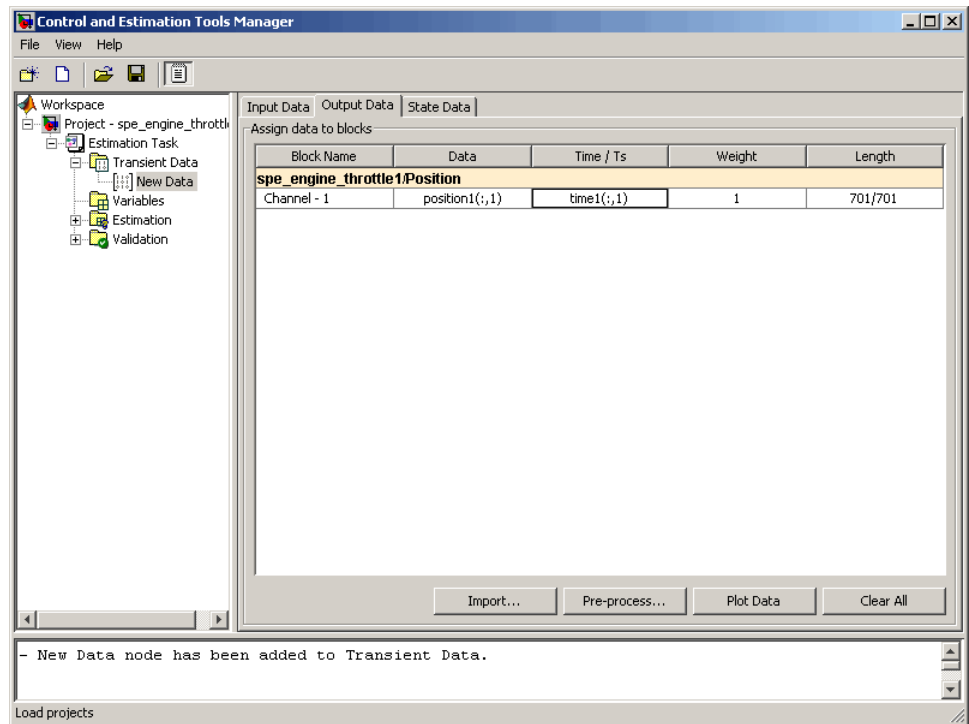
This action assigns the output data `position1` to the model output signal `spe_engine_throttle1/Position`.



4 Select the **Time / Ts** cell for **Channel - 1**.

5 In the Data Import dialog box, select **time1**, and click **Import**.

This action assigns the time vector to the model output signal **spe_engine_throttle1/Position**.



6 In Data Import dialog box, click **Close**.

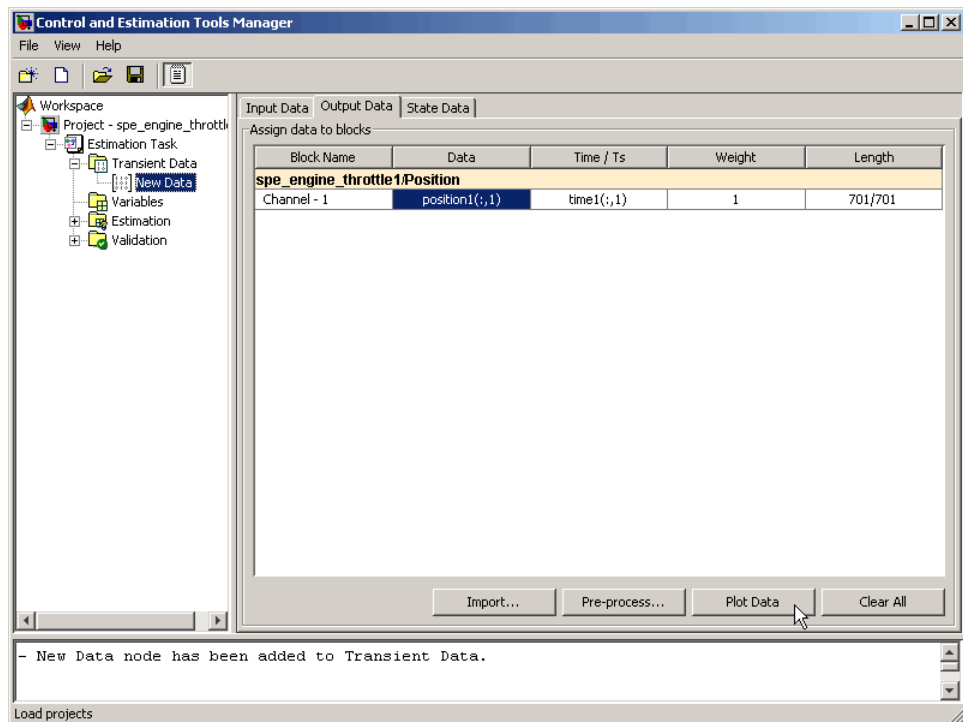
You have now imported the I/O data into the Control and Estimation Tools Manager GUI, and assigned the data to the corresponding model signals.

Analyzing Data

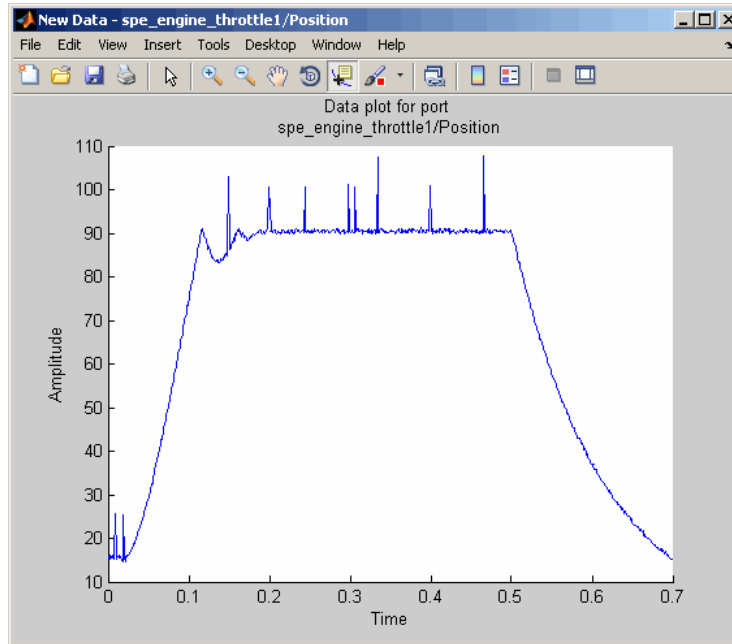
In this portion of the tutorial, you analyze the output data quality by viewing the data characteristics on a time plot. Based on the analysis, you decide whether to preprocess the data before estimating parameters. For example, if the data contains noise, you might want to filter the noise from the system dynamics before estimating parameters.

You must have already imported the data into the Control and Estimation Tools Manager GUI, as described in “Importing Data into the GUI” on page 2-6. If you have not imported the data, click [here](#).

To plot the output data on a time plot, select the `position1(:,1)` cell in the **Output Data** tab, and click **Plot Data**.



This action plots the measured output data `position1(:,1)`, as shown in the next figure.



The time plot shows the output data in response to a step input, as described in “About the Sample Data” on page 2-4. The plot shows a rapid decrease in the response after $t = 0.5$ s because the system is shut down. To focus parameter estimation on the time period when the system is active, you select the data samples between $t = 0$ s and $t = 0.5$ s, as described in “Selecting Data for Estimation” on page 2-17 section of this tutorial.

The spikes in the data indicate *outliers*, defined as data values that deviate from the mean by more than three standard deviations. They may be caused by measurement errors or sensor problems. The response also contains noise. Before estimating model parameters from this data, you remove the outliers and filter the noise, as described in “Removing Outliers” on page 2-25, and “Filtering Data” on page 2-29 sections of this tutorial.

Tip You can also plot the input data on a time plot by selecting the `input1(:,1)` cell in the **Input Data** tab, and clicking **Plot Data**.

Selecting Data for Estimation

- “Selecting Output Data” on page 2-17
- “Selecting Input Data” on page 2-23

Selecting Output Data

In this portion of the tutorial, you select a subset of I/O data for estimation. As described in “Analyzing Data” on page 2-15, the system is shut down at $t = 0.5$ s. To focus the estimation on the time period before $t = 0.5$ s, you exclude the data samples beyond $t = 0.5$ s. This operation selects the data between $t = 0$ s and $t = 0.5$ s for estimation.

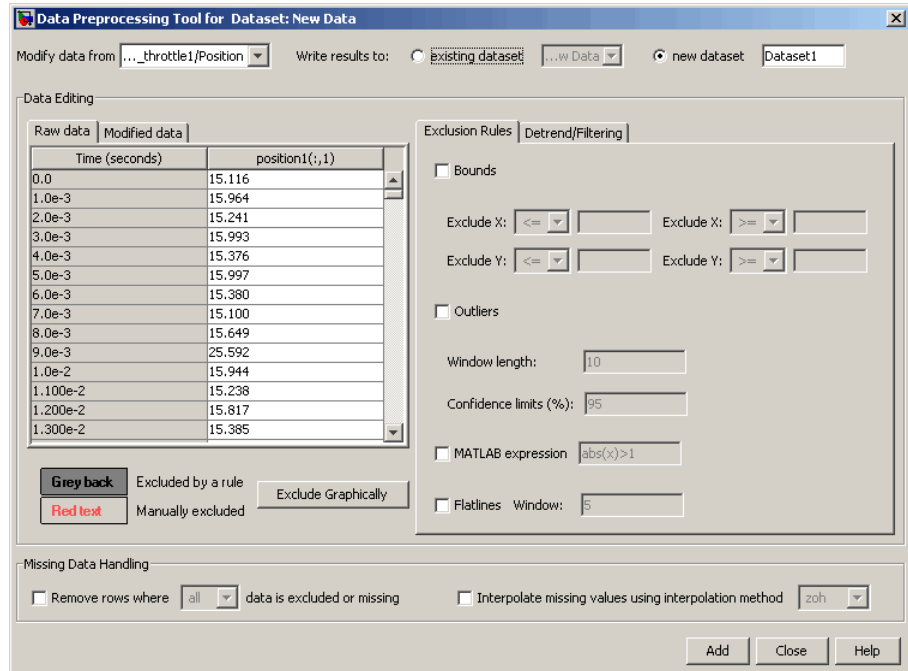
You must have already imported the data into the Control and Estimation Tools Manager GUI, as described in “Importing Data into the GUI” on page 2-6. If you have not imported the data, [click here](#).

To select the portion of data between $t = 0$ s and $t = 0.5$ s:

- 1 In the Control and Estimation Tools Manager window, select the **New Data** node under the **Transient Data** node.


- 2 Select the `position1(:,1)` cell in the **Output Data** tab, and click **Pre-process**.

This action opens the Data Preprocessing Tool GUI.

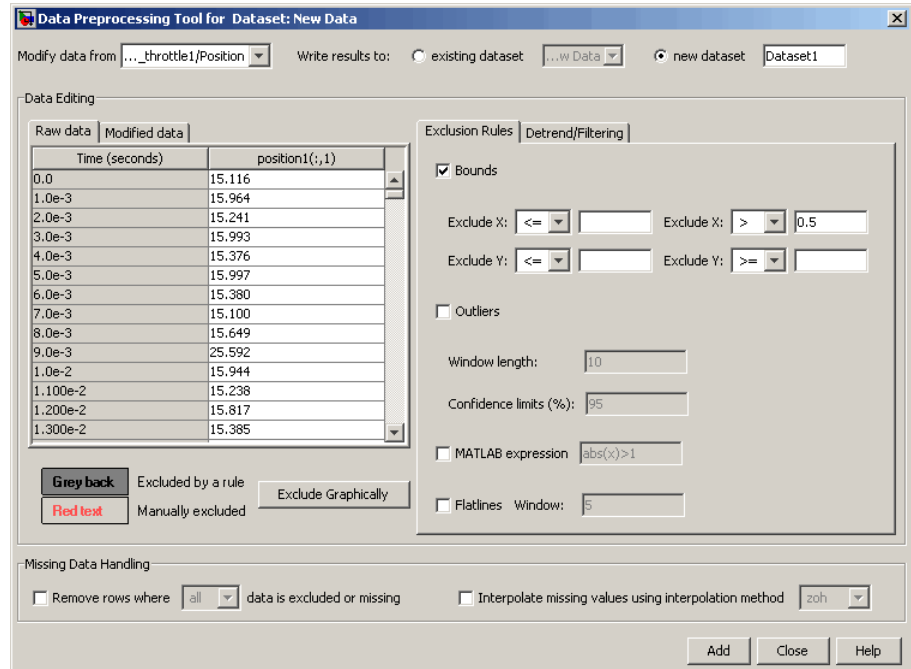


The **Data Editing** area of the Data Preprocessing Tool GUI shows the output data and time vector in the **position1(:,1)** and **Time (seconds)** columns, respectively. The Data Preprocessing Tool GUI lets you perform the following types of preprocessing operations:

- Excluding data
 - Detrending and filtering data
 - Handling missing data
- 3 To exclude the output data beyond $t = 0.5$ s:
 - a In the **Exclusion Rules** tab, select the **Bounds** check box.

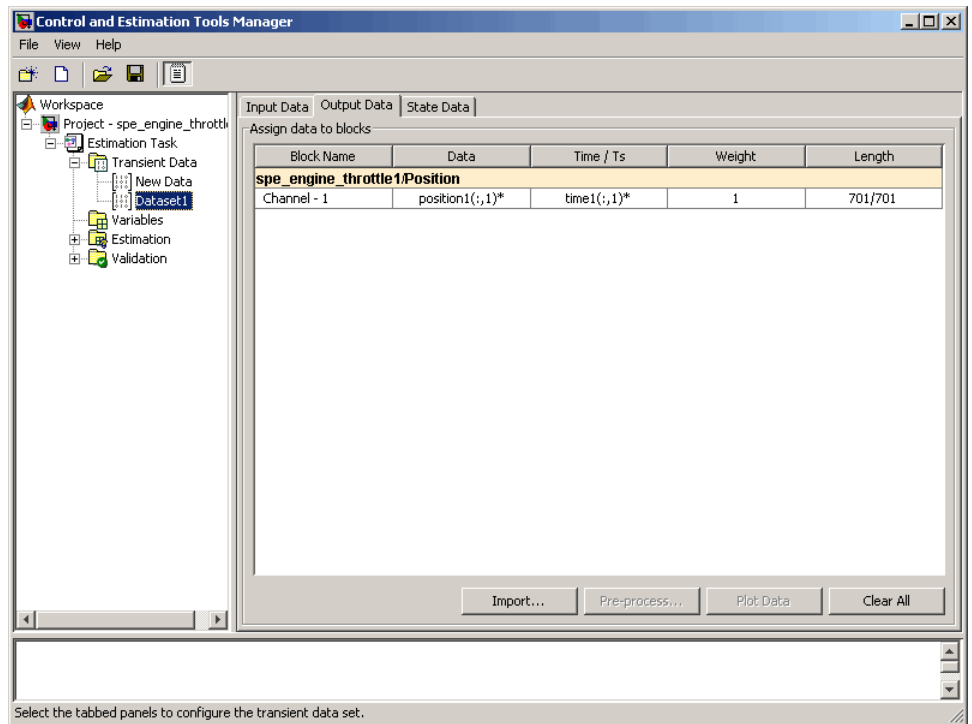
- b In the **Exclude X**  field, where **X** corresponds to the time vector, select **>** from the drop-down list. Enter **0.5** in the adjacent field to specify the upper limit of the data to select for estimation.

The Data Preprocessing Tool GUI resembles the next figure.

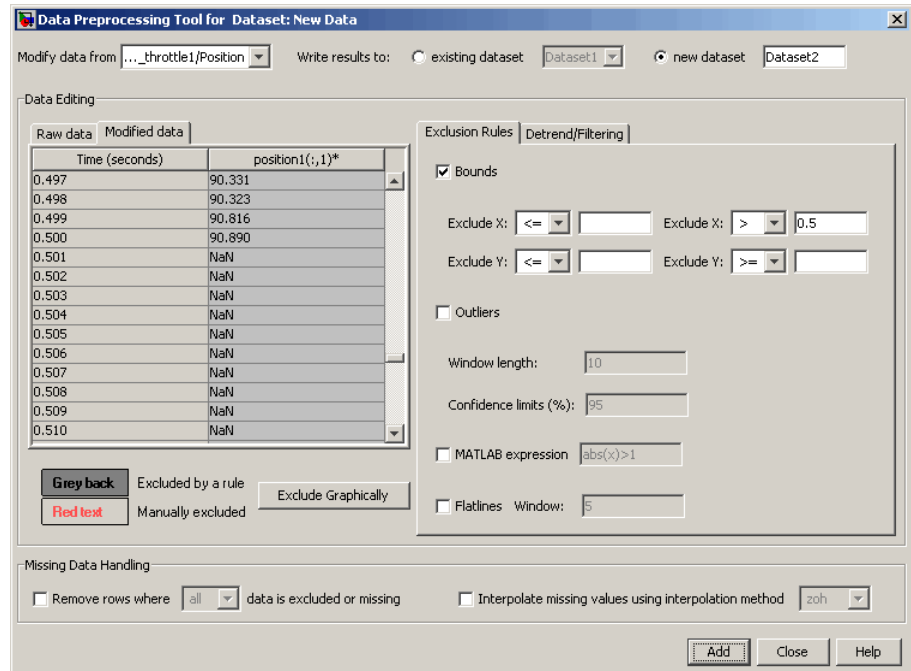


c Click **Add**.

This action adds a new **Dataset1** node under the **Transient Data** node in the Control and Estimation Tools Manager GUI. The **Dataset1** node contains the modified output data $\text{position1}(:,1)^*$ in the **Output Data** tab.



This operation also replaces the output data samples beyond $t = 0.5$ s in `position1(:,1)*` with NaNs. You can view the NaNs by selecting the **Modified data** tab in the Data Preprocessing Tool GUI, as shown in the next figure.

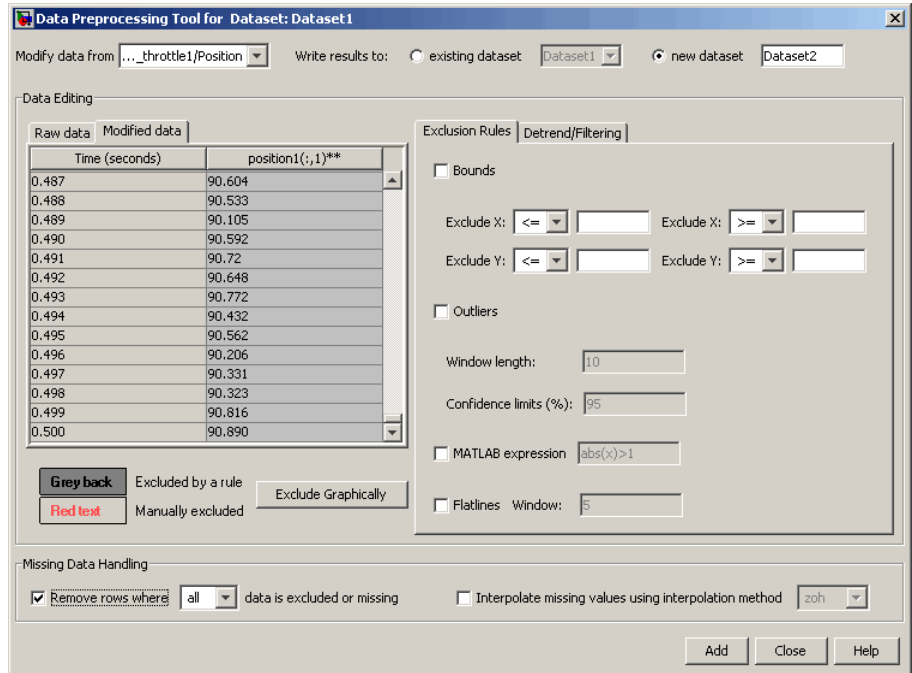


4 To remove the NaNs:

- a In the Control and Estimation Tools Manager GUI, select the **Output Data** tab of the **Dataset1** node.
- b Select the `position1(:,1)*` cell, and click **Pre-process**.

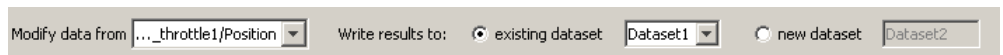
This action updates the Data Preprocessing Tool GUI with the selected data `position1(:,1)*`.

- c In the **Missing Data Handling** area, select the **Remove rows where** **data is excluded or missing** check box.



Tip You can view the results of this operation in the **Modified data** tab.

- d In the **Write results to** area, select the **existing dataset** option.

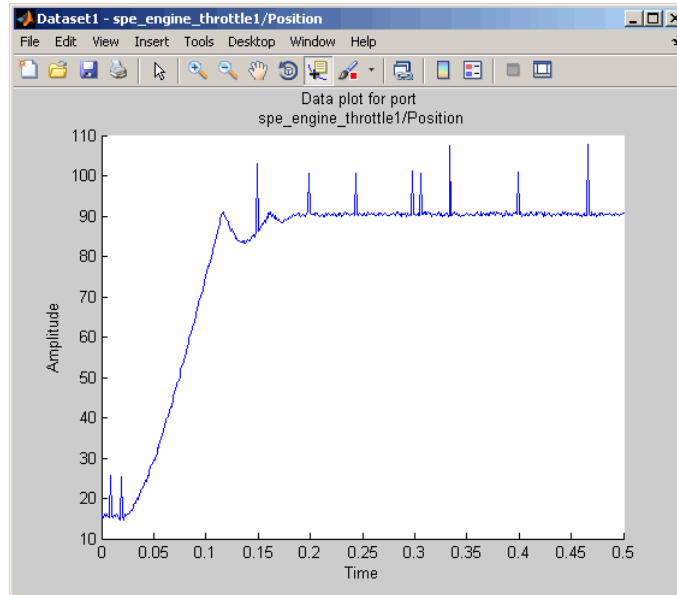


- e Click **Add**.

The Update table data dialog box appears. Click **Yes** to overwrite the `position1(:,1)*` data set with the modified data.

- 5 To plot the data, select the `position1(:,1)*` cell in the **Output Data** tab of the **Dataset1** node, and click **Plot Data**.

The selected output data from $t = 0$ s to $t = 0.5$ s is shown in the next figure.



Selecting Input Data

After you select the output data between $t = 0$ s and $t = 0.5$ s, as described in “Selecting Output Data” on page 2-17, you must also select the corresponding input data samples. This operation makes the number of I/O data samples equal.

- 1 In the Control and Estimation Tools Manager GUI, select the **New Data** node under the **Transient Data** node.
- 2 In the **Input Data** tab, select the `input1(:,1)` cell, and click **Pre-process**.

This action updates the Data Preprocessing Tool GUI with the selected data `input1(:,1)`.

3 To exclude the data beyond $t = 0.5$ s:

- a** In the **Exclusion Rules** tab, select the **Bounds** check box.
- b** In the **Exclude X** field, where **X** corresponds to the time vector, select **>** from the drop-down list. Enter **0.5** in the adjacent field to specify the upper limit of the input data to select for estimation.
- c** In the **Write results to** area, verify that the **existing dataset** option remains selected.
- d** Click **Add**.

This action adds the modified data $\text{input1}(:,1)^*$ to the **Input Data** tab of the **Dataset1** node. This operation also replaces the input data samples beyond $t = 0.5$ s in $\text{input1}(:,1)^*$ with NaNs.

4 To remove the NaNs:

- a** In the Control and Estimation Tools Manager GUI, select the $\text{input1}(:,1)^*$ cell in the **Input Data** tab of the **Dataset1** node, and click **Pre-process**.

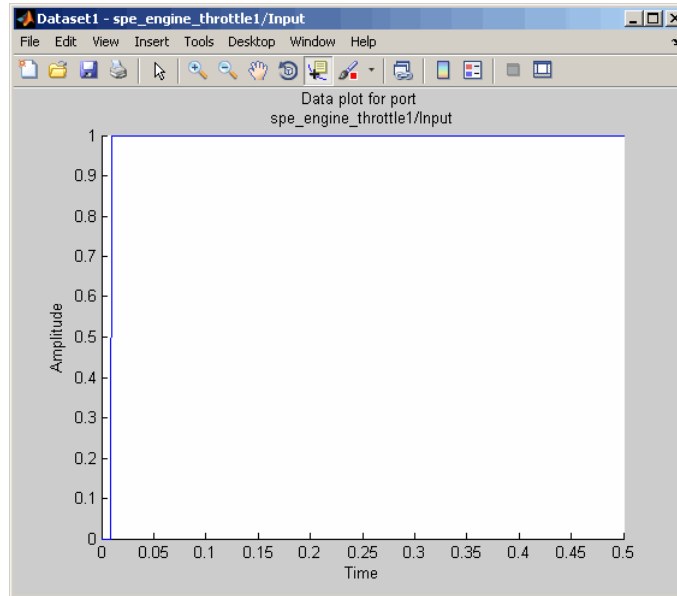
This action updates the Data Preprocessing Tool GUI with the selected data $\text{input1}(:,1)^*$.

- b** In the **Missing Data Handling** area, select the **Remove rows where** **data is excluded or missing** check box.
- c** In the **Write results to** area, verify that the **existing dataset** option remains selected.
- d** Click **Add**.

The Update table data dialog box appears. Click **Yes** to overwrite the $\text{input1}(:,1)^*$ data set with the modified data.

- 5 To plot the data, select the `input1(:,1)` cell in the **Input Data** tab of the **Dataset1** node, and click **Plot Data**.

The selected input data from $t = 0$ s to $t = 0.5$ s is shown in the next figure.



Removing Outliers

- “Why Remove Outliers” on page 2-25
- “How to Remove Outliers” on page 2-26

Why Remove 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.

Removing outliers replaces the data samples containing outliers with NaNs, which represent missing data. You interpolate the missing data values in a subsequent operation, as described in “Interpolating Missing Data” on page 2-34.

How to Remove Outliers

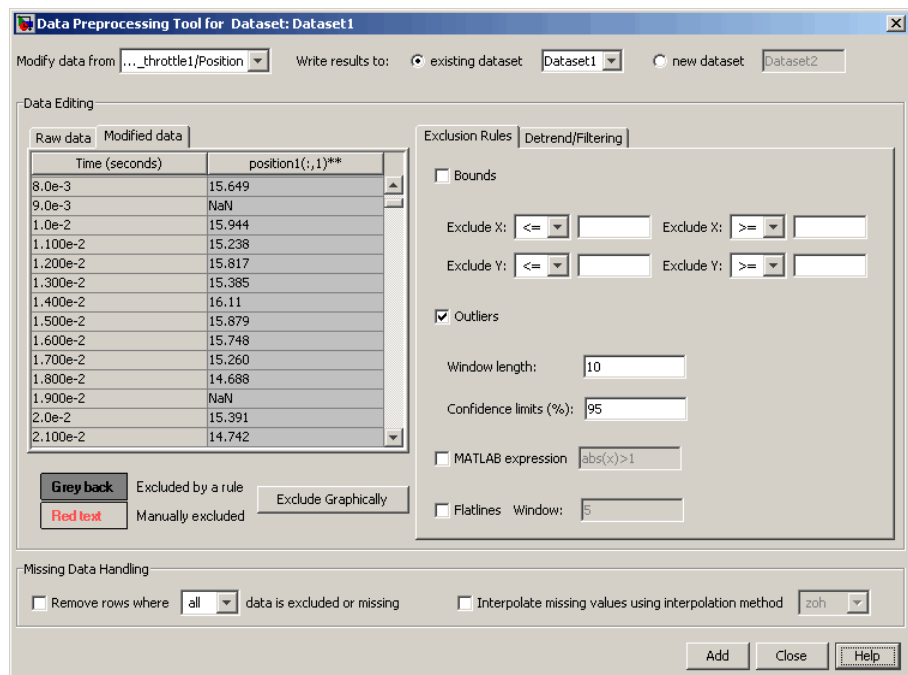
In this portion of the tutorial, you remove outliers from the output data. You must have already selected a subset of the data, as described in “Selecting Data for Estimation” on page 2-17. If you have not done this preparation, [click here](#).

- 1 In the Control and Estimation Tools Manger, select the `position1(:,1)*` cell in the **Output Data** tab of the **Dataset1** node, and click **Pre-process**.

This action updates the Data Preprocessing Tool GUI with the selected data `position1(:,1)*`.

2 In the **Exclusion Rules** tab, select the **Outliers** check box.

By default, the **Window length** and **Confidence limits** fields are set to 10 and 95 respectively. The **Window length** field specifies the number of successive data samples the software uses to compute the mean and standard deviation. The **Confidence limits** field specifies the threshold number for identifying outliers. In this example, the mean and standard deviation of 10 successive data samples are computed, and data values that exceed 95% of standard deviation are identified as outliers.



Note The data samples containing outliers are replaced with NaNs.

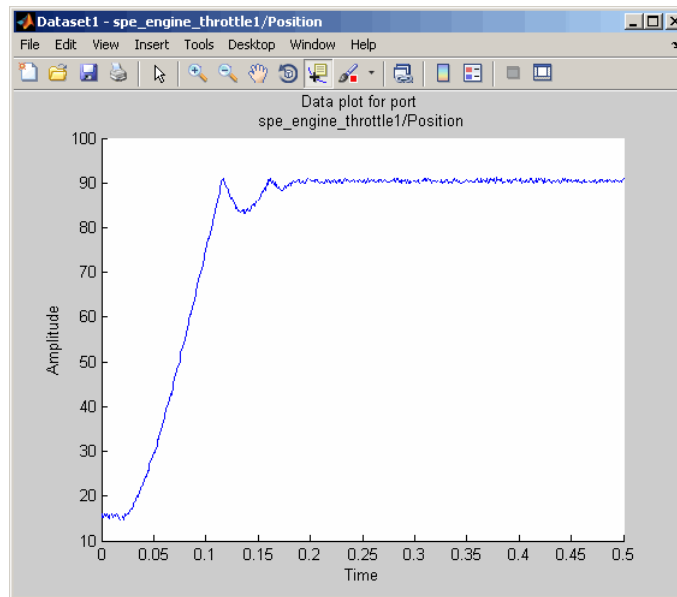
3 In the **Write results to** area, verify that the **existing dataset** **Dataset1** option remains selected.

4 Click **Add**.

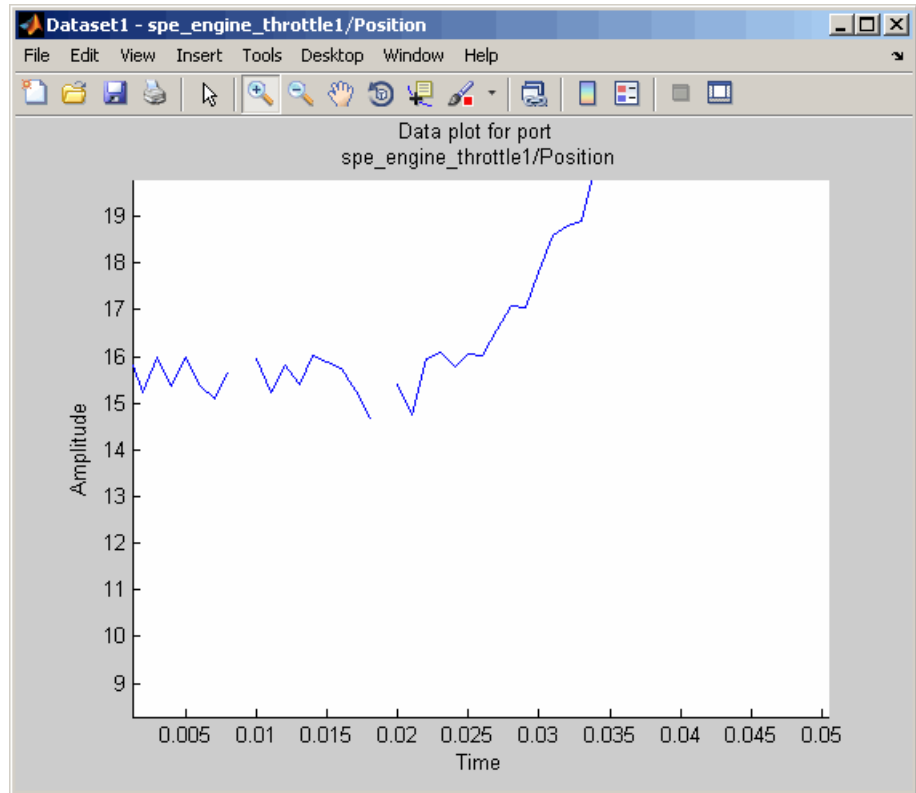
The Update table data dialog box appears. Click **Yes** to overwrite the `position1(:,1)*` data set with the modified data.

5 To plot the data, select the `position1(:,1)*` cell in the **Output Data** tab of the **Dataset1** node, and click **Plot Data**.

The spikes, which indicate outliers, no longer appear on the time plot, as shown in the next figure.



The missing data samples, represented by NaNs, appear as gaps on the time plot. To see an example, zoom in to the bottom-left corner of the plot. As shown in the next figure, the data values corresponding to $t = 0.009$ s and $t = 0.019$ s are missing.



Filtering Data

- “Filtering Output Data” on page 2-30
- “Filtering Input Data” on page 2-33

Filtering Output Data

In this portion of the tutorial, you filter the noise, and remove any periodic trends from the output data. To avoid relative phase shift between the I/O data, you must also apply the same filter to the input data.

You must have already removed outliers from the output data, as described in “Removing Outliers” on page 2-25. If you have not done this preparation, [click here](#).

- 1 In the Control and Estimation Tools Manager window, select the `position1(:,1)*` cell in the **Output Data** tab of the **Dataset1** node, and click **Pre-process**.

This action updates the Data Preprocessing Tool GUI with the selected data `position1(:,1)*`.

2 In the **Detrend/Filtering** tab:

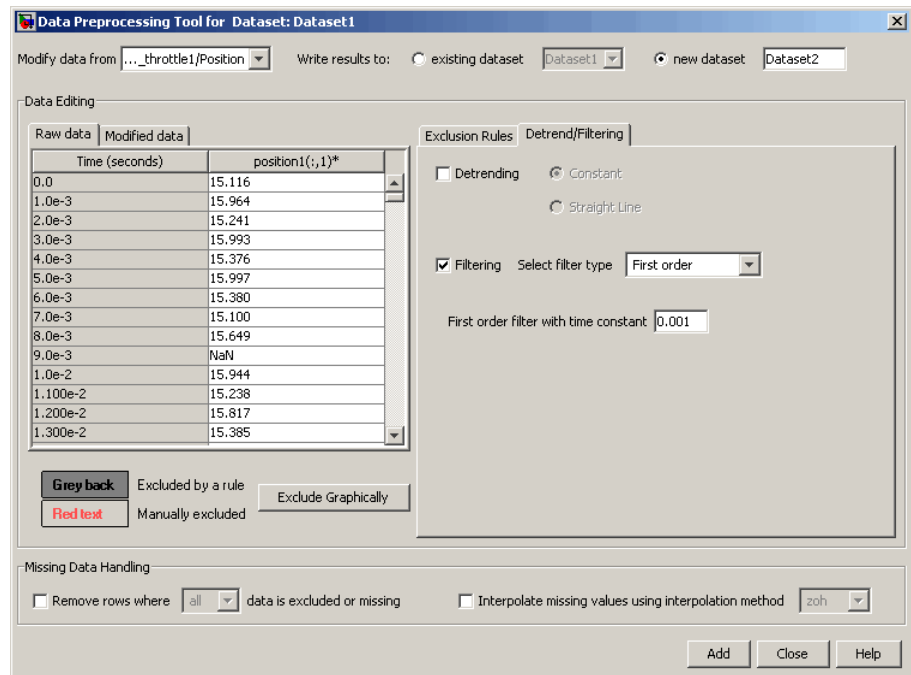
- a** Select the **Filtering** check box.

By default, **First order** is selected as the filter type. To learn more about the filters, see “Filtering Data”.

- b** Specify **0.001** in the **First order filter with time constant** field.

This field specifies the time constant for the first-order filter.

Tip For calculating the time constant, you can visually inspect the time plot to determine the frequency components.



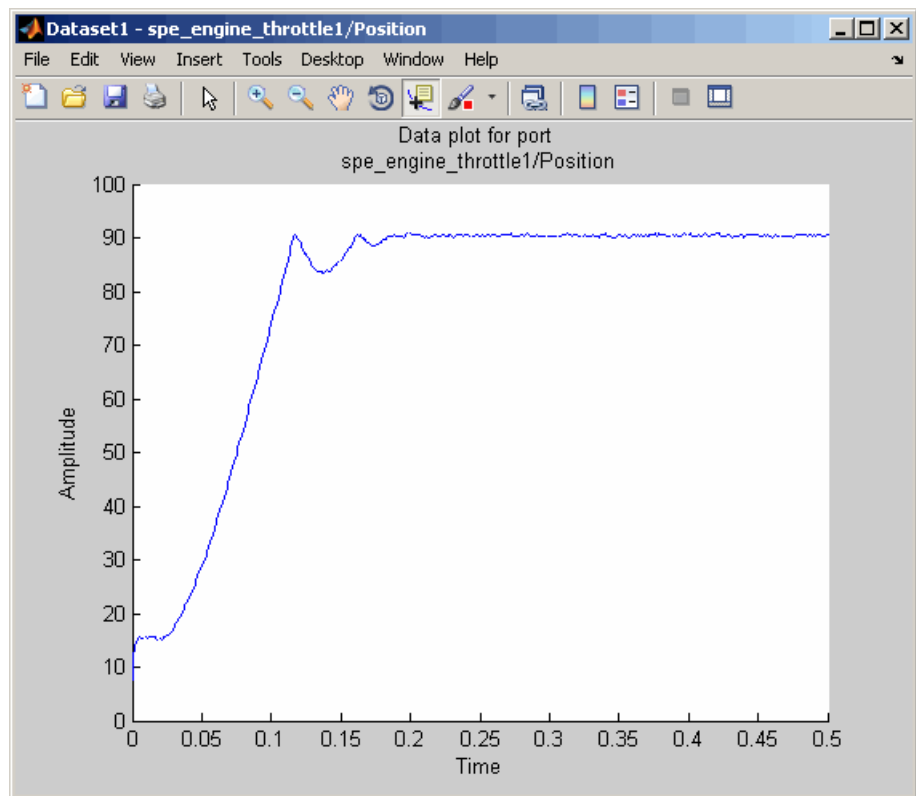
- 3** In the **Write results to** area, verify that the **existing dataset** **Dataset1** option remains selected.

4 Click **Add**.

The Update table data dialog box appears. Click **Yes** to overwrite the `position1(:,1)*` data set with the modified data.

5 To plot the data, select the `position1(:,1)*` cell in the **Output Data** tab of the **Dataset1** node, and click **Plot Data**.

The noise is filtered and the output data appears smooth, as shown in the next figure.



Filtering Input Data

After you filter the output data, as described in “Filtering Output Data” on page 2-30, you must also filter the input data with the same filter.

- 1 In the Control and Estimation Tools Manager window, select the `input1(:,1)*` cell in the **Input Data** tab of the **Dataset1** node, and click **Pre-process**.

This action updates the Data Preprocessing Tool GUI with the selected data `input1(:,1)*`.

- 2 In the **Detrend/Filtering** tab:

- a Select the **Filtering** check box.

By default, **First order** is selected as the filter type.

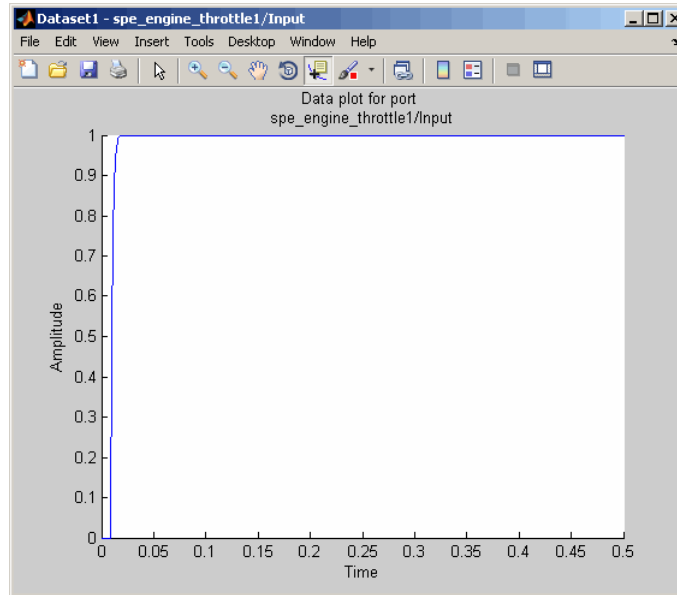
- b Specify 0.001 in the **First order filter with time constant** field.

- 3 In the **Write results to** area, verify that the **existing dataset** option remains selected.

- 4 Click **Add**.

The Update table data dialog box appears. Click **Yes** to overwrite the `input1(:,1)*` data set with the modified data.

- 5 To plot the data, select the `input1(:,1)*` cell in the **Input Data** tab of the **Dataset1** node, and click **Plot Data**.



Interpolating Missing Data

In this portion of the tutorial, you interpolate the output data to compute the missing values created when removing outliers, as described in “Removing Outliers” on page 2-25. The interpolation operation uses the known data values to compute the missing data values.

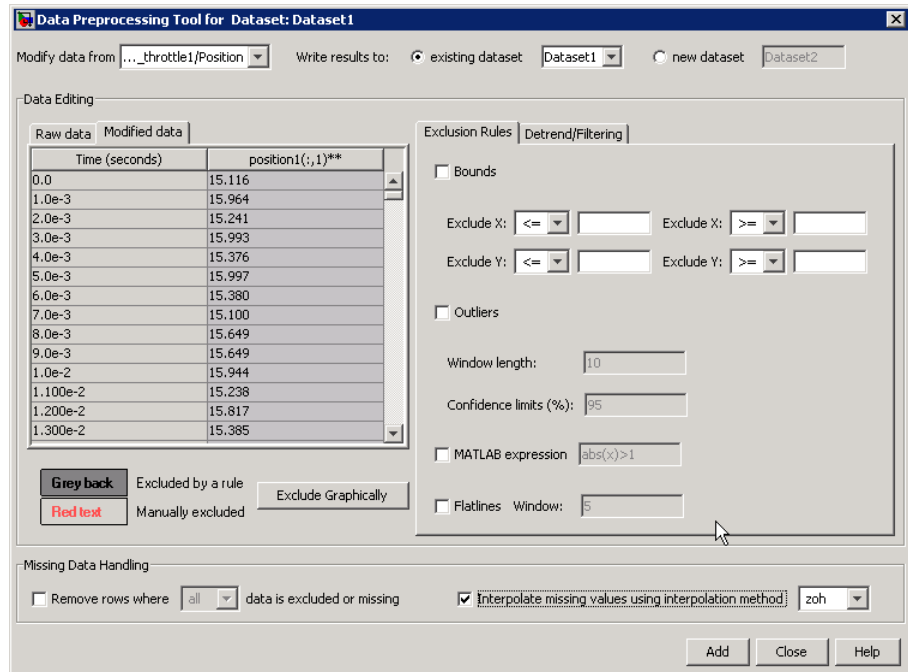
You must have already filtered the noise, as described in “Filtering Data” on page 2-29. If you have not already done this preparation, click [here](#).

- 1 In the Control and Estimation Tools Manager GUI, select the `position1(:,1)*` cell in the **Output Data** tab of the **Dataset1** node, and click **Pre-process**.

This action updates the Data Preprocessing Tool GUI with the selected data `position1(:,1)*`.

- 2** In the **Missing Data Handling** area, select the **Interpolate missing values using interpolation method** check box.

By default, `zoh` is selected as the interpolation method. This method fills the missing data sample with the data value immediately preceding it.



Tip You can view the interpolated data samples in the **Modified data** tab.

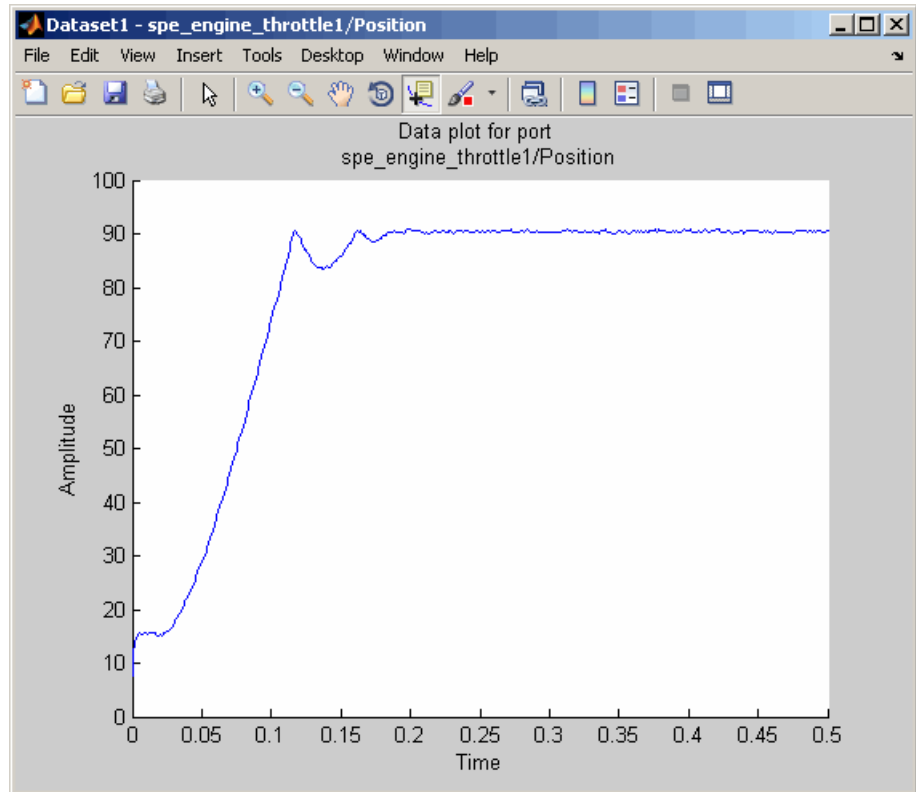
- 3** In the **Write results to** area, verify that the **existing dataset** `Dataset1` option remains selected.

- 4** Click **Add**.

The Update table data dialog box appears. Click **Yes** to overwrite the `position1(:,1)*` data set with the modified data.

- 5 To plot the data, select the `position1(:,1)*` cell in the **Output Data** tab of the **Dataset1** node, and click **Plot Data**.

The new estimation data, prepared by removing outliers, filtering noise, and interpolating missing data, is shown the next figure.



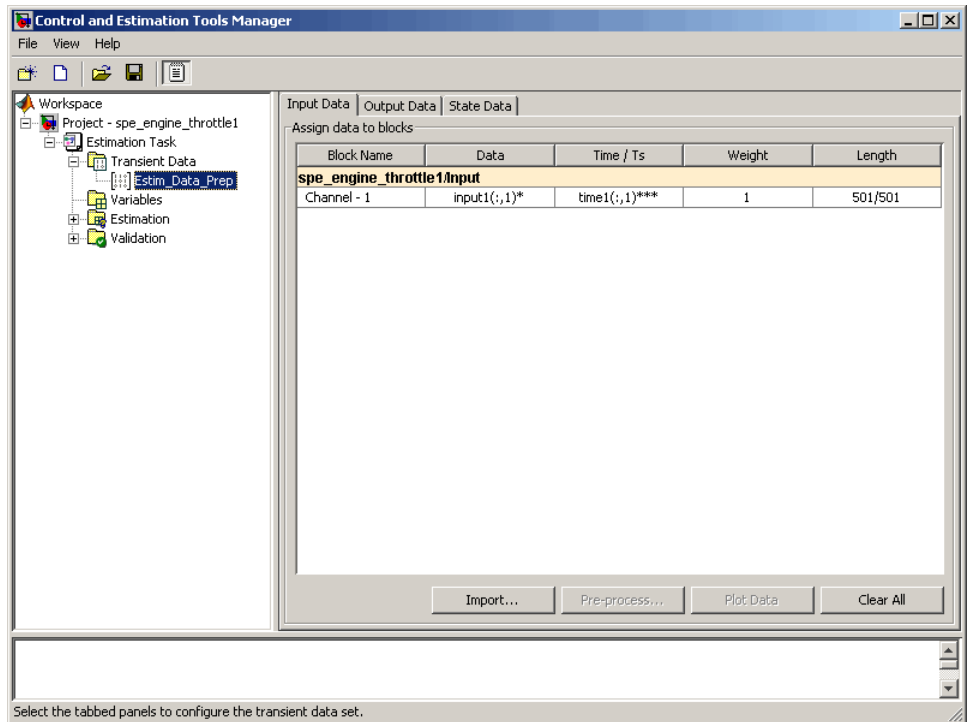
Saving the Project

After you prepare the data, you can delete the data in the **New Data** node, rename the prepared data, and save the session. To skip the data preparation steps, click [here](#).

- 1 In the Control and Estimation Tools Manger GUI, select the **New Data** node under the **Transient Data** node.

- 2 Right-click the **New Data** node, and select **Delete**.
- 3 Select the **Dataset1** node under the **Transient Data** node.
- 4 Right-click the **Dataset1** node, and select **Rename**. Specify `Estim_Data_Prep` as the name of the new estimation data set.

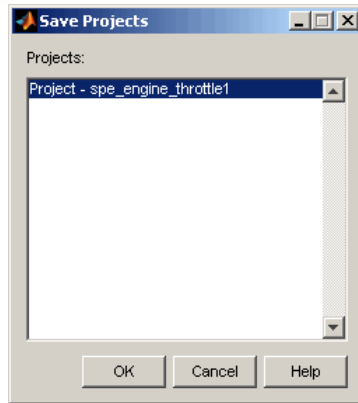
The Control and Estimation Tools Manager GUI resembles the next figure.



- 5 Save the Control and Estimation Tools Manager project:
 - a In the Control and Estimation Tools Manager GUI, select **File > Save**.

This action opens the Save Projects dialog box.

- b In the Save Projects dialog box, click **OK**.



- c In the Save Projects window, specify `spe_engine_throttle1p.mat` in the **File name** field, and click **Save**.

The action saves the project as a MAT-file.

To learn how to estimate parameters from this data, see “Estimate Parameters from Measured Data (GUI)” on page 2-39.

Estimate Parameters from Measured Data (GUI)

In this section...

“About This Tutorial” on page 2-39

“Estimating Model Parameters Using Default Estimation Settings” on page 2-43

“Improving Estimation Results Using Parameter Bounds” on page 2-56

“Validating Estimated Model Parameters” on page 2-61

About This Tutorial

- “Objectives” on page 2-39
- “About the Model” on page 2-40

Objectives

In this tutorial, you learn how to estimate parameters of a single-input single-output (SISO) Simulink model from measured input and output (I/O) data.

Note Simulink Design Optimization software estimates parameters from real, time-domain data only.

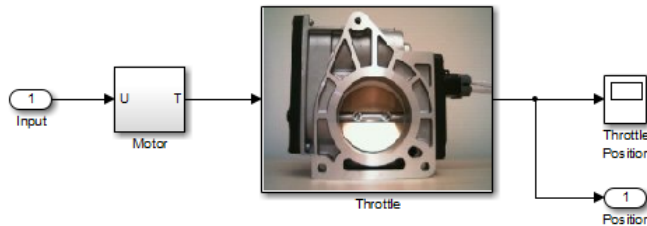
You learn to perform the following tasks using the GUI:

- Load a saved project containing data.
- Estimate model parameters using default settings.
- Validate the model, and refine the estimation results.

About the Model

In this tutorial, you use the `spe_engine_throttle1` Simulink model. This model represents an engine throttle system, as shown in the next figure.

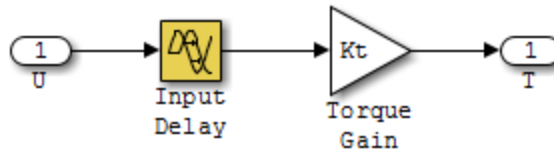
Engine Throttle Model



The throttle system controls the flow of air and fuel mixture to the engine cylinders. The throttle body contains a butterfly valve which opens when a driver presses the accelerator pedal. Opening this valve increases the amount of fuel mixture entering the cylinders, which increases the engine speed. A DC motor controls the opening angle of the butterfly valve in the throttle system. The models for these components are described in “Motor Subsystem” on page 2-41, and “Throttle Subsystem” on page 2-41.

The input to the throttle system is the motor current (in amperes), and the output is the angular position of the butterfly valve (in degrees).

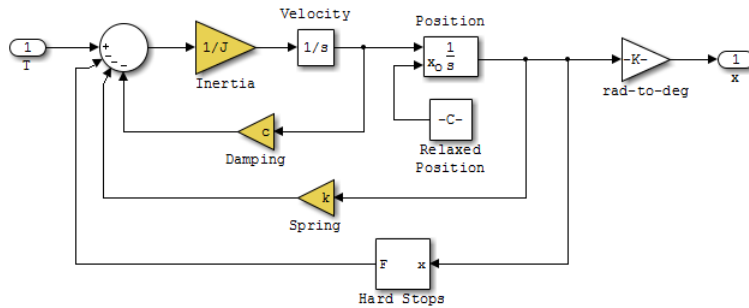
Motor Subsystem. The Motor subsystem contains the DC motor model. To open the model, double-click the corresponding block.



The following table describes the variables, parameters, equation, input, and output of the Motor subsystem.

Variables	U is the input current to the motor. T is the torque applied by the motor.
Parameters	K_t is the torque gain of the motor, represented by Kt in the model. t_d is the input time delay of the motor, represented by <code>input_delay</code> in the model.
Equation	The torque applied by the motor is described in the following equation: $T(t) = K_t U(t - t_d)$ where t is time.
Input	U
Output	T

Throttle Subsystem. The Throttle subsystem contains the butterfly valve model. To open the model, right-click the corresponding block, and select **Mask > Look Under Mask**.



The Hard Stops block models the valve angular position limit of 15° to 90° .

The following table describes the variables, parameters, states, differential equations, inputs, and outputs of the Throttle subsystem.

Variables	<p>T is the torque applied by the DC motor.</p> <p>θ is the angular position of the valve, represented by x in the model.</p> <p>$T_{hardstop}$ is the torque applied by the hard stop.</p>
Parameters	<p>J is the inertia.</p> <p>c is the viscous friction.</p> <p>k is the spring constant.</p>
States	<p>θ is the angular position.</p> <p>$\dot{\theta}$ is the angular velocity.</p>

Equations	<p>The mathematical system for the butterfly valve is described in the following equation:</p> $J\ddot{\theta} + c\dot{\theta} + k\theta = T + T_{hardstop}$ <p>where $15^\circ \leq \theta \leq 90^\circ$, with initial conditions $\theta_0 = 15^\circ$, and $\dot{\theta}_0 = 0$. The torque applied by the Hard Stops block is described in the following equation:</p> $T_{hardstop} = \left\{ \begin{array}{l} 0, \quad 15^\circ \leq \theta \leq 90^\circ \\ K(90^\circ - \theta), \theta > 90^\circ \\ K(15^\circ - \theta), \theta < 15^\circ \end{array} \right\}$ <p>where K is the gain of the Hard Stops block.</p>
Input	T
Output	θ

Estimating Model Parameters Using Default Estimation Settings

- “Overview of the Estimation Process” on page 2-44
- “Specifying Parameters and Estimation Data” on page 2-45
- “Validating Model Parameters” on page 2-49

Overview of the Estimation Process

Simulink Design Optimization software uses optimization techniques to estimate model parameters. In each optimization iteration, the model is simulated with the current parameter values. The error between the simulated and measured output is computed and minimized. The estimation is complete when the optimization method finds a local minimum.

You perform the following tasks to estimate the model parameters:

- “Specifying Parameters and Estimation Data” on page 2-45
- “Validating Model Parameters” on page 2-49

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

Specifying Parameters and Estimation Data

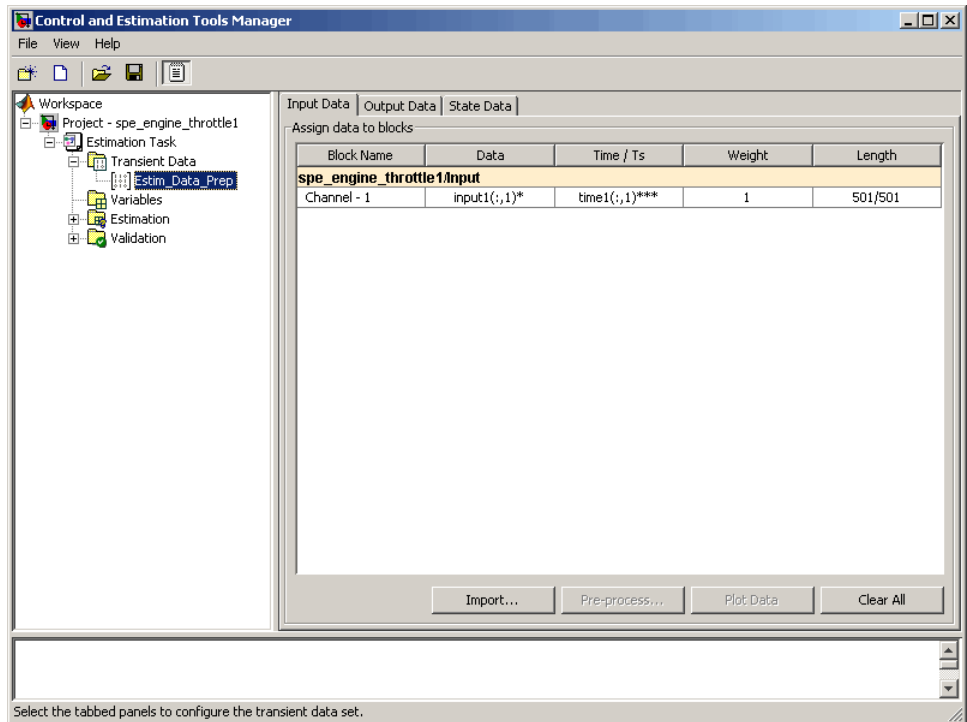
To specify parameters and estimation data:

- 1 Load the preconfigured project `spe_engine_throttle1p.mat`.

This MAT-file contains the estimation data. To learn more about the data and how to prepare it, see “Prepare Data for Parameter Estimation (GUI)” on page 2-3.

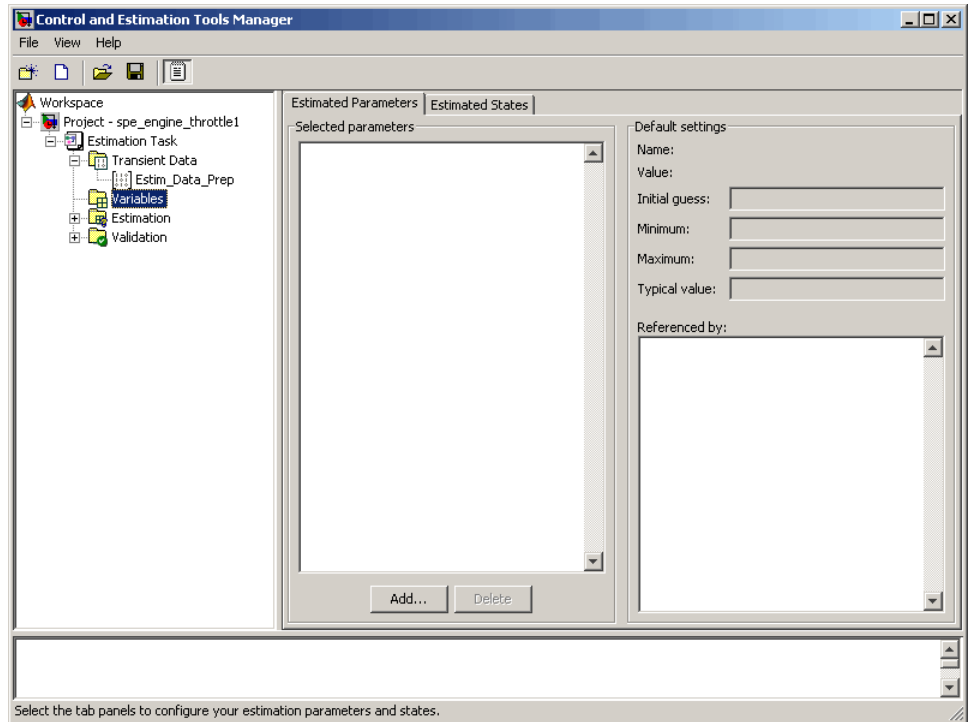
To load the preconfigured project:

The Control and Estimation Tools Manager GUI opens as shown in the next figure.



2 Specify parameters for estimation.

- a Select the **Variables** node under the **Estimation Task** node.

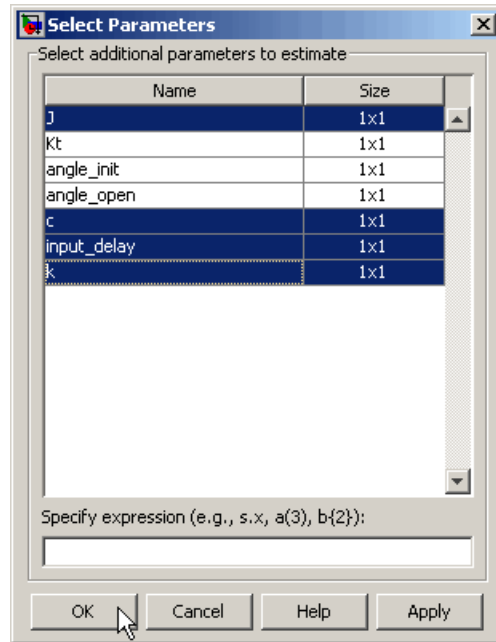


- b Click **Add**.

This action opens the Select Parameters dialog box, which shows the model parameters for the Simulink model.

- c Select the parameters **J**, **c**, **input_delay**, and **k** by pressing the **Ctrl** key while clicking each name, and then click **OK**.

This action adds the selected parameters to the **Estimated Parameters** tab.



Tip When estimating a large number of parameters, you can select a subset of parameters to estimate. For more information, see the example [Inverted Pendulum Parameter Estimation](#).

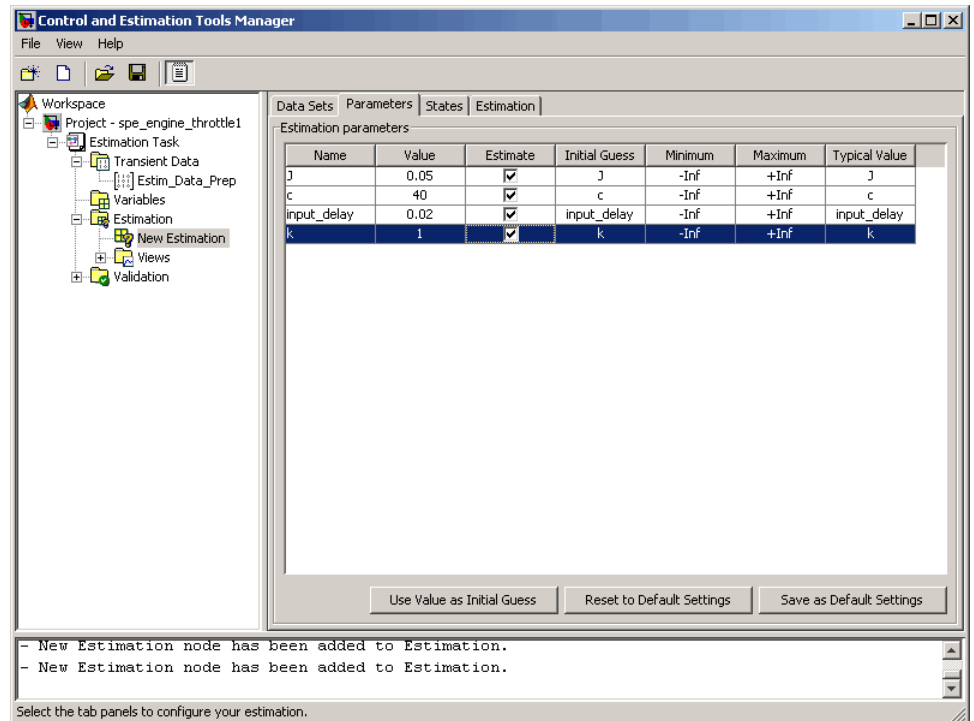
- d** Select the **Estimation** node, and click **New**.

This action adds a **New Estimation** node under the **Estimation** node.

- e** Select the **New Estimation** node.

- f In the **Parameters** tab, select the **Estimate** check box for all the parameters.

This action specifies the selected parameters for estimation.



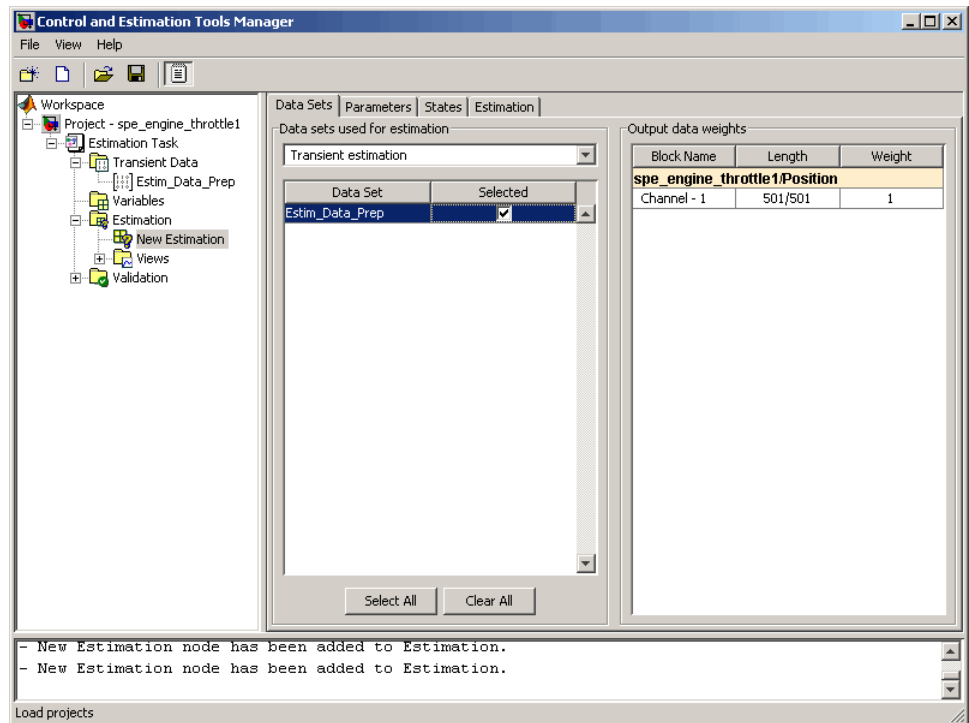
The **Parameters** tab, as shown in the previous figure, displays the following information for each parameter:

- **Value:** Current parameter value. By default, it is equal to the value specified in the Simulink model.

During estimation, the optimization method might change a parameter value to minimize the error between the measured and simulated output.

- **Initial Guess:** Initial parameter value. By default, it is equal to the value in the **Value** field and is used at the start of estimation.

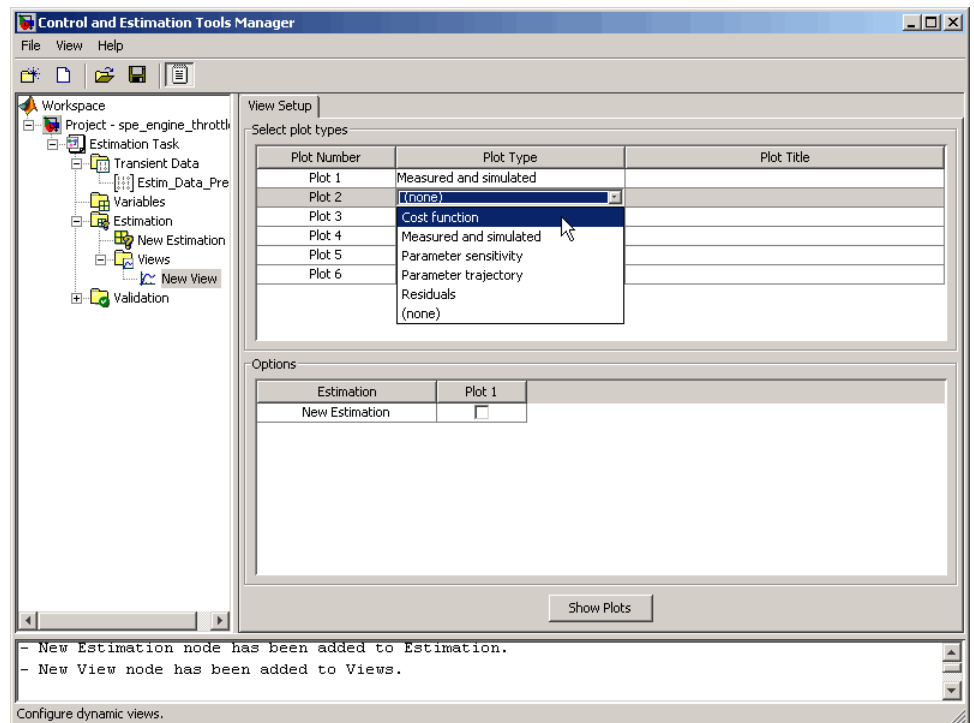
- **Minimum** and **Maximum**: Bounds on the parameter value. By default, they are set to $-\text{Inf}$, and $+\text{Inf}$, respectively. This means that the optimization method searches for a solution in the range $[-\text{Inf}, +\text{Inf}]$.
 - **Typical Value**: Order of magnitude of the parameter value. By default, it is equal to the value in the **Initial Guess** field.
- 3 Specify data for estimation by checking the **Selected** check box for **Estim_Data_Prep** in the **Data Sets** tab.



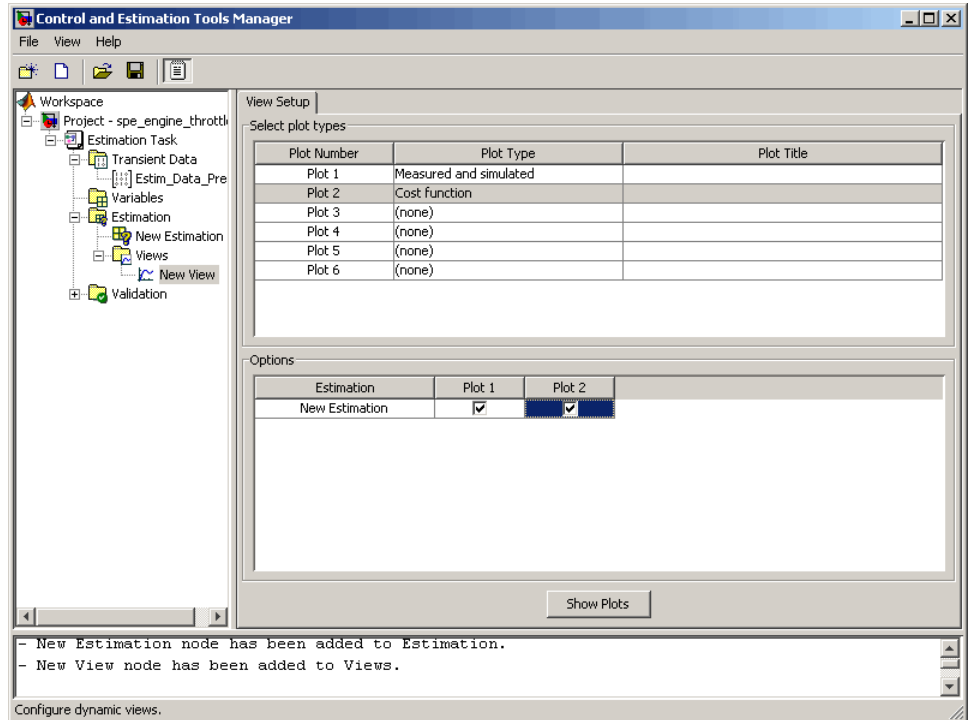
Validating Model Parameters

After you specify parameters and data for estimation, as described in “Specifying Parameters and Estimation Data” on page 2-45, estimate the parameters and analyze the results to determine if the model needs to be refined.

- 1 Create plots to view the estimation results.
 - a Select the **Views** node under the **Estimation** node, and click **New**.
This action creates a **New View** node under the **Views** node.
 - b Select the **New View** node.
 - c In the **View Setup** tab, select the following plots from the drop-down list in the **Plot Type** column:
 - For **Plot 1**, select **Measured and simulated**.
 - For **Plot 2**, select **Cost function**.



- d In the **Options** area, select the **Plot 1** and **Plot 2** check boxes, and click **Show Plots**.

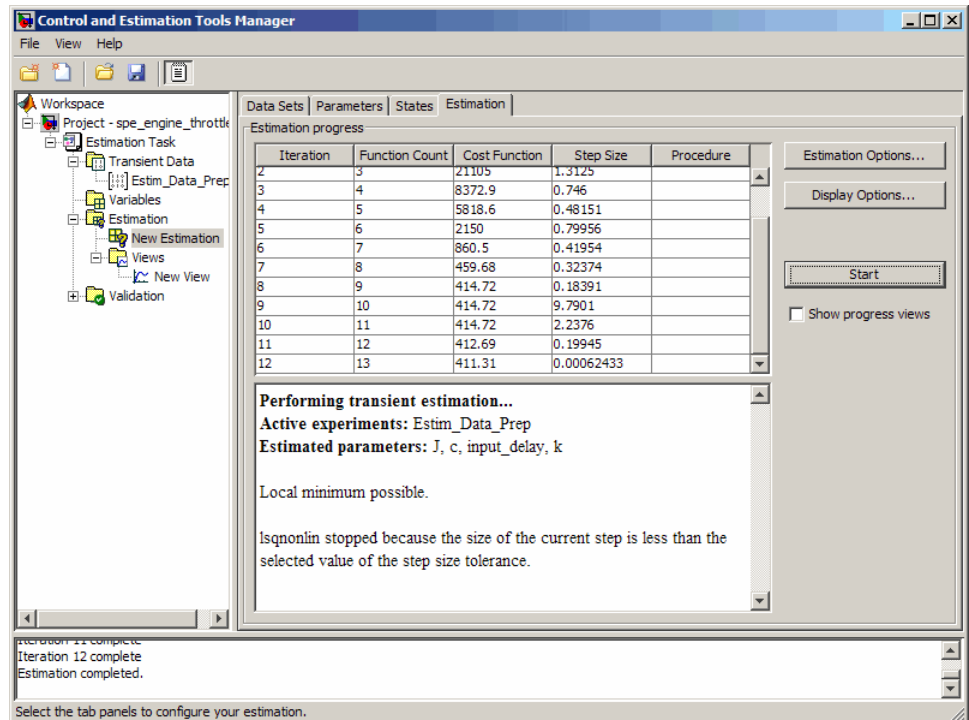


Clicking **Show Plots** opens the following figures:

- **New View - Plot 1 (Measured and simulated)**: Displays the measured output data `Estim_Prep_Data`. During estimation, this plot updates to display the simulated response at each iteration.
- **New View - Plot 2 (Cost function)**: Displays the error between the measured and simulated output, computed at each iteration. By default, the error is computed using *sum-of-squared-errors*, which is a least-squares method.

- 2 Estimate the parameters by selecting the **Estimation** tab of the **New Estimation** node, and clicking **Start**.

The **Estimation** tab updates at each iteration, and provides information about the estimation progress. When the estimation completes, the **Estimation** tab resembles the next figure.



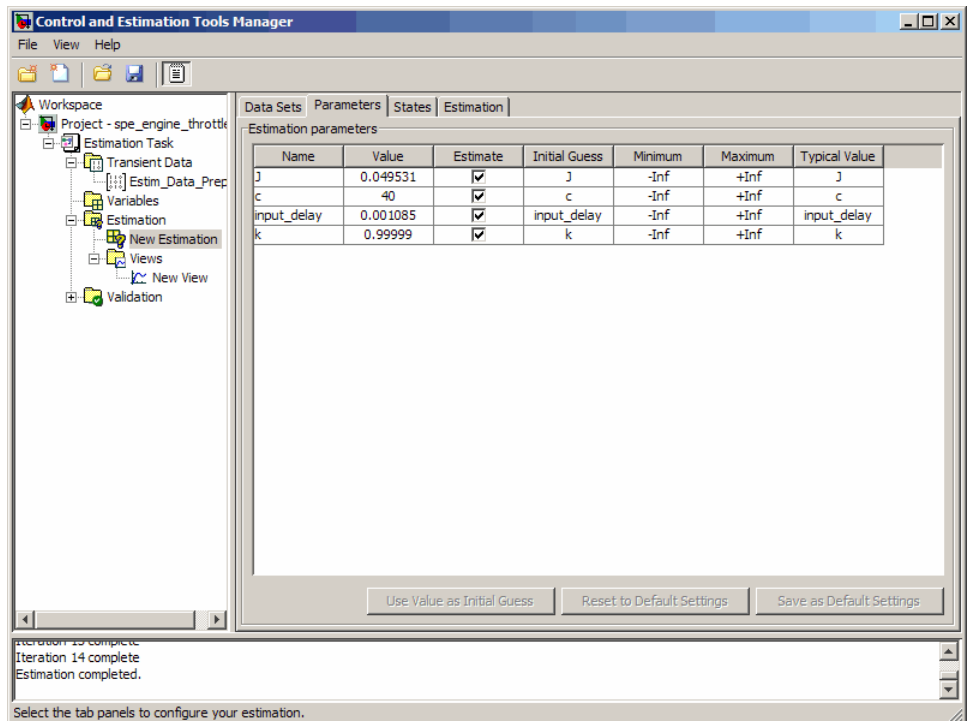
Note The results of the optimization may differ slightly because of different numerical precision across platforms.

The table displays the following information for each iteration:

- **Cost Function:** Error between the simulated and measured output.

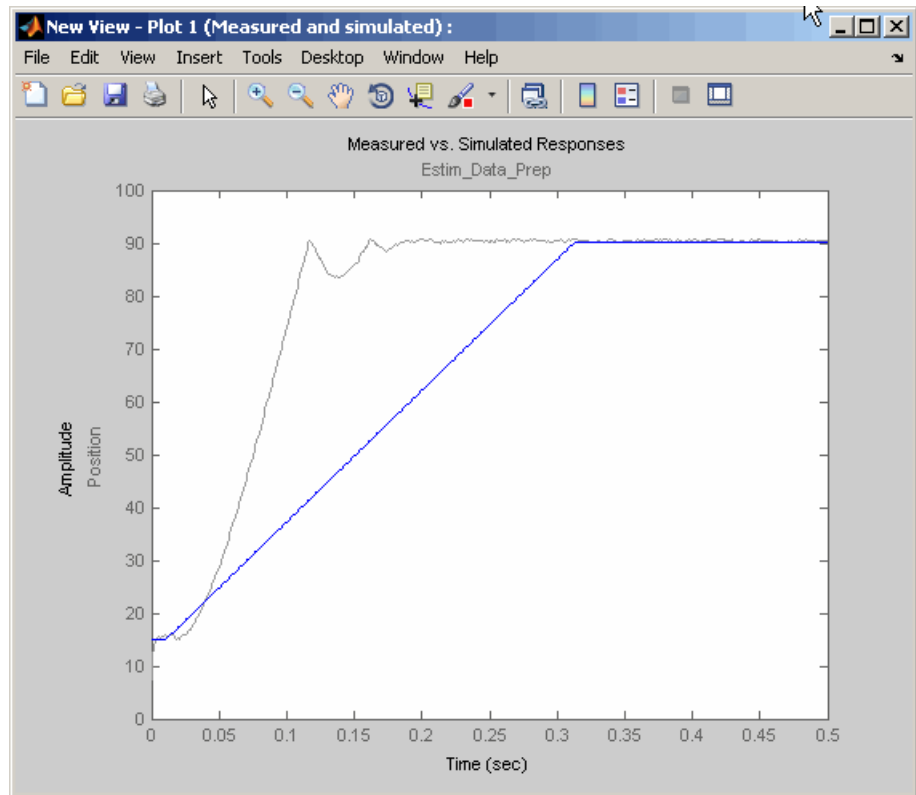
During estimation, the default optimization method `NonLinear least squares`, `lsqnonlin`, minimizes the cost function by changing the parameter values. As shown in the previous figure, the cost function decreases by only 27% from $2.69e5$ to $1.95e5$.

- **Step Size:** Displacement of the optimization method in the current search direction when minimizing the cost function. A final value close to zero indicates that the method has found a local minimum. As shown in the previous figure, the final step size in the last iteration is $5.96e-7$.
- 3 View the estimated parameter values in the **Value** column of the **Parameters** tab.



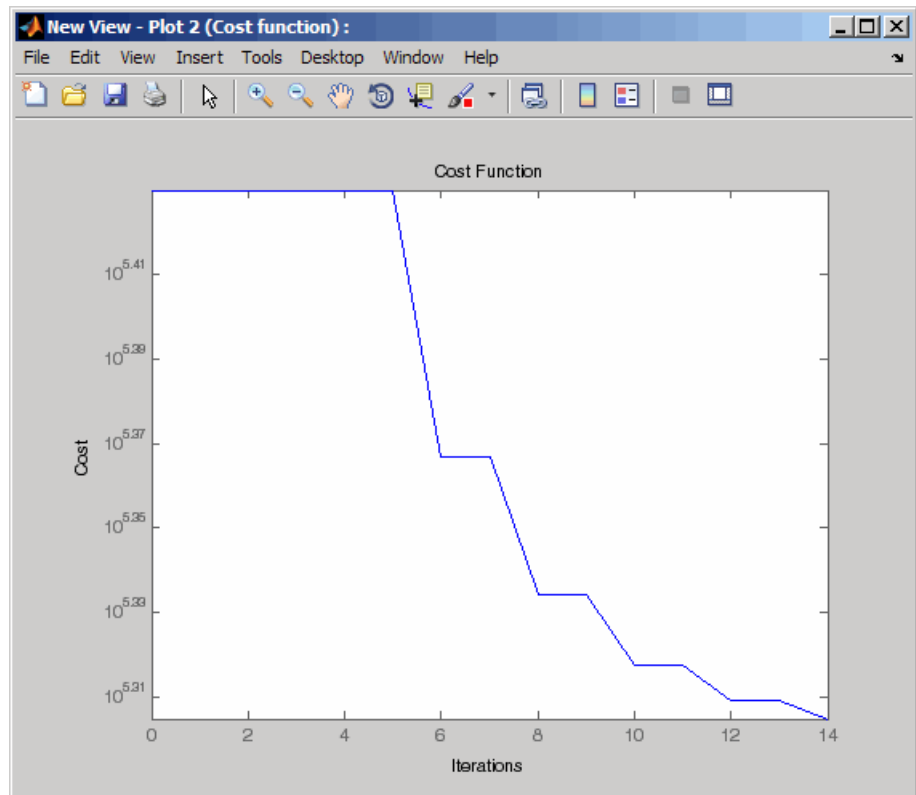
- 4 Examine the simulated response plot to see how well the simulated output matches the measured output.

The simulated response with the estimated parameters is shown in blue on the New View - Plot 1 (Measured and simulated) plot, and significantly differs from the measured data. This difference indicates that the estimated parameters are not accurate.



- 5 Examine the cost function plot to see how the cost function changes during the estimation.

The cost function values shown in step 2 are plotted on the New View - Plot 2 (Cost function) plot. The cost function decreases by only 27% from its initial value, and in conjunction with the measured and simulated output plot, indicates that the estimated parameters are not accurate.



To improve the estimation results, you apply bounds on the parameter values, as described in “Improving Estimation Results Using Parameter Bounds” on page 2-56.

Improving Estimation Results Using Parameter Bounds

- “Strategy for Improving the Estimation Results” on page 2-56
- “How to Specify Parameter Bounds” on page 2-56

Strategy for Improving the Estimation Results

Analyzing the Measured and Simulated and Cost function plots, as described in “Validating Model Parameters” on page 2-49, you see that the model parameters estimated using the default estimation settings are not accurate. You must run additional estimations to improve the accuracy of the model. There are several techniques to improve the accuracy of the estimated parameters. For more information, see “How to Specify Estimation Options in the GUI”.

In this portion of the tutorial, you improve the results by specifying bounds on parameter values. This technique restricts the region in which the optimization method searches for a local minima.

Based on physical insight, you know the following characteristics of the engine throttle system:

- All the parameter values are positive.
- Maximum time delay of the system, represented by `input_delay`, is 0.1 s.

Therefore, you specify 0 as the minimum value for all parameters, and 0.1 as the maximum value of `input_delay`.

After you estimate the parameters, analyze the results using the Measured and Simulated and Cost function plots.

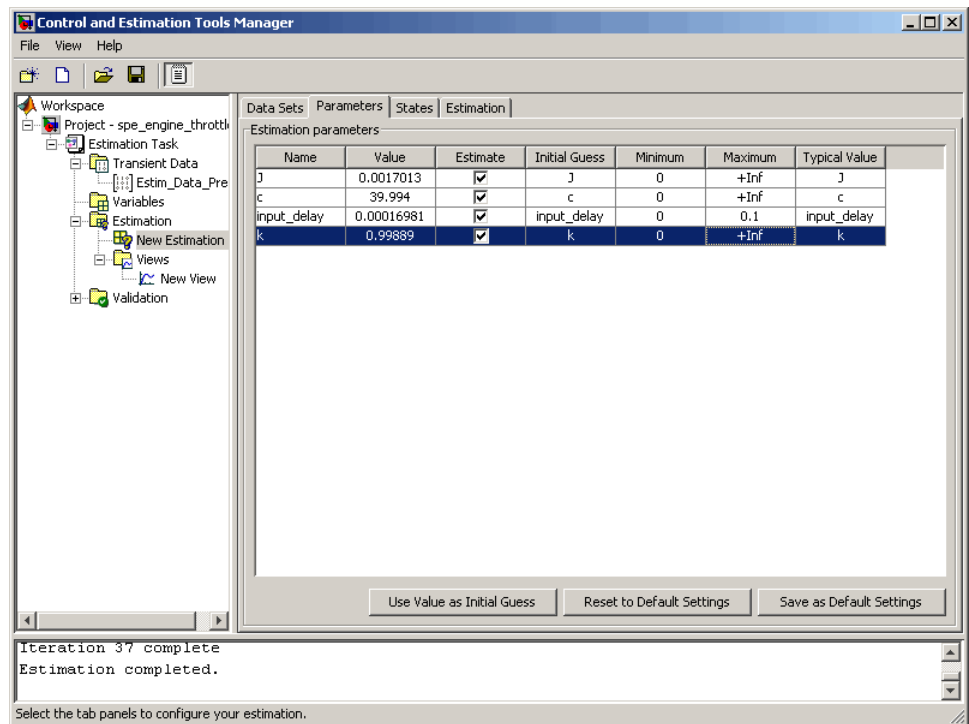
How to Specify Parameter Bounds

You must have already estimated the parameters using default settings, as described in “Estimating Model Parameters Using Default Estimation Settings” on page 2-43.

To improve the estimation results by specifying parameter bounds:

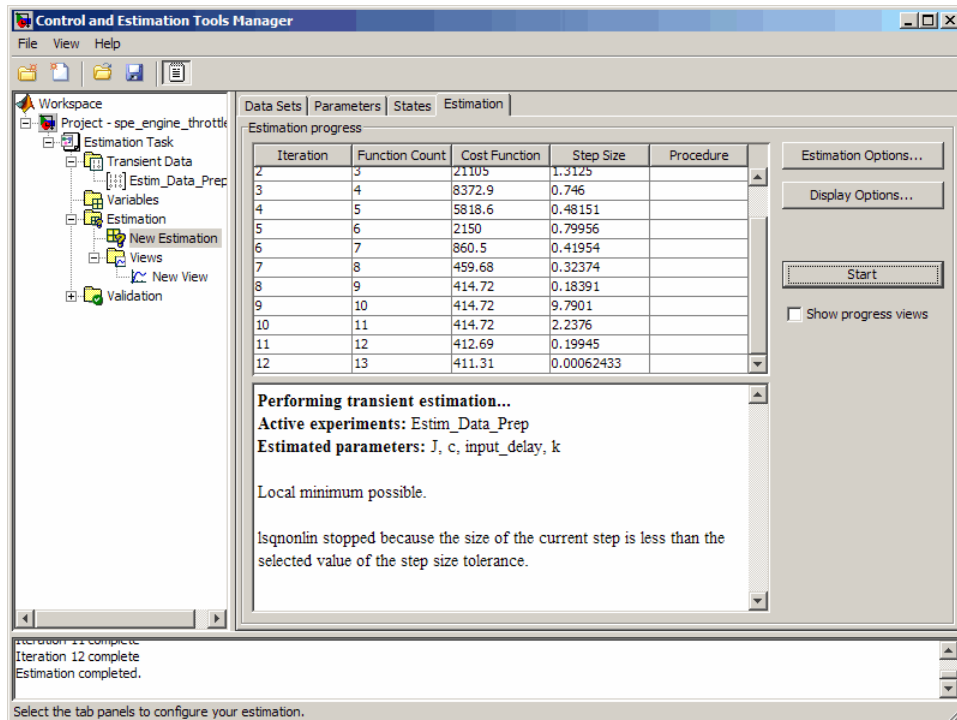
- 1 Select the **Parameters** tab of the **New Estimation** node.
- 2 Specify the minimum value for each parameter by double-clicking the corresponding **Minimum** cell, and replacing - Inf with 0.
- 3 Specify the maximum value for `input_delay` by double-clicking the corresponding **Maximum** cell and replacing +Inf with 0.1.

The **Parameters** tab resembles the next figure.



4 Select the **Estimation** tab, and click **Start**.

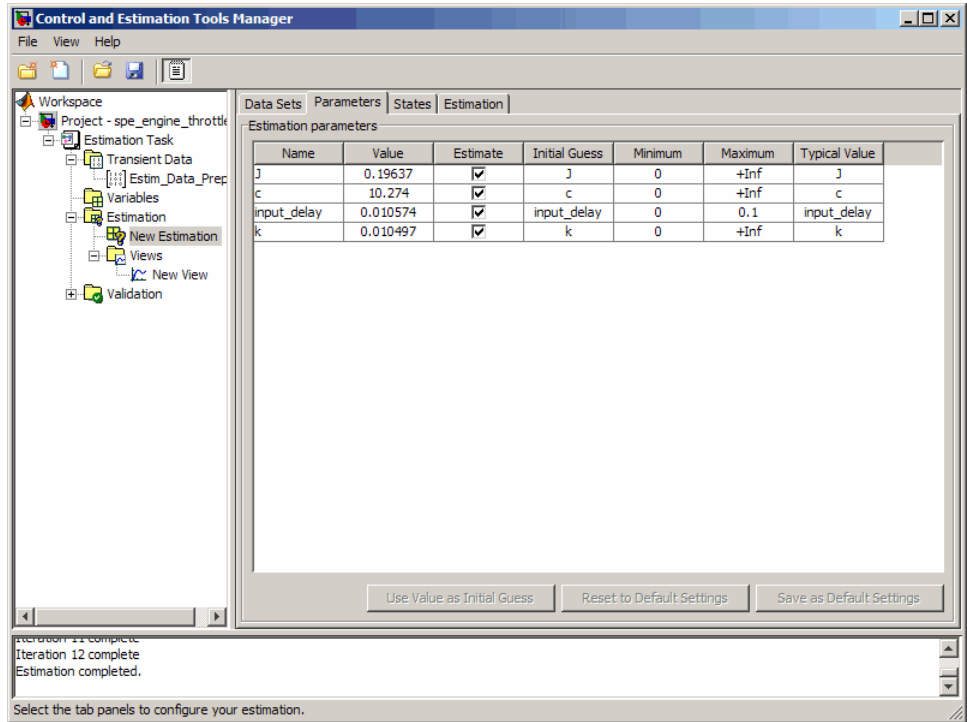
The **Estimation** tab updates at each iteration, and provides information about the estimation progress. When the estimation completes, the **Estimation** tab resembles the next figure.



Note The results of the optimization may differ slightly because of different numerical precision across platforms.

The cost function decreases by 99% from $1.95e5$ to 239.94. The final step size $8.75e-7$ is close to 0, which indicates that the optimization method has found a local minimum.

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

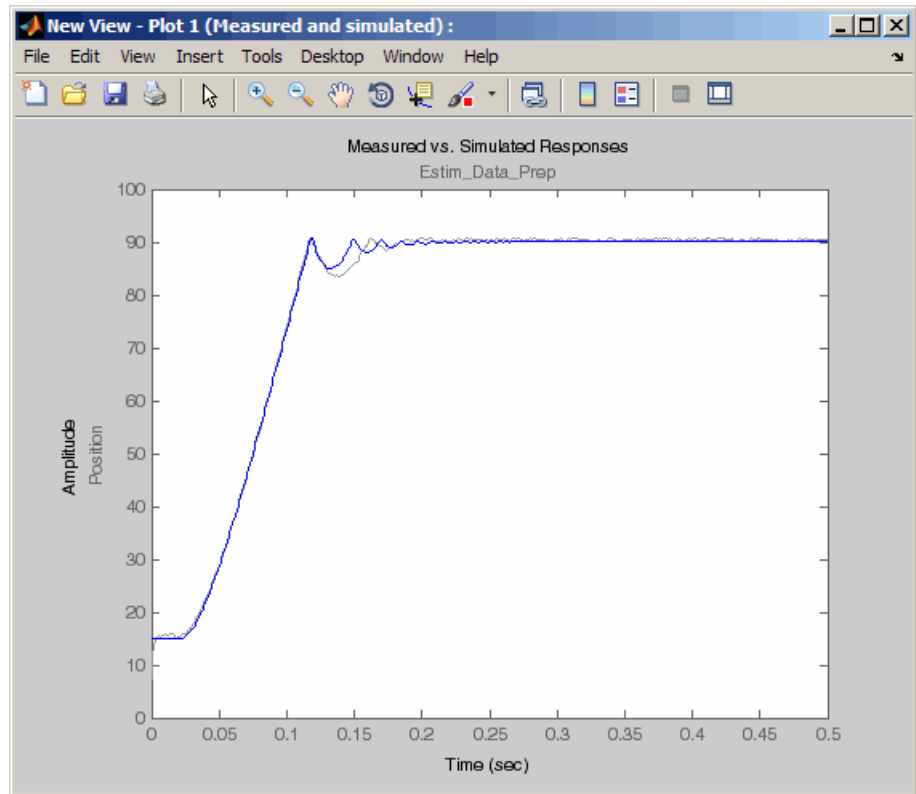


The screenshot shows the 'Control and Estimation Tools Manager' window. The 'Parameters' tab is active, displaying a table of 'Estimation parameters'. The table has columns for Name, Value, Estimate, Initial Guess, Minimum, Maximum, and Typical Value. The parameters listed are J, c, input_delay, and k. The 'Value' column shows the estimated values: 0.19637 for J, 10.274 for c, 0.010574 for input_delay, and 0.010497 for k. The 'Estimate' column has checkmarks for all parameters. The 'Initial Guess' column shows 'J', 'c', 'input_delay', and 'k'. The 'Minimum' column shows '0' for all. The 'Maximum' column shows '+Inf' for all. The 'Typical Value' column shows 'J', 'c', 'input_delay', and 'k'. Below the table are three buttons: 'Use Value as Initial Guess', 'Reset to Default Settings', and 'Save as Default Settings'. The status bar at the bottom indicates 'Iteration 12 complete' and 'Estimation completed.'.

Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Value
J	0.19637	<input checked="" type="checkbox"/>	J	0	+Inf	J
c	10.274	<input checked="" type="checkbox"/>	c	0	+Inf	c
input_delay	0.010574	<input checked="" type="checkbox"/>	input_delay	0	0.1	input_delay
k	0.010497	<input checked="" type="checkbox"/>	k	0	+Inf	k

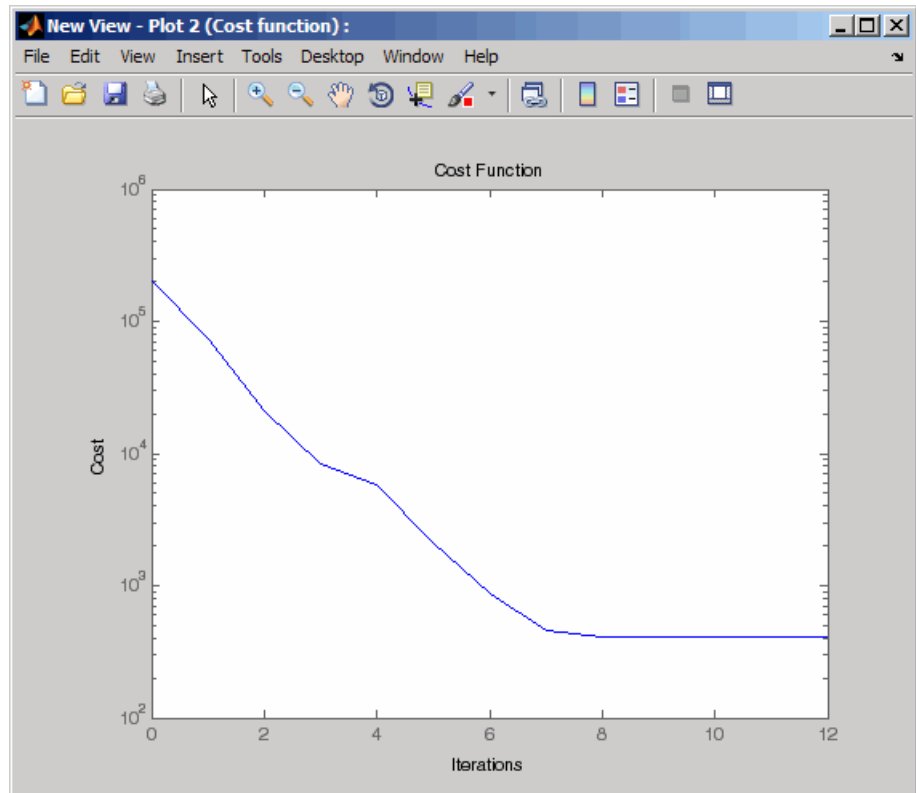
- 6 Examine the simulated response plot to see how well the simulated output matches the measured output.

The simulated response plot, shown in blue on the New View - Plot 1 (Measured and simulated) plot, is overlaid on the measured output data. The simulated output closely matches the measured data.



- 7 Examine the cost function plot to see how the cost function changes during the estimation.

The cost function values shown in step 4 are plotted on the New View - Plot 2 (Cost function) plot. The cost function also decreases by 99% from its initial value, and in conjunction with the measured and simulated response plot indicates a good fit of the simulated response with the measured data.



Validating Estimated Model Parameters

After you estimate the model parameters, as described in “Improving Estimation Results Using Parameter Bounds” on page 2-56, validate the model using another data set (*validation data*). A good match between the

simulated response and the validation data indicates that you have not overfitted the model.

To validate the estimated parameters using a validation data set:

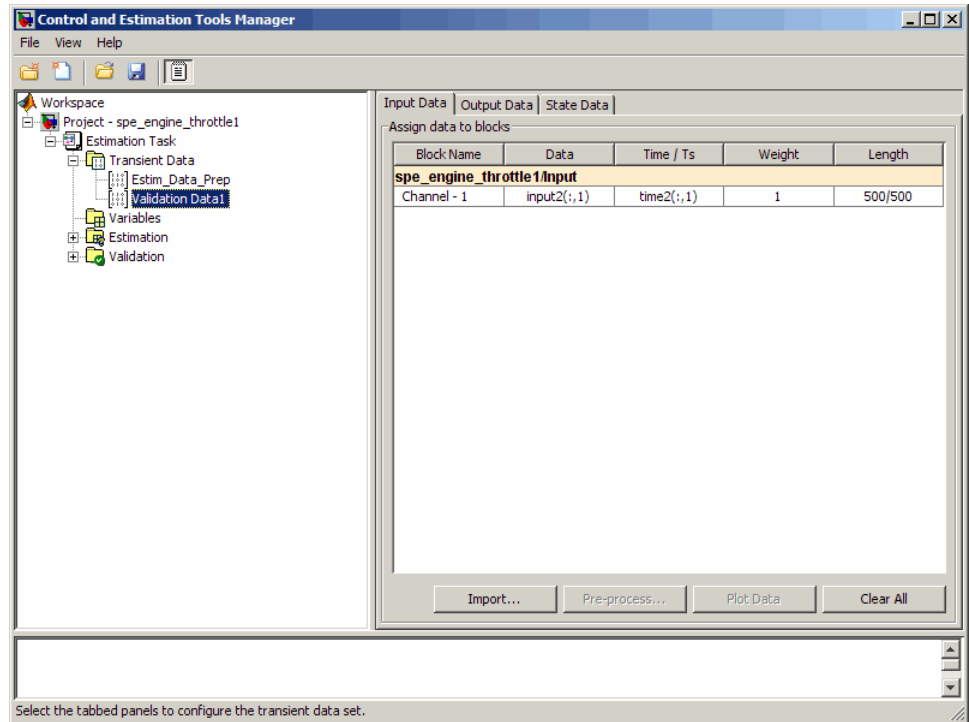
- 1 Load the preconfigured project `spe_engine_throttle1_importValData1.mat`.

This MAT-file contains a validation data set already imported into the GUI, and the estimated parameters as described in “Improving Estimation Results Using Parameter Bounds” on page 2-56.

Tip To learn how to import data and time vector into the Control and Estimation Tools Manager GUI, see “Importing Data into the GUI” on page 2-6.

To load the preconfigured project:

The Control and Estimation Tools Manager GUI now has a node that contains the validation data, as shown in the next figure.



The validation data contains the input data, output data and time vector in the MATLAB variables `input2`, `position2` and `time2` respectively.

2 Plot the measured and simulated response, and residuals.

- a** Select the **Validation** node, and click **New**.

This action creates a **New Validation** node.

- b** Select the **New Validation** node.

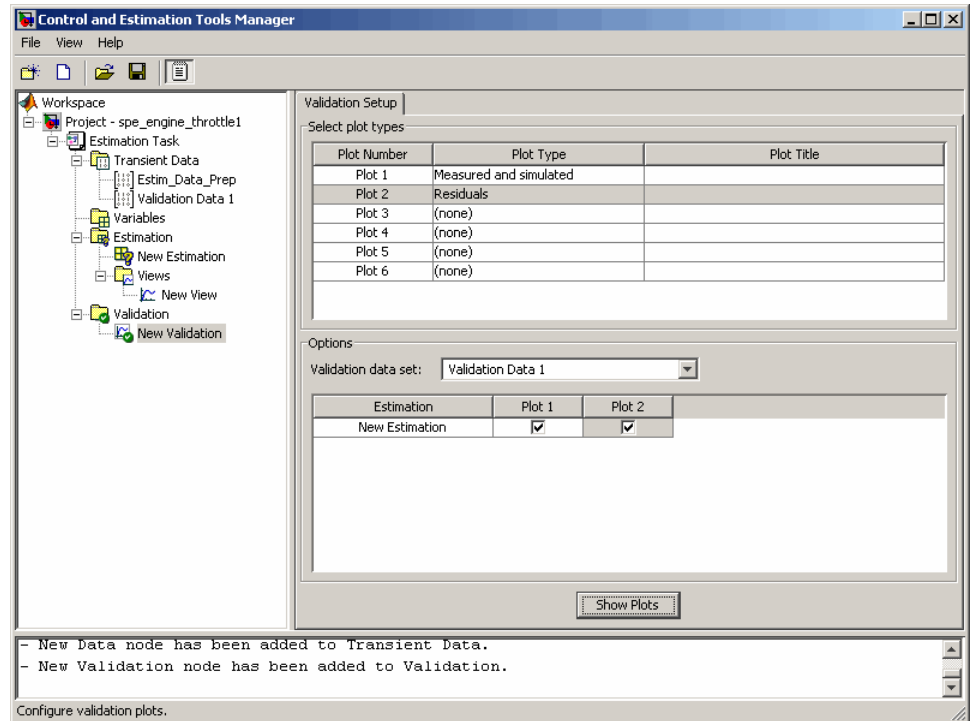
- c** In the **Validation Setup** tab, select the following plots in the **Plot Type** column.

- For **Plot 1**, select Measured and simulated.
- For **Plot 2**, select Residuals.

- d** In the **Options** area, select the **Plot 1**, and **Plot 2** check boxes.

- e Select **Validation Data1** from the **Validation data set** drop-down list.

The **Validation Setup** tab resembles the next figure.

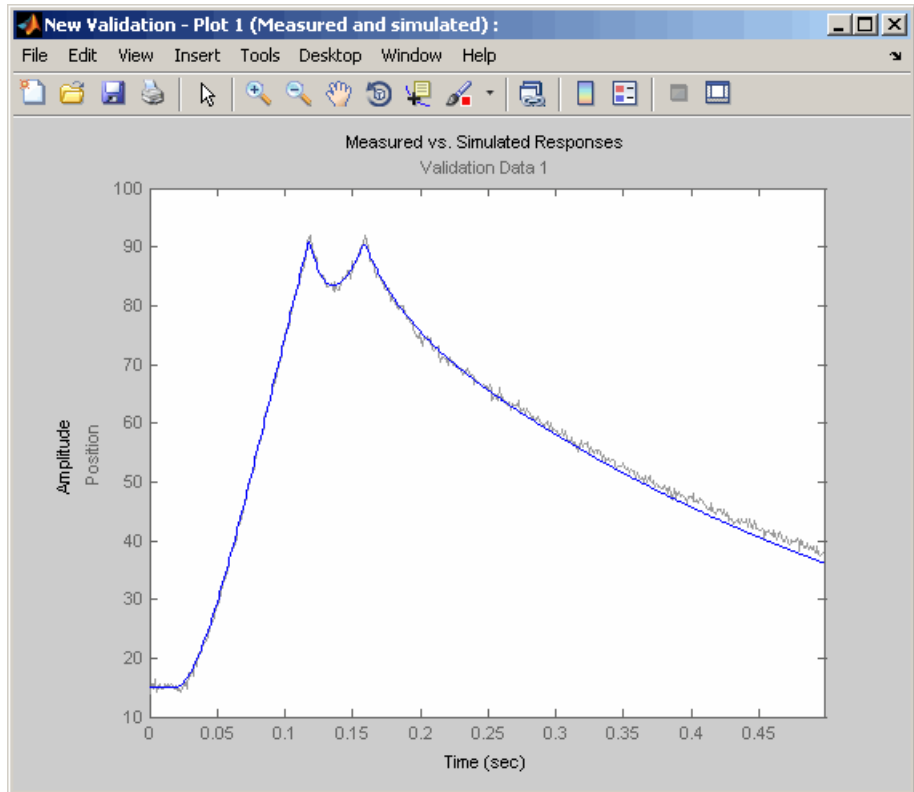


- f Click **Show Plots**.

This action opens the following figures:

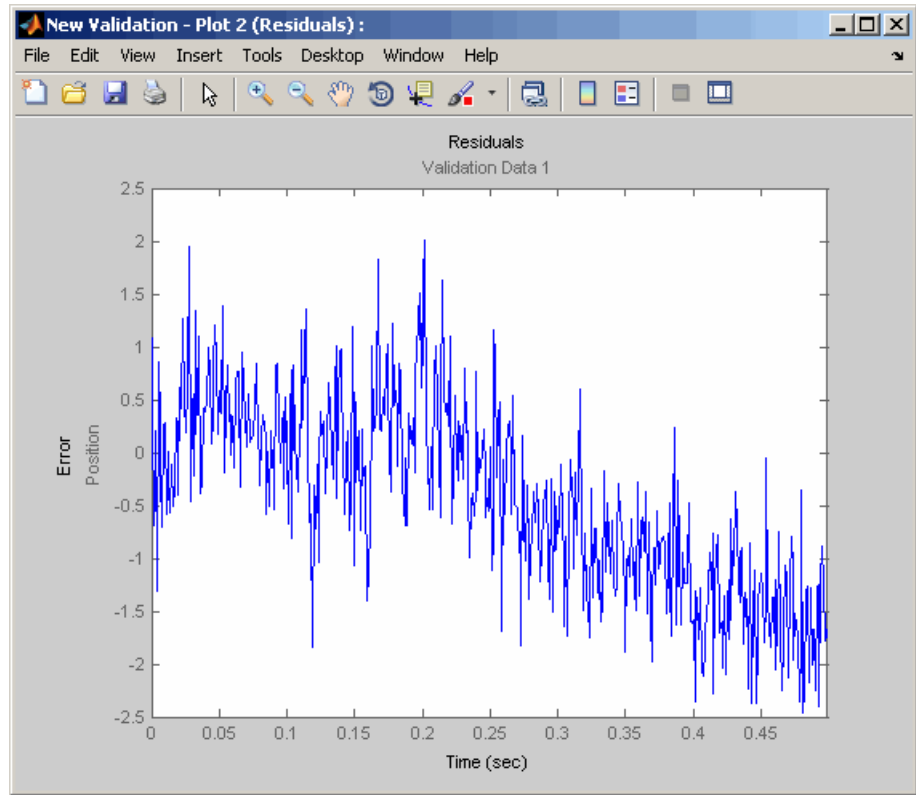
- New Validation - Plot 1 (Measured and simulated): Displays the validation data and simulated response.
 - New Validation - Plot 2(Residuals): Displays the difference between the measured data and simulated response.
- 3 Examine the simulated response plot to see how well the simulated output matches the validation data.

The simulated response, as shown in blue on the New Validation - Plot 1 (Measured and simulated) plot, is overlaid on the measured output data, and closely matches the measured data Validation Data 1.



- 4 Examine the residuals plot to compare the difference between the simulated response and measured data.

The difference between the simulated and measured data, as shown in the New Validation - Plot 2(Residuals) plot, varies between 2 and -2.5. The residuals lie within 6% of the maximum output variation, and do not display any systematic patterns. This indicates a good fit between the simulated output and measured data.



5 Save the Control and Estimation Tools Manager project.

a In the Control and Estimation Tools Manager GUI, select **File > Save**.

This action opens the Save Projects dialog box.

b In the Save Projects dialog box, click **OK**.

c In the Save Projects window, specify the name of the project in the **File name** field, and click **Save**.

This action saves the project as a MAT-file.

Tip You can load a project by selecting **File > Load** in the Control and Estimation Tools Manager GUI. To view the estimated parameter values, select the **Parameters** tab of **New Estimation** node.

Response Optimization

- “Supported Design Requirements” on page 3-2
- “Design Optimization to Meet Step Response Requirements (GUI)” on page 3-4
- “Design Optimization to Meet Step Response Requirements (Code)” on page 3-18
- “Design Optimization to Track Reference Signal (GUI)” on page 3-25
- “Design Optimization Using Frequency-Domain Check Blocks (GUI)” on page 3-40
- “Time-Domain Model Verification” on page 3-52

Supported Design Requirements

You can optimize response of Simulink models to meet time- and frequency-domain design requirements.

Simulink Design Optimization software optimizes model response by formulating the requirements into a constrained optimization problem. It then solves the problem using optimization methods.

- For time-domain requirements, the software simulates the model during optimization, compares the current response with the requirement and uses gradient methods to modify design variables (model parameters) to meet the objectives.

You can specify time-domain requirements either in blocks from the **Signal Constraints** library or without adding blocks to the model. You can also include requirements specified in Check Static Range, Check Static Lower Bound and Check Static Upper Bound blocks from the Simulink **Model Verification** library.

- For frequency-domain requirements, the software linearizes the portion of the model between specified linearization inputs and outputs, compares the linear system with the requirement and uses gradient methods to modify the design variables to meet the objectives.

If you have Simulink Control Design software, you can optimize the model to meet frequency-domain requirements, such as Bode magnitude and gain and phase margin bounds. You can specify the frequency-domain requirements without adding blocks to the model or by using the “Model Verification” blocks of the Simulink Control Design software library.

Related Examples

“Design Optimization to Meet Step Response Requirements (GUI)” on page 3-4

“Design Optimization to Meet Step Response Requirements (Code)” on page 3-18

“Design Optimization to Track Reference Signal (GUI)” on page 3-25

“Design Optimization to Meet Frequency-Domain Requirements (GUI)”

“Design Optimization Using Frequency-Domain Check Blocks (GUI)” on page 3-40

Design Optimization to Meet Time- and Frequency-Domain Requirements (GUI)

More About

“How the Optimization Algorithm Formulates Minimization Problems”

“Specify Time-Domain Design Requirements”

“Specify Frequency-Domain Design Requirements”

Design Optimization to Meet Step Response Requirements (GUI)

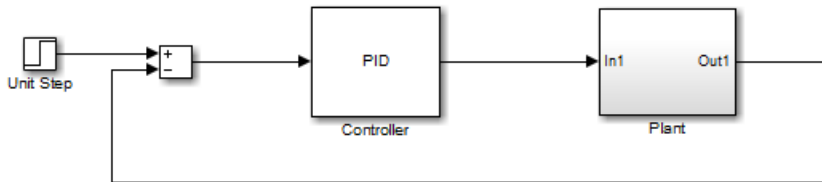
In this section...

- “Model Structure” on page 3-4
- “Design Requirements” on page 3-5
- “Specify Step Response Requirements” on page 3-5
- “Specify Design Variables” on page 3-8
- “Optimize Model Response” on page 3-12
- “Save the Session” on page 3-15

This example shows how to optimize controller parameters to meet step response design requirements using the Design Optimization tool. You specify the design requirements in a Check Step Response Characteristics block.

Model Structure

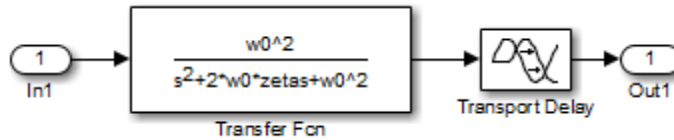
The model `sldo_model1` includes the following blocks:



- **Controller block**, which is a PID controller. This block controls the output of the Plant subsystem.
- **Unit Step block** applies a step input and produces the model output that should meet step response requirements.

You can also use other types of inputs, such as ramp, to optimize the response to meet step response requirements generated by such inputs.

- **Plant subsystem** is a second-order system with delay. It contains Transfer Function and Transport Delay blocks.



Design Requirements

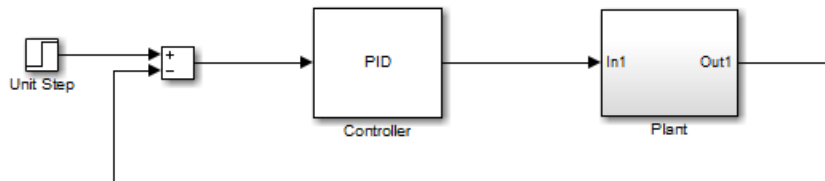
The plant output must meet the following step response requirements:

- Rise time less than 2.5 seconds
- Settling time less than 30 seconds
- Overshoot less than 5%

Specify Step Response Requirements

1 Open Simulink model.

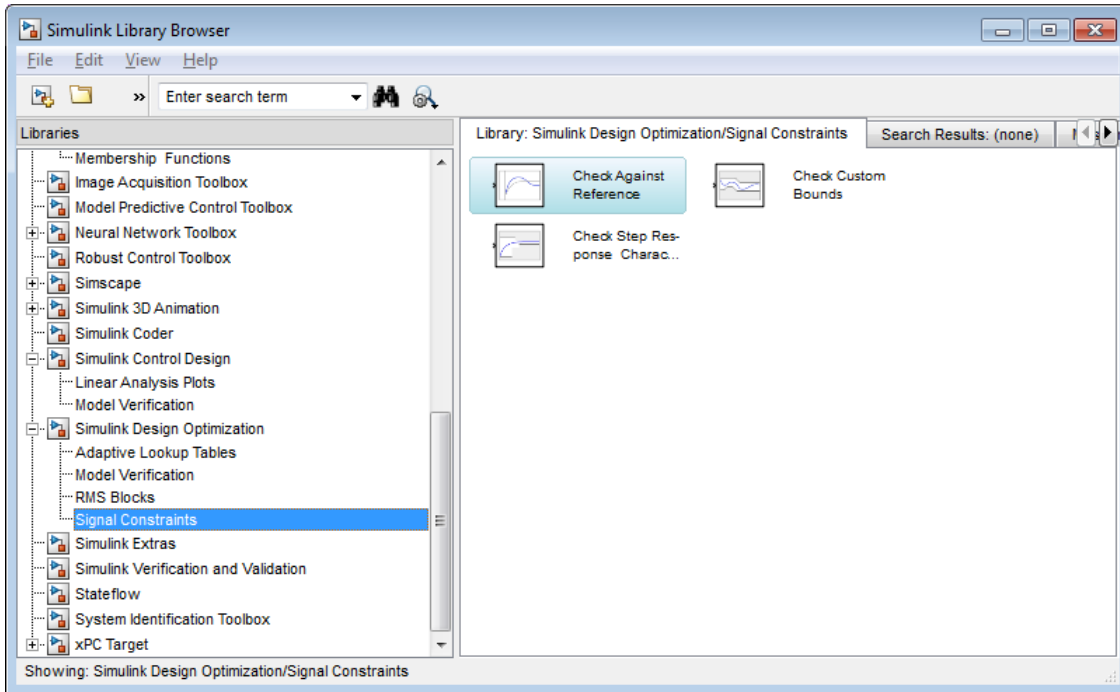
```
sys = 'sldo_model1';
open_system(sys);
```



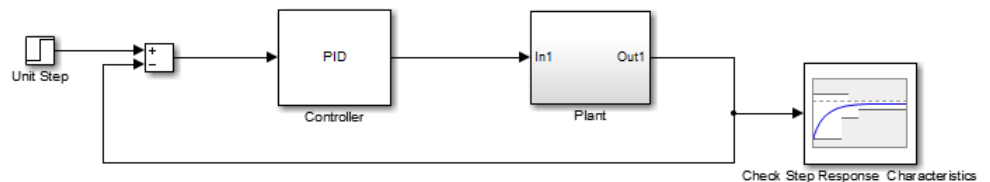
To learn more about the model, see “Model Structure” on page 3-4.

2 Add a Check Step Response Characteristics block to the model.

- In the Simulink model window, select **View > Library Browser**.
- In the **Libraries** pane, expand the **Simulink Design Optimization** node and select **Signal Constraints**.

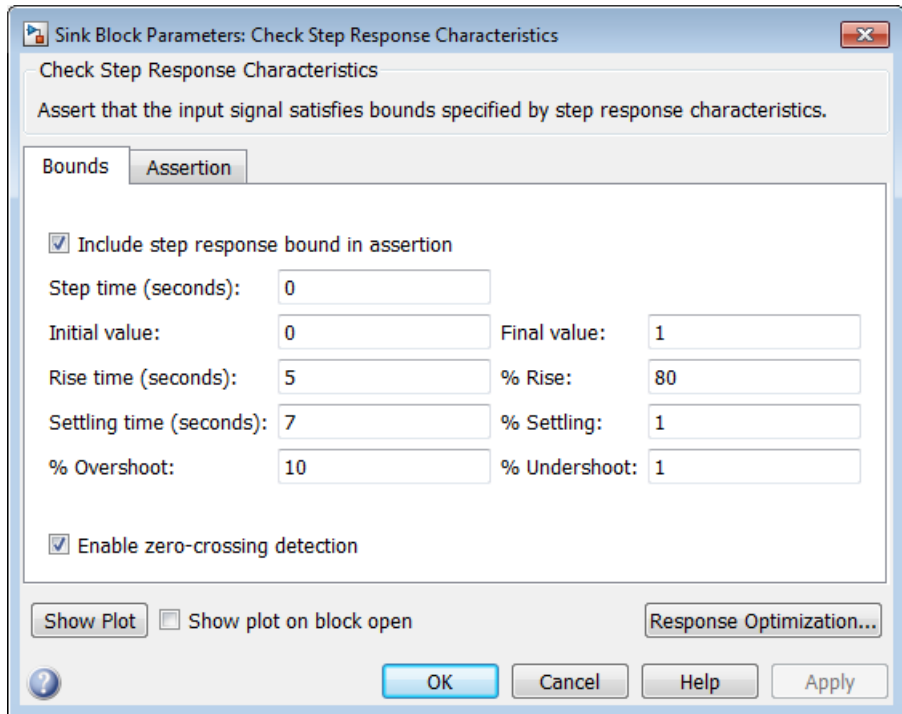


- c Drag and drop the Check Step Response Characteristics block into the model window.
- d Connect the block to the output.

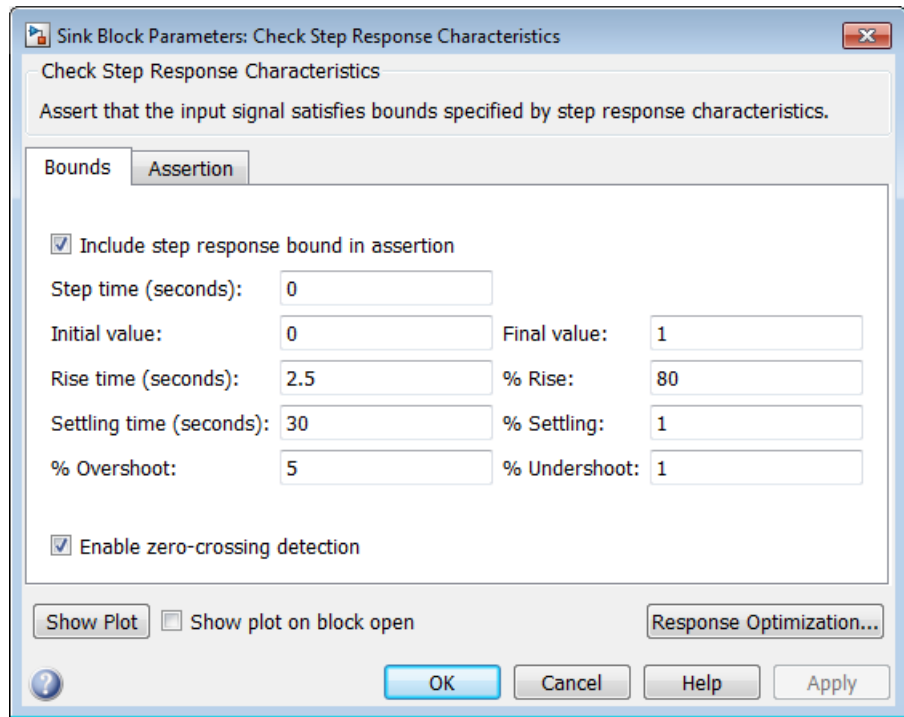


You must connect the block to the signal on which you want to specify design requirements.

- 3 Double-click the Check Step Response Characteristics block to open the Sink Block Parameters: Check Step Response Characteristics dialog box.



- 4 Specify step response requirements:
- In **Rise time (seconds)**, enter 2.5
 - In **Settling time (seconds)**, enter 30
 - In **% Overshoot**, enter 5



Click **OK**.

Instead of specifying time-domain requirements in the Check blocks, you can specify them in the Design Optimization tool without adding blocks. For an example, see “Specify Reference Signal” on page 3-26.

Specify Design Variables

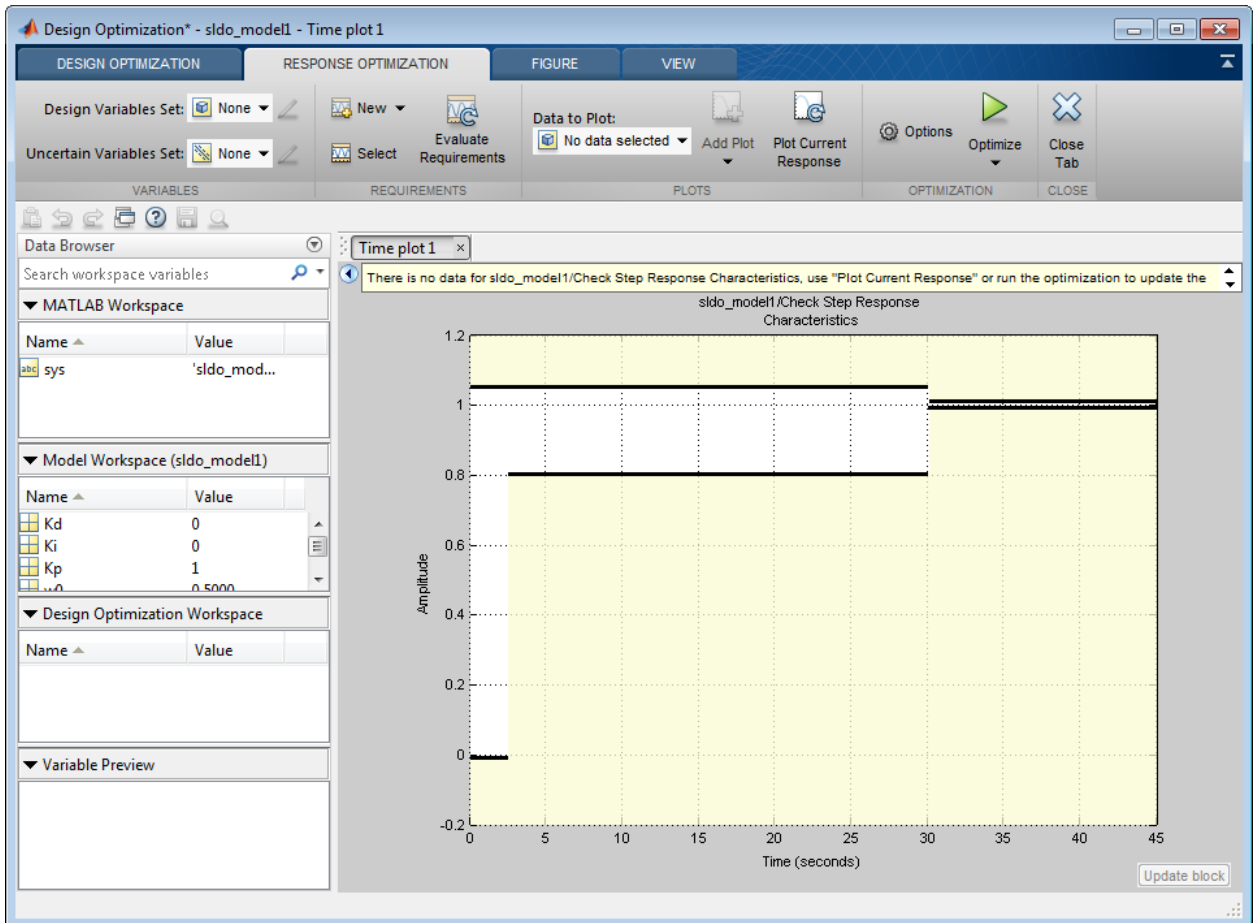
Before you begin this task, specify the design requirements as described in “Specify Step Response Requirements” on page 3-5.

When you optimize the model response, the software modifies the design variable (parameter) values to meet the design requirements.

- 1 In the Simulink model window, select **Analysis > Response Optimization**.

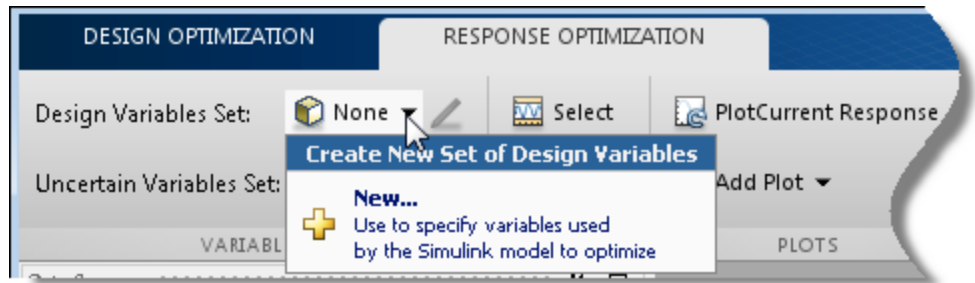
Alternatively, click **Response Optimization** in the Block Parameters dialog box.

A Design Optimization tool for the model opens.

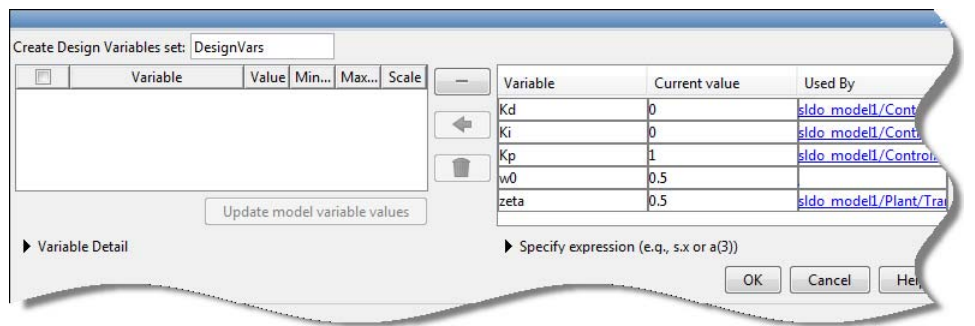


The amplitude versus time plot graphically shows the step response requirements specified in the Check Step Response Characteristics block.


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



A window opens where you specify design variables.



3 Click Kd, Ki and Kp to select them.

4 Click  to add the selected parameters to a design variables set.

Create Design Variables set: DesignVars

<input checked="" type="checkbox"/>	Variable	Value	Minimum	Maximum	Scale
<input checked="" type="checkbox"/>	Kd	0	-Inf	Inf	1
<input checked="" type="checkbox"/>	Ki	0	-Inf	Inf	1
<input checked="" type="checkbox"/>	Kp	1	-Inf	Inf	1

The software displays the following parameter settings:

- **Variable** — Parameter name

- **Value** — Current parameter value
- **Minimum** and **Maximum** — Parameter bounds
- **Scale** — Scaling factor for the parameter

The check-box indicates that the parameter is included in the design variable set. The default design variable set name is `DesignVars`.


- 5 To limit the parameters to positive values, enter the minimum value of each parameter as 0 in the corresponding **Minimum** field and press **Enter**.

Create Design Variables set:

<input checked="" type="checkbox"/>	Variable	Value	Minimum	Maximum	Scale
<input checked="" type="checkbox"/>	Kd	0	0	Inf	1
<input checked="" type="checkbox"/>	Ki	0	0	Inf	1
<input checked="" type="checkbox"/>	Kp	1	0	Inf	1

Click **OK**. A new variable `DesignVars` appears in **Design Optimization Workspace** of the Design Optimization tool. You can click the variable to view its contents in the **Variable Preview** area.

▼ Design Optimization Workspace

Name ▲	Value
 DesignVars	<3x1 para...

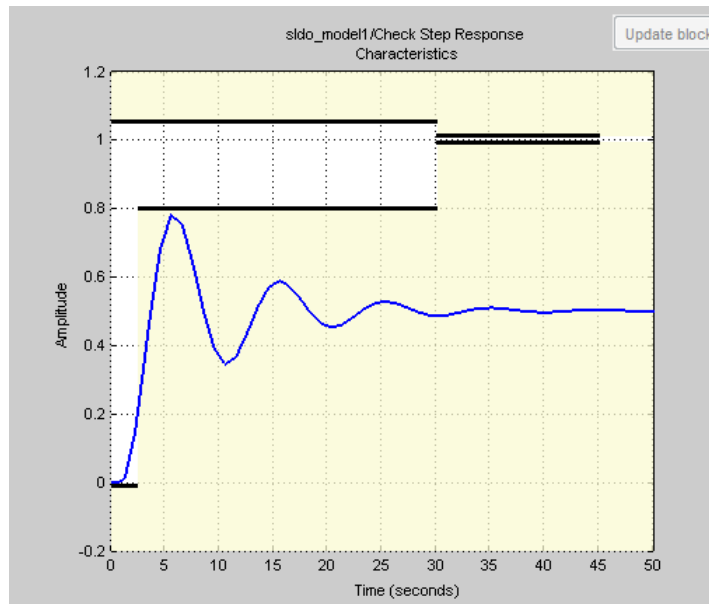
▼ Variable Preview

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

Optimize Model Response

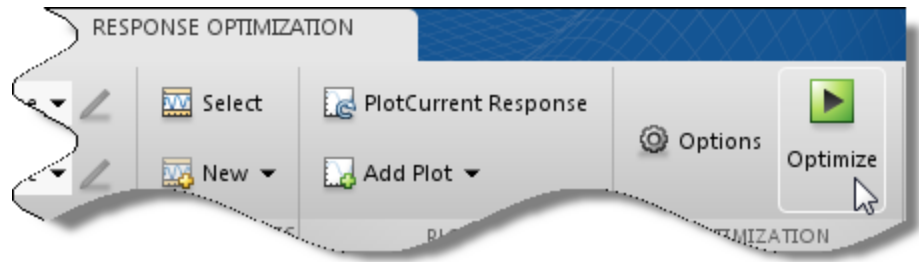
Before you begin this task, you must have already specified the design requirements and design variables as described in “Specify Step Response Requirements” on page 3-5, and “Specify Design Variables” on page 3-8, respectively.

- 1 (Optional) View the current response of the model. Click

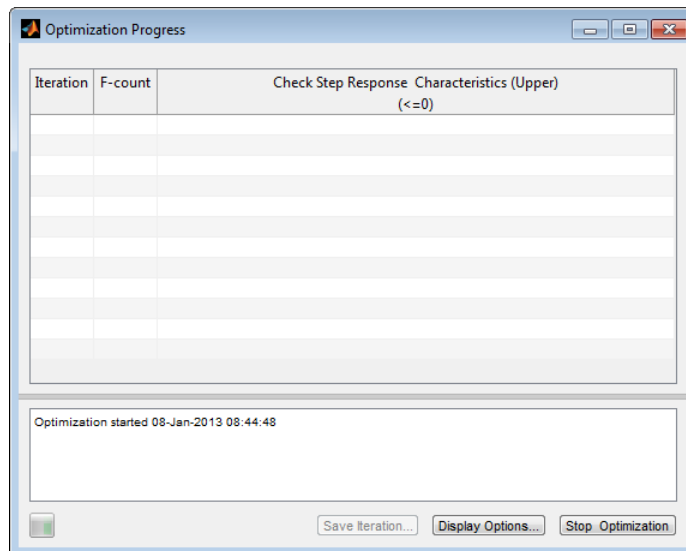



The plot shows that the model output does not meet the specified step response requirements.

- 2 Click **Optimize**.



An Optimization Progress window opens.

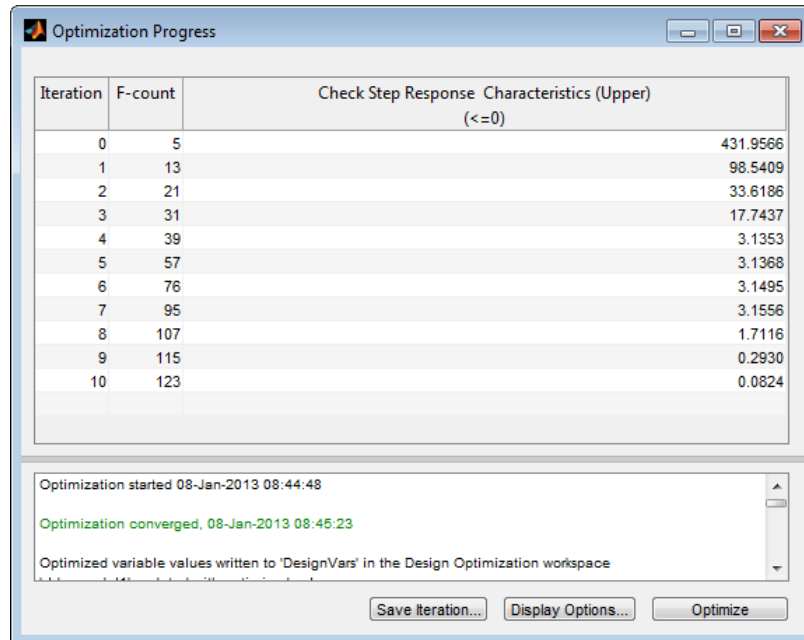


Tip To view the model response and optimization progress windows simultaneously, tile them using the plot layout area .

At each optimization iteration, the software simulates the model, and the default optimization solver Gradient descent (fmincon) modifies the design variables to reduce the distance between the simulated response and the design requirement line segments. For more information, see

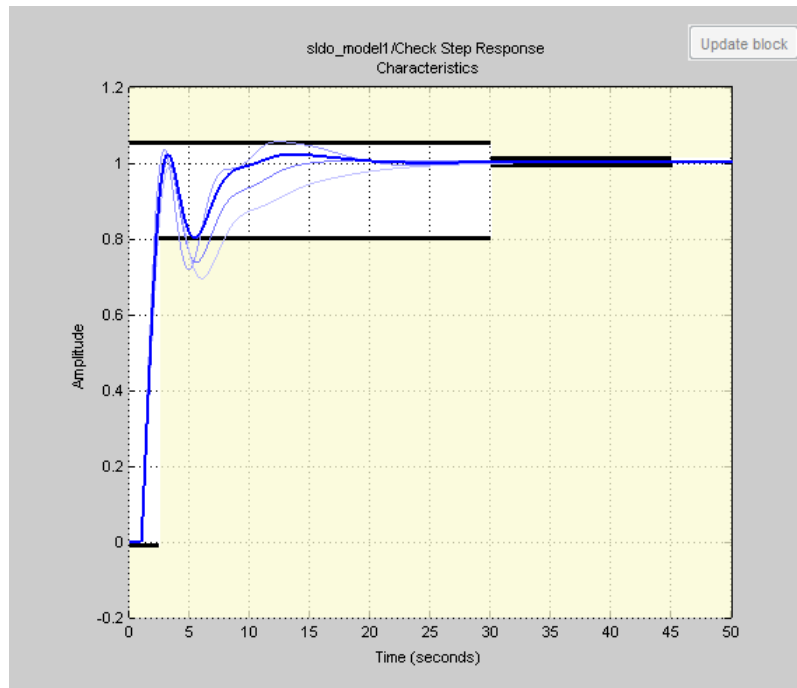
“Selecting Optimization Methods” and “How the Optimization Algorithm Formulates Minimization Problems”.

After the optimization completes, the optimization progress window resembles the next figure.



The message `Optimization converged` indicates that the optimization solver found a solution that meets the design requirements within the tolerances and parameter bounds. For more information about the outputs displayed in the optimization, see “Iterative Display” in the Optimization Toolbox documentation.

- 3 Verify that the model output meets the step response requirements.



The plot displays the last five iterations. The final response using the optimized parameter values appears as the thick line.

The optimized response lies in the white region bounded by the design requirement line segments and thus meets the requirements.

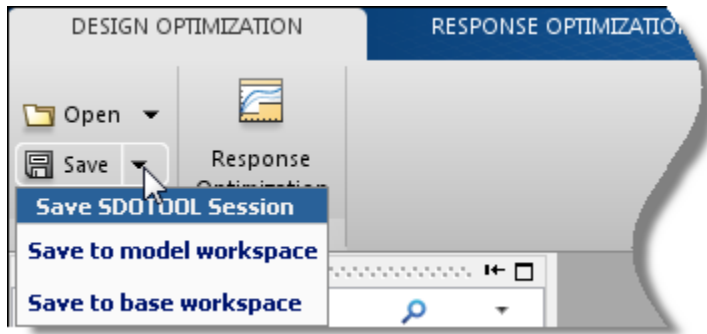
- 4** View the optimized parameter values. Click **DesignVars** in **Design Optimization Workspace** and view the updated values in the **Variable Preview** area.

The optimized values of the design variables are automatically updated in the Simulink model.

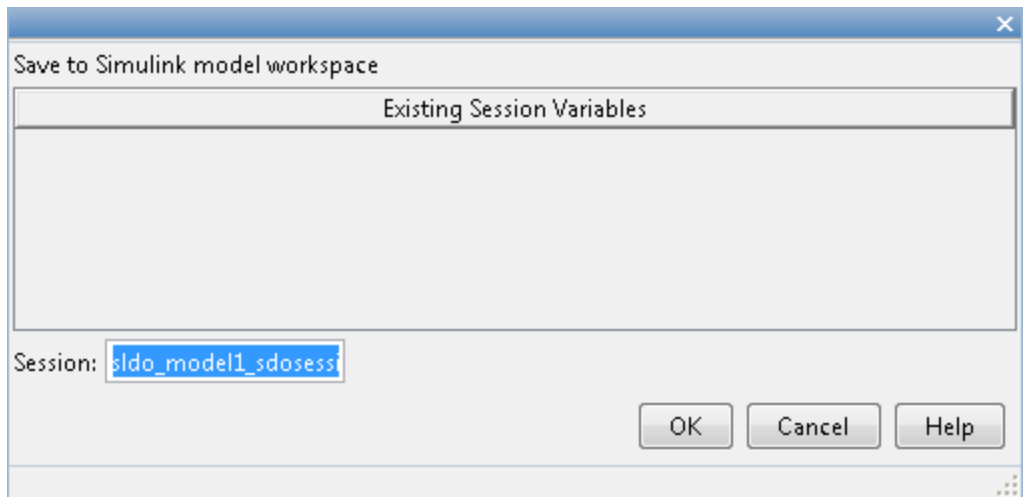
Save the Session

After you optimize the model response to meet design requirements, you can save the Design Optimization tool session which includes the optimized parameter values.

- 1 Click the **Design Optimization** tab of the Design Optimization tool.
- 2 In the **Save** drop-down list, select **Save to model workspace**.



A window opens where you specify the session name.



- 3 Specify a session name in the **Session** field.
Click **OK**.

Tip To open the saved session, click the **Open from model workspace** option in the **Open** drop-down list of the Design Optimization tool for the model.

Related Examples

- “Design Optimization to Meet Step Response Requirements (Code)” on page 3-18
- “Design Optimization to Track Reference Signal (GUI)” on page 3-25

Design Optimization to Meet Step Response Requirements (Code)

In this section...

“Model Structure” on page 3-18

“Design Requirements” on page 3-19

“Specify Step Response Requirements” on page 3-19

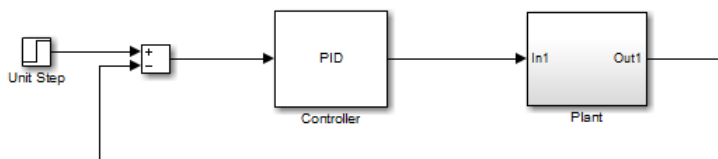
“Specify Design Variables” on page 3-20

“Optimize Model Response” on page 3-20

This example shows how to programmatically optimize controller parameters to meet step response requirements using `sdo.optimize`.

Model Structure

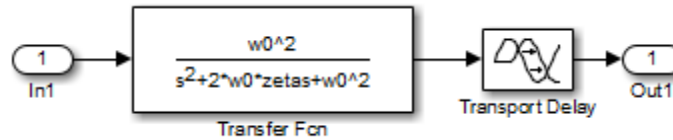
The model `sldo_model1` includes the following blocks:



- **Controller block**, which is a PID controller. This block controls the output of the Plant subsystem.
- **Unit Step block** applies a step input and produces the model output that should meet step response requirements.

You can also use other types of inputs, such as ramp, to optimize the response to meet step response requirements generated by such inputs.

- **Plant subsystem** is a second-order system with delay. It contains Transfer Function and Transport Delay blocks.



Design Requirements

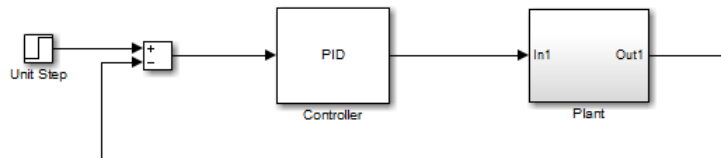
The plant output must meet the following step response requirements:

- Rise time less than 2.5 seconds
- Settling time less than 30 seconds
- Overshoot less than 5%

Specify Step Response Requirements

1 Open the Simulink model.

```
sys = 'sldo_model1';
open_system(sys);
```



2 Log the model output signal.

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

```
PlantOutput = Simulink.SimulationData.SignalLoggingInfo;
PlantOutput.BlockPath = [sys '/Plant'];
PlantOutput.OutputPortIndex = 1;
PlantOutput.LoggingInfo.NameMode = 1;
PlantOutput.LoggingInfo.LoggingName = 'PlantOutput';
```

3 Store the logging information.

```
simulator = sdo.SimulationTest(sys);
simulator.LoggingInfo.Signals = PlantOutput;
```

`simulator` is a `sdo.SimulationTest` object that you also use later to simulate the model.

4 Specify step response requirements.

```
StepResp = sdo.requirements.StepResponseEnvelope;
StepResp.RiseTime = 2.5;
StepResp.SettlingTime = 30;
StepResp.PercentOvershoot = 5;
```

`StepResp` is a `sdo.requirements.StepResponseEnvelope` object.

Specify Design Variables

When you optimize the model response, the software modifies parameter (design variable) values to meet the design requirements.

1 Select model parameters to optimize.

```
p = sdo.getParameterFromModel(sys,{'Kp','Ki','Kd'});
```

`p` is an array of 3 `param.Continuous` objects.

2 To limit the parameters to positive values, set the minimum value of each parameter to 0.

```
p(1).Minimum = 0;
p(2).Minimum = 0;
p(3).Minimum = 0;
```

Optimize Model Response

1 Create a design function.

```
evalDesign = @(p) sldo_model1_design(p,simulator,StepResp);
```

`evalDesign` is an anonymous function that calls the cost function `sldo_model1_design`. The cost function simulates the model and evaluates the design requirements.

Tip Type `edit sldo_model1_design` to view this function.

2 (Optional) Evaluate the current response.

- a** Compute the model response using the current values of the design variables.

```
initDesign = evalDesign(p);
```

During simulation, the Step Response block throws assertion warnings at the MATLAB prompt which indicate that the requirements specified in the block are not satisfied.

- b** Examine the nonlinear inequality constraints.

```
initDesign.Cleq
```

```
ans =
```

```
-0.2571
-0.4968
-0.5029
-1.0000
 0.7060
 0.5092
 0.4964
201.4682
```

Some `Cleq` values are positive, beyond the specified tolerance, indicating that the response using the current parameter values violates the design requirements.

3 Specify optimization options.

```
opt = sdo.OptimizeOptions;
opt.MethodOptions.Algorithm = 'active-set';
```

The software configures `opt` to use the default optimization method, `fmincon`. However, because the optimization problem specifies only constraints, you specify the Active Set algorithm for `fmincon`. The Active

Set algorithm is good for solving feasibility (constraints only) optimization problems.

4 Optimize the response.

```
[pOpt,optInfo] = sdo.optimize(evalDesign,p,opt);
```

At each optimization iteration, the software simulates the model, and the default optimization solver `fmincon` modifies the design variables to meet the design requirements. For more information, see “Selecting Optimization Methods” and “How the Optimization Algorithm Formulates Minimization Problems”.

After the optimization completes, the command window displays the following results:

Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
0	5	0	201.5		
1	12	0	167.5	0.687	0
2	19	0	0.8462	1.22	5.86
3	29	0	0.8169	0.133	0
4	37	0	0.7819	0.115	0
5	44	0	3.121	0.265	3.29
6	52	0	3.328	2.92	0
7	59	0	0.1413	1.26	2.81
8	66	0	0.02418	0.443	0.0029
9	73	0	8.125e-05	0.0563	1.03e-05

Local minimum found that satisfies the constraints.

Optimization completed because the objective function is non-decreasing in feasible directions, to within the selected value of the function tolerance, and constraints are satisfied to within the selected value of the constraint tolerance.

The message `Local minimum found that satisfies the constraints` indicates that the optimization solver found a solution that meets the design requirements within specified tolerances. For more information about the outputs displayed during the optimization, see “Iterative Display” in the Optimization Toolbox documentation.

- 5 Examine the optimization termination information, contained in the `optInfo` output argument. This information helps you verify that the response meets the step response requirements.

For example, check the following fields:

- `Cleq`, which shows the optimized nonlinear inequality constraints.

```
optInfo.Cleq
```

```
ans =
```

```
0.0001  
-0.0099  
-0.0099  
-1.0000  
0.0004  
-0.0100  
-0.0101  
-0.1997
```

All values satisfy $Cleq \leq 0$, within the optimization tolerances, which indicates that the step response requirements are satisfied.

- `exitflag`, which identifies why the optimization terminated.

The value is 1 which indicates that the solver found a solution which was less than the specified tolerances on the function value and constraint violations.

- 6 View the optimized parameter values.

```
pOpt
```

```
pOpt(1,1) =
```

```
Name: 'Kp'  
Value: 1.2364  
Minimum: 0  
Maximum: Inf  
Free: 1  
Scale: 1  
Info: [1x1 struct]
```

```
pOpt(2,1) =  
  
    Name: 'Ki'  
    Value: 0.3901  
    Minimum: 0  
    Maximum: Inf  
    Free: 1  
    Scale: 1  
    Info: [1x1 struct]
```

```
pOpt(3,1) =  
  
    Name: 'Kd'  
    Value: 2.5937  
    Minimum: 0  
    Maximum: Inf  
    Free: 1  
    Scale: 1  
    Info: [1x1 struct]
```

7 Simulate the model with the optimized values.

a Update optimized parameter values in the model.

```
sdo.setValueInModel(sys,pOpt);
```

b Simulate the model.

```
sim(sys);
```

See Also

[sdo.optimize](#) | [Simulink.SimulationData.SignalLoggingInfo](#) | [sdo.SimulationTest](#) | [sdo.getParameterFromModel](#) | [sdo.requirements.StepResponseEnvelope](#) | [param.Continuous](#) | [sdo.OptimizeOptions](#)

Related Examples

- “Design Optimization to Meet Step Response Requirements (GUI)” on page 3-4
- “Design Optimization to Track Reference Signal (GUI)” on page 3-25

Design Optimization to Track Reference Signal (GUI)

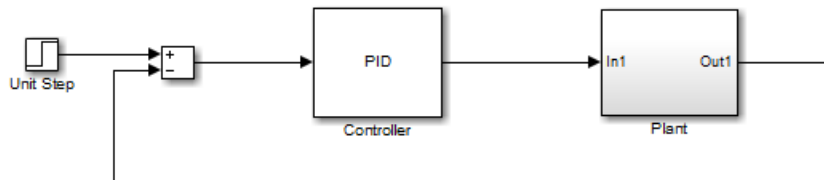
In this section...

- “Model Structure” on page 3-25
- “Design Requirements” on page 3-26
- “Specify Reference Signal” on page 3-26
- “Specify Design Variables” on page 3-33
- “Optimize Model Response” on page 3-36

This example shows how to optimize controller parameters to track a reference signal using the Design Optimization tool. You specify the reference signal without adding any Check blocks to the model.

Model Structure

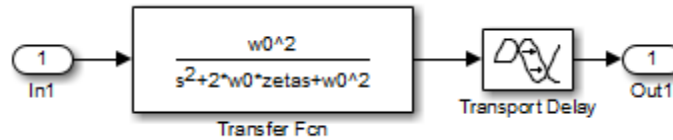
The model `sldo_model1` includes the following blocks:



- **Controller block**, which is a PID controller. This block controls the output of the Plant subsystem.
- **Unit Step block** applies a step input and produces the model output that should meet step response requirements.

You can also use other types of inputs, such as ramp, to optimize the response to meet step response requirements generated by such inputs.

- **Plant subsystem** is a second-order system with delay. It contains Transfer Function and Transport Delay blocks.



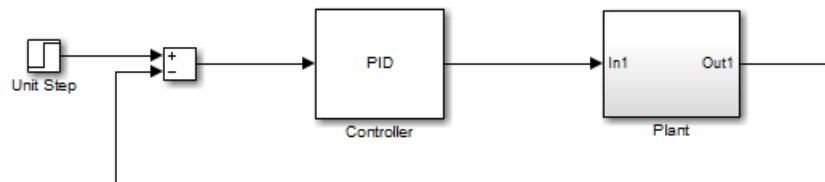
Design Requirements

The model output must track a reference signal $y = 1 - \exp(-0.1 \times t)$, where t is time.

Specify Reference Signal

- 1 Open Simulink model.

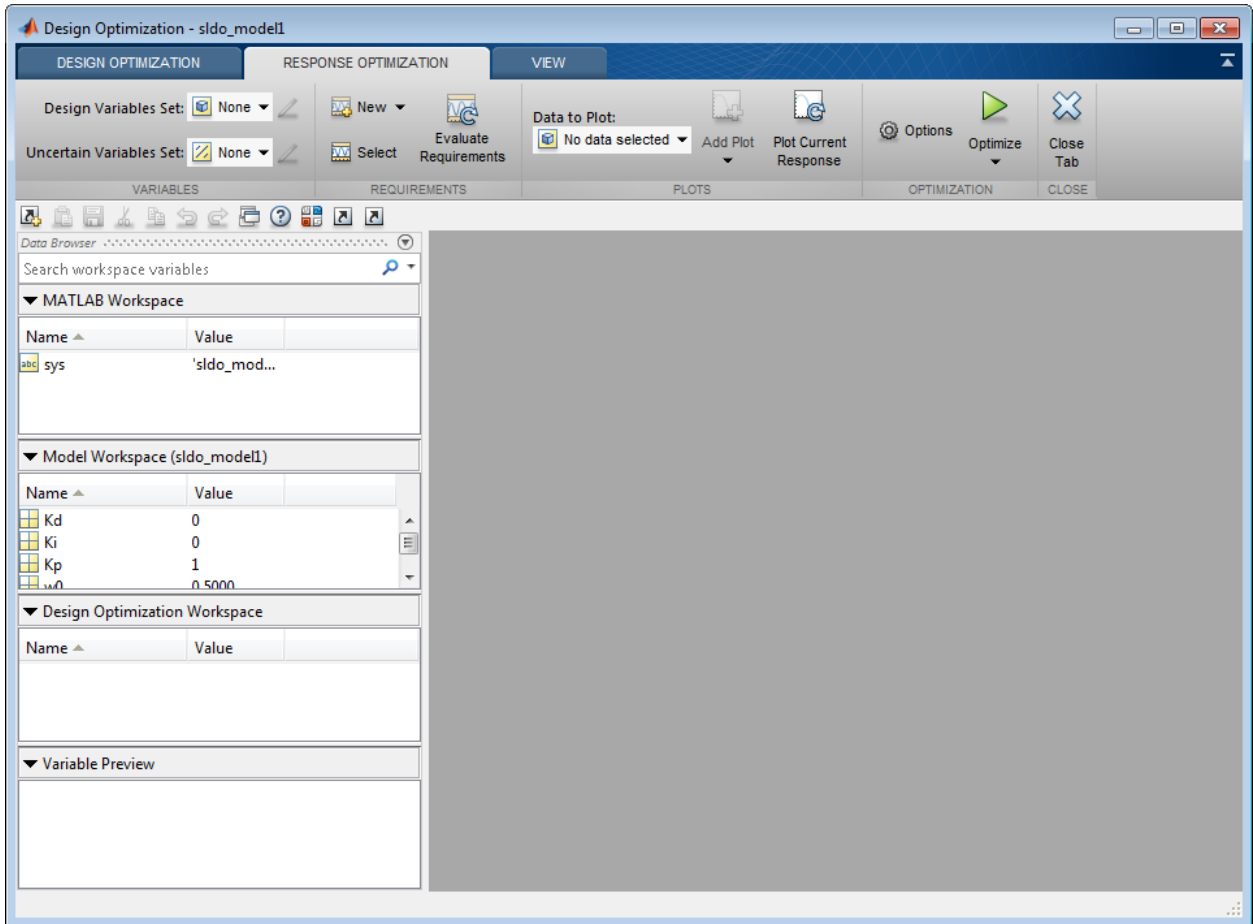
```
sys = 'sldo_model11';  
open_system(sys);
```



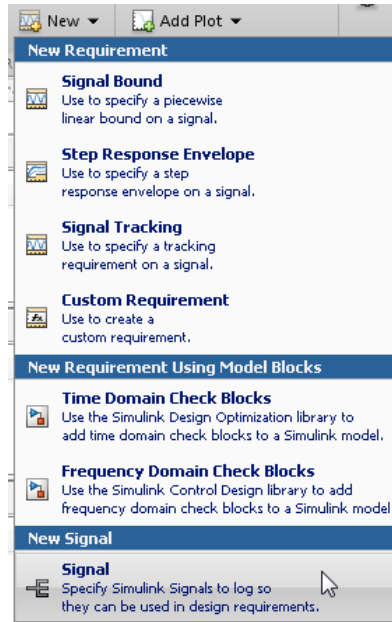
To learn more about the model, see “Model Structure” on page 3-25.

- 2 In the Simulink model window, select **Analysis > Response Optimization**.

A Design Optimization tool for the model opens.



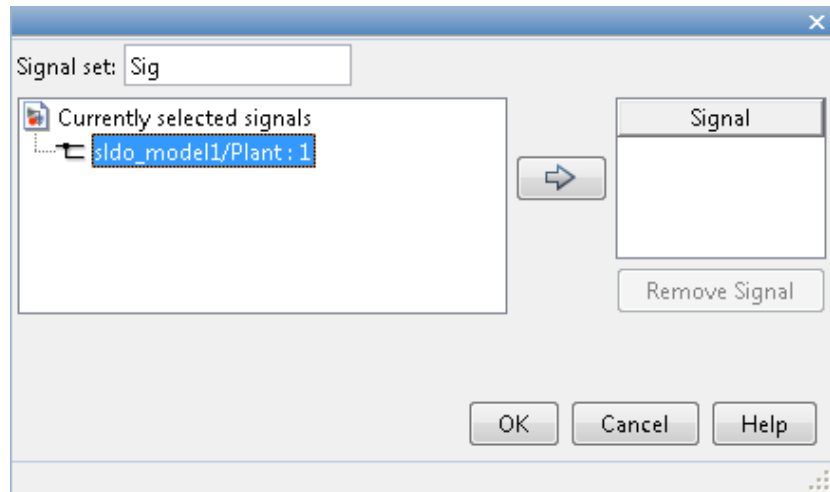
- 3 Select the model signal which must track the reference signal.
 - a In the **New** drop-down list, select **Signal**.



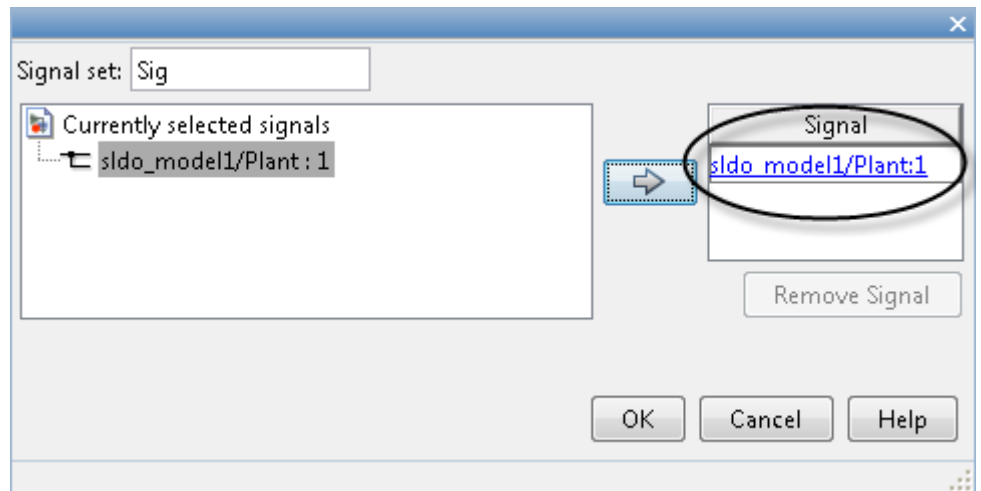
A window opens where you select a model signal.



- b** In the Simulink model window, click the output of the Plant block.
The window updates to display the selected signal.



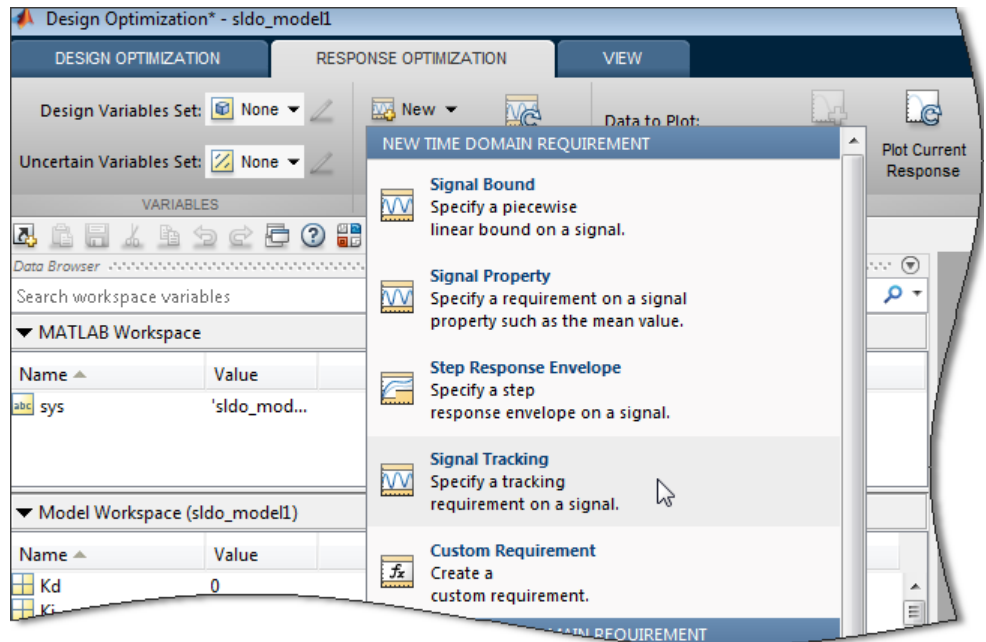
- c** Select the signal and click  to add it to the signal set.



- d** In **Signal set**, enter PlantOutput as the selected signal name.

Click **OK**. A new variable PlantOutput appears in the **Design Optimization Workspace** of the Design Optimization tool.

- 4 Specify the reference signal that the model output must track.
 - a In the **New** drop-down list, select **Signal Tracking**.



A window opens where you specify the reference signal.

Create Requirement

Signal Tracking
Specify a tracking requirement on a signal.

Name:

▼ **Specify Reference Signal**

Time vector:

Amplitude:

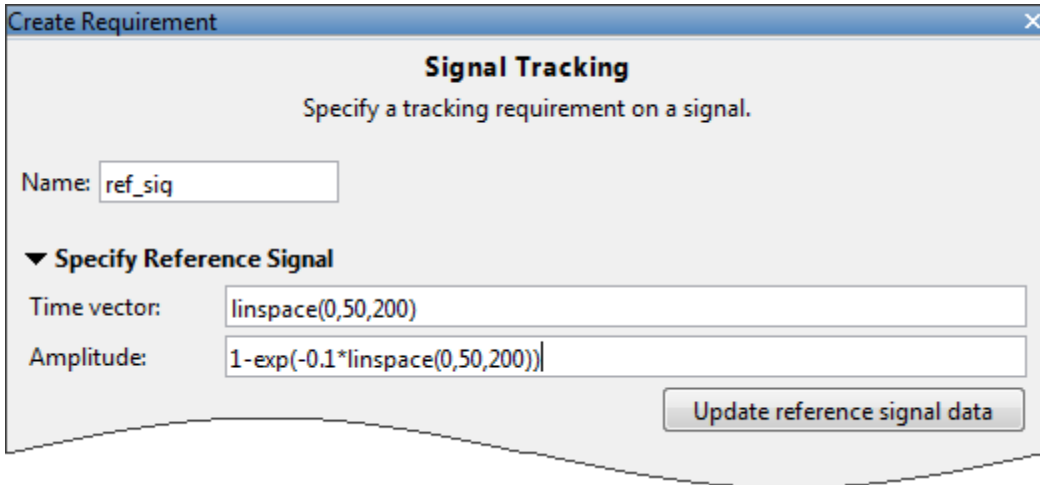
Tracking Method:

▼ **Specify Signal to Track Reference Signal**

	Signal
<input type="checkbox"/>	PlantOutput (sldo_model1/Plant:1)

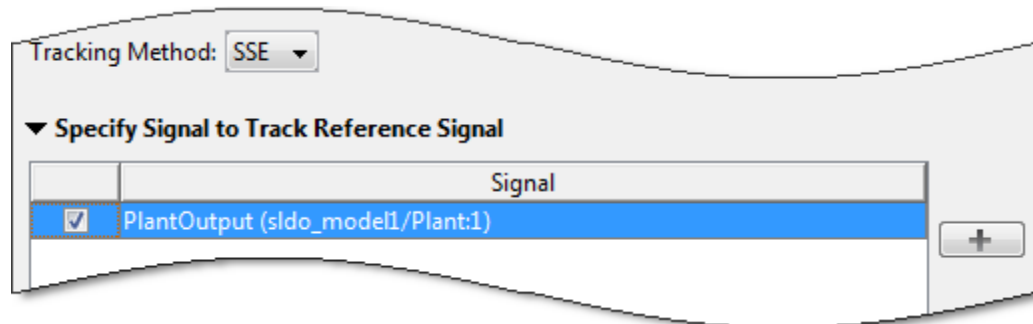
Create Plot

- b** In the **Name** edit box, enter `ref_sig`.
- c** In the **Time vector** edit box, enter `linspace(0,50,200)`
- d** In the **Amplitude** edit box, enter `1 - exp(-0.1*linspace(0,50,200))`.

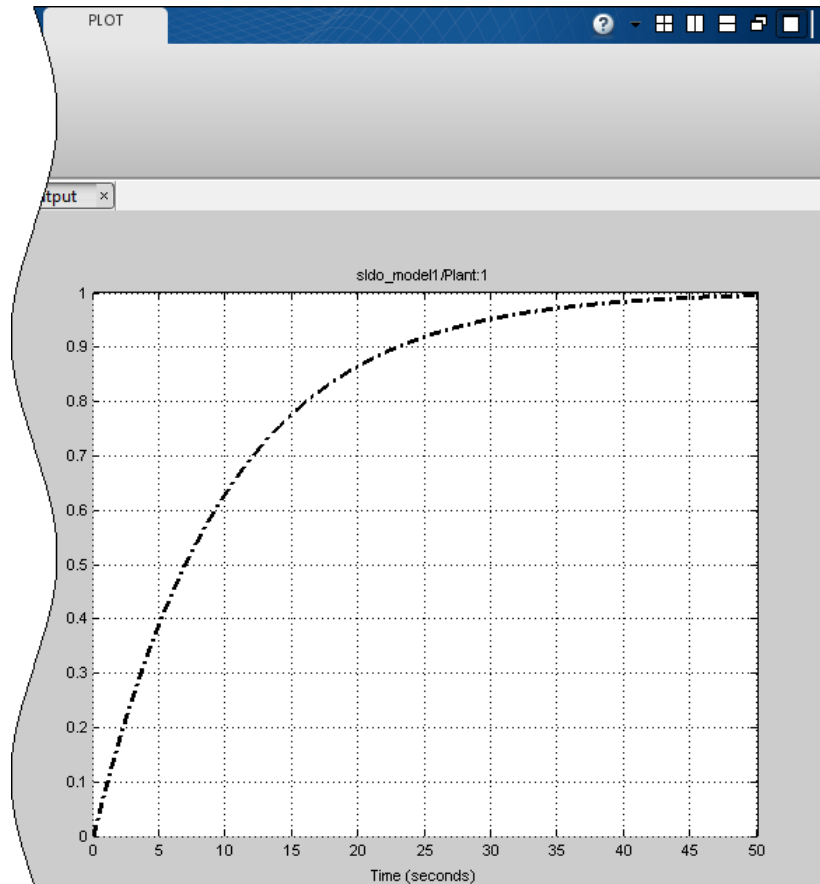


The **Tracking Method** is SSE, which means that at each optimization iteration, the solver attempts to reduce the sum of squared errors between the simulated output and reference signal.

- e Click **Update reference signal data**.
- f Select the check-box corresponding to the signal you selected in the previous step in the **Specify Signal to Track Reference Signal** area.



Click **OK**. A new variable `ref_sig` appears in the **Design Optimization Workspace** and the Design Optimization window updates to plot the reference signal.



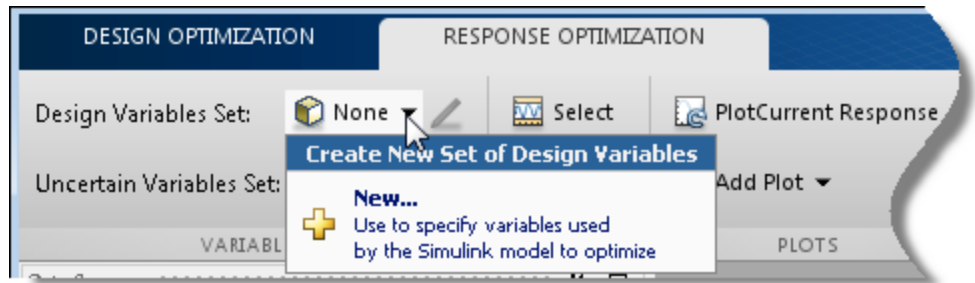
Specify Design Variables

Before you begin this task, specify the reference signal to track as described in “Specify Reference Signal” on page 3-26.

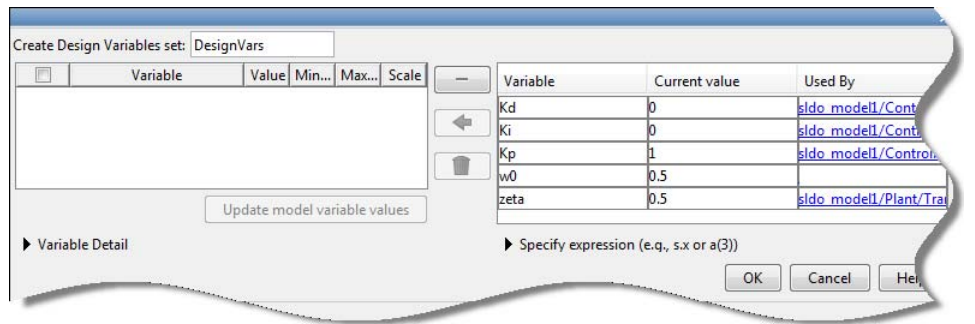
When you optimize the model response, the software modifies the design variable (model parameter) values to meet the design requirements.

In the **Response Optimization** tab:


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

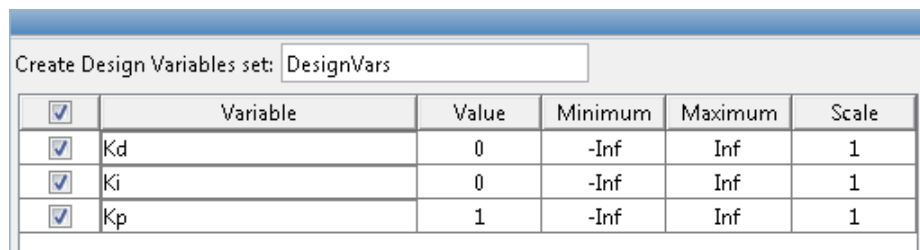


A window opens where you specify design variables.



2 Click Kd, Ki and Kp to select them.

3 Click  to add the selected parameters to a design variables set.



The software displays the following parameter settings:

- **Variable** — Parameter name

- **Value** — Current parameter value
- **Minimum** and **Maximum** — Parameter bounds
- **Scale** — Scaling factor for the parameter

The check-box indicates that the parameter is included in the design variable set. The default design variable set name is `DesignVars`.

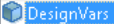
- 4 To limit the parameters to positive values, enter the minimum value of each parameter as 0 in the corresponding **Minimum** field and press **Enter**.

Create Design Variables set:

<input checked="" type="checkbox"/>	Variable	Value	Minimum	Maximum	Scale
<input checked="" type="checkbox"/>	Kd	0	0	Inf	1
<input checked="" type="checkbox"/>	Ki	0	0	Inf	1
<input checked="" type="checkbox"/>	Kp	1	0	Inf	1

Click **OK**. A new variable `DesignVars` appears in **Design Optimization Workspace** of the Design Optimization tool. You can click the variable to view its contents in the **Variable Preview** area.

▼ Design Optimization Workspace

Name ▲	Value
 DesignVars	<3x1 para...

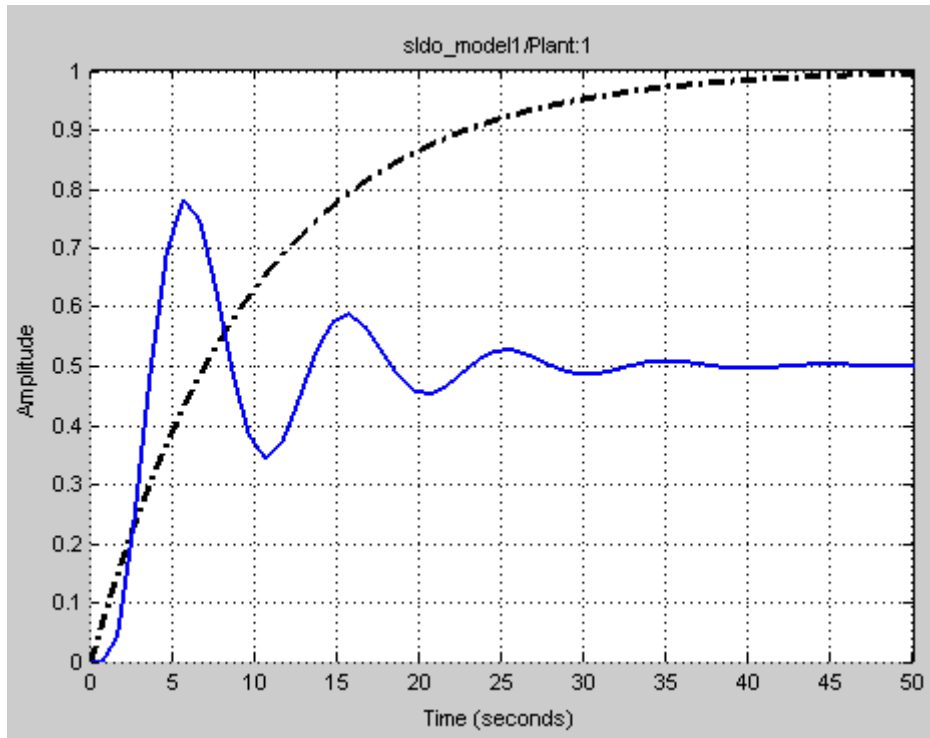
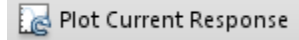
▼ Variable Preview

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

Optimize Model Response

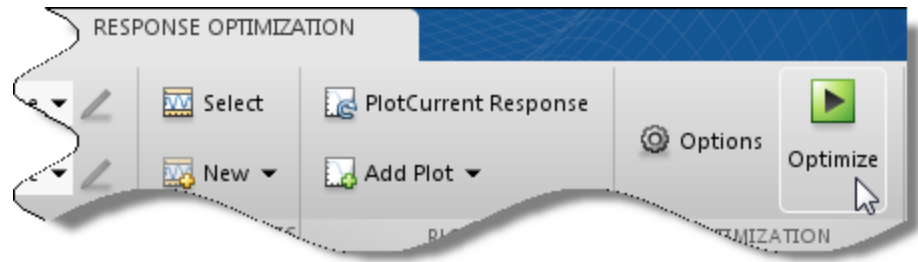
Before you begin this task, you must have specified the reference signal to track and design variables, as described in “Specify Reference Signal” on page 3-26 and “Specify Design Variables” on page 3-33, respectively.

1 (Optional) View the current model response. Click




The plot shows that the response does not track the reference signal.

2 Click **Optimize**.

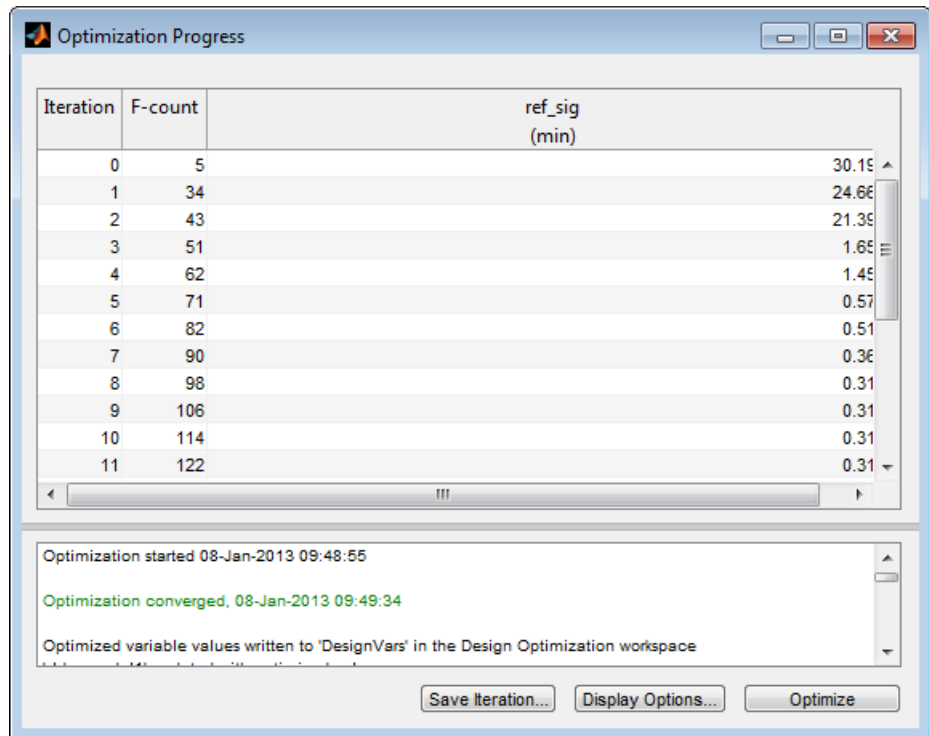


An optimization progress window opens.

Tip To view the model response and optimization progress windows simultaneously, tile them using the plot layout area .

At each iteration, the optimization solver Gradient descent (fmincon) modifies the controller parameters to minimize the error between the simulated response and the reference signal. To learn more, see “How the Optimization Algorithm Formulates Minimization Problems”.

After the optimization completes, the optimization progress window resembles the following figure.



The screenshot shows the 'Optimization Progress' window. It contains a table with the following data:

Iteration	F-count	ref_sig (min)
0	5	30.15
1	34	24.66
2	43	21.36
3	51	1.65
4	62	1.45
5	71	0.57
6	82	0.51
7	90	0.36
8	98	0.31
9	106	0.31
10	114	0.31
11	122	0.31

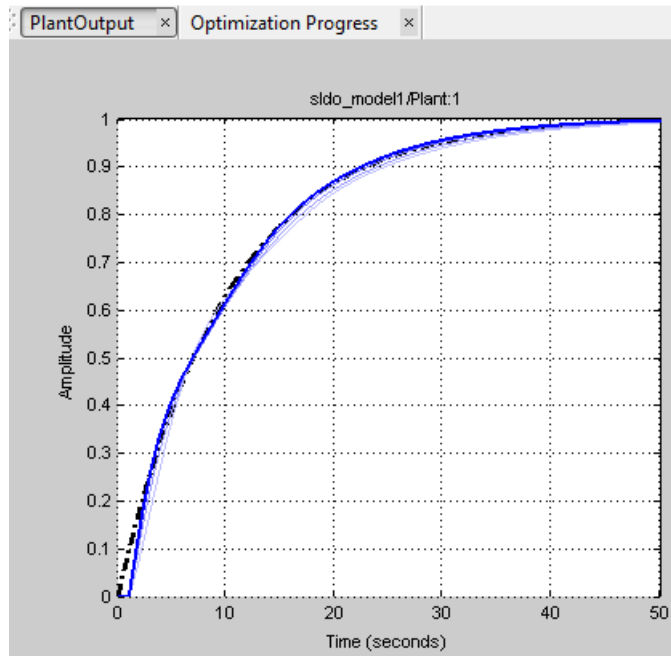
Below the table, the log shows the following messages:

- Optimization started 08-Jan-2013 09:48:55
- Optimization converged, 08-Jan-2013 09:49:34
- Optimized variable values written to 'DesignVars' in the Design Optimization workspace

At the bottom of the window, there are three buttons: 'Save Iteration...', 'Display Options...', and 'Optimize'.

The message **Optimization converged** indicates that the optimization method found a solution that tracks the reference signal within the tolerances and parameter bounds. For more information about the outputs displayed in the optimization progress window, see “Iterative Display” in the Optimization Toolbox documentation.

- 3 Verify that the response tracks the reference signal.



The optimized response closely tracks the reference signal.

- 4** View the optimized parameter values. Click **DesignVars** in **Design Optimization Workspace** and view the updated values in the **Variable Preview** area.

The optimized values of the design variables are automatically updated in the Simulink model.

Related Examples

- “Design Optimization to Meet Step Response Requirements (GUI)” on page 3-4
- “Design Optimization to Meet Step Response Requirements (Code)” on page 3-18

Design Optimization Using Frequency-Domain Check Blocks (GUI)

In this section...

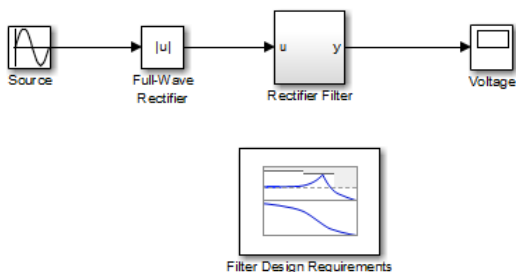
- “Model Structure” on page 3-40
- “Design Requirements” on page 3-41
- “Specify Design Requirements” on page 3-41
- “Specify Design Variables” on page 3-44
- “Optimize Design” on page 3-46

This example shows how to optimize a design to meet frequency-domain requirements using the Design Optimization tool. Simulink Control Design software must be installed to optimize a design to meet frequency-domain design requirements.

In this example, you specify the design requirements in a Check Bode Characteristics block and optimize a rectifier filter parameters to meet gain and bandwidth requirements, and minimize a custom objective.

Model Structure

The model `sdoirectifier` includes the following blocks:



- **Full-Wave Rectifier block** — An Abs block
- **Rectifier Filter subsystem** — RLC filter implemented using integrator and gain blocks

- **Filter Design Requirements block** — Check Bode Characteristics block that specifies the gain and bandwidth design requirements

Design Requirements

The design optimization problem is multi-objective. The design must:

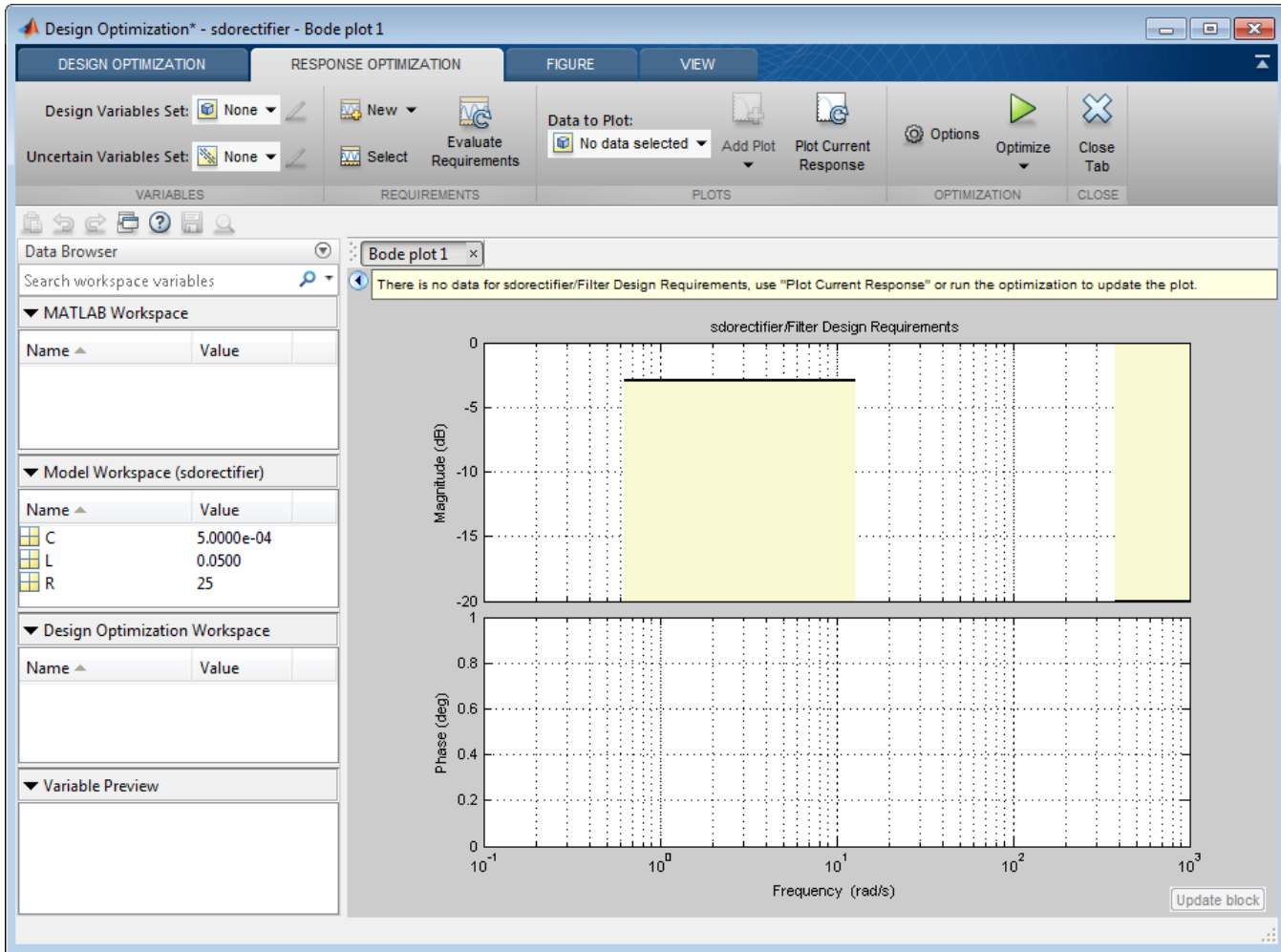
- Have a -3db bandwidth of at least 2Hz
- Limit the gain across the frequency range 2Hz — 60Hz to at most 0db
- Limit the gain above 60Hz to at most -20db
- Maximize the filter resistance R
- Minimize the filter inductance L

The requirements ensure that the rectifier filter combination has minimal high frequency content, responds quickly to voltage changes, and limits filter currents.

Specify Design Requirements

- 1 Open the Design Optimization tool for the model.

```
sdotool('sdorectifier')
```



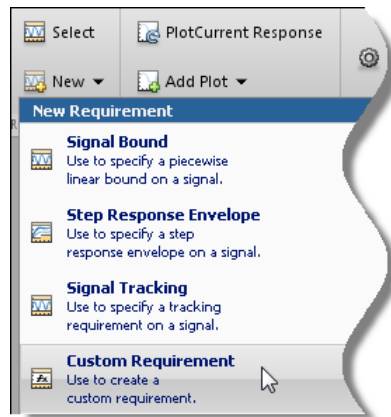
The plot shows the gain and bandwidth requirements specified in the Filter Design Requirements block. See their values in the **Bounds** tab of the Block Parameters dialog box.

- 2 Specify a custom objective to minimize the filter inductance and maximize the resistance.

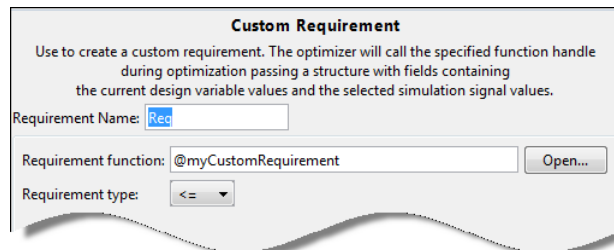
The custom objective is specified in the `sdorectifier_cost` function. The function accepts the design variables R and L , and returns the objective to be minimized.

Tip Type `edit sdorectifier_cost` to view this function.

- a** Select **Custom Requirement** in the **New** drop-down list.

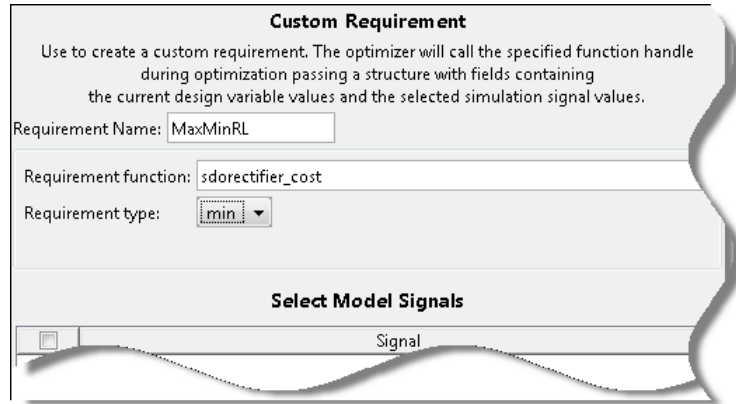


A window opens where you specify the custom requirement.



- b** Specify the following:
- Enter `MaxMinRL` in **Requirement Name**
 - Enter `@sdorectifier_cost` in **Requirement function**. The optimization solver calls the specified function handle.

- Select **min** in **Requirement type**



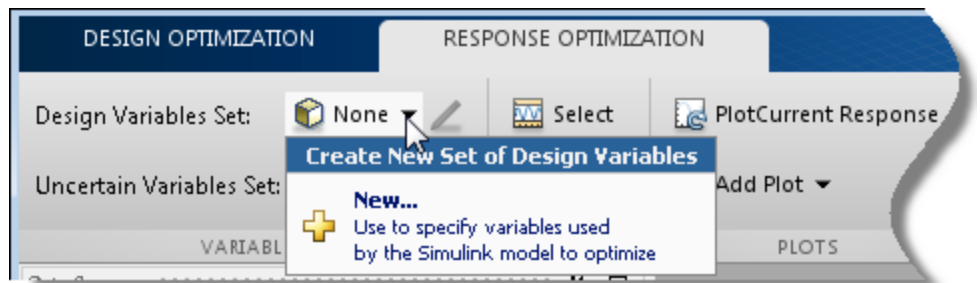
Click **OK**. A new variable **MaxMinRL** appears in **Design Optimization Workspace** of the Design Optimization tool.

Specify Design Variables

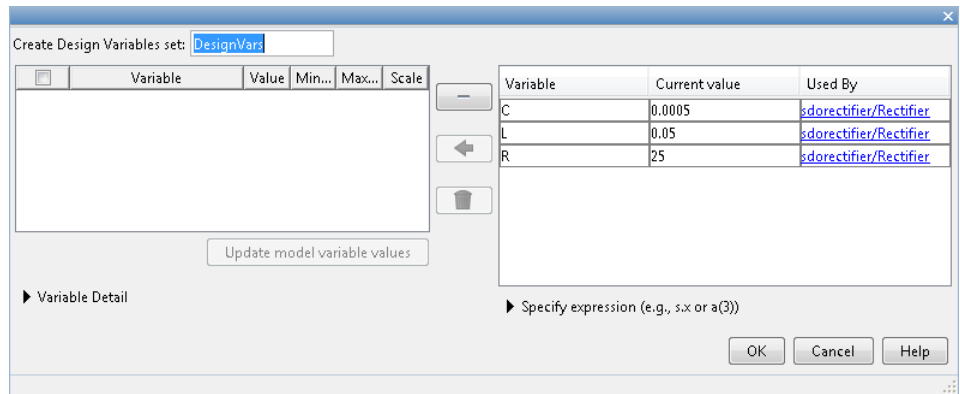
Before you begin this task, specify the design requirements as described in “Specify Design Requirements” on page 3-41.

When you optimize the model response, the software modifies the design variable (parameter) values to meet the design requirements.


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



A window opens where you specify design variables.



2 Click R, L and C to select them.

3 Click  to add the selected parameters to a design variables set.

4 Specify the value range for each design variable:

- R in the range 1–50 ohms
- L in the range 1–500mH
- C in the range 1 μ F–1mF
- Because the variables are different orders of magnitude, specify scaling factors in the corresponding **Scale** column of the variables:
 - R by 25
 - L by 0.05
 - C by 0.0005

Create Design Variables set:

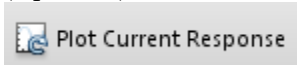
<input checked="" type="checkbox"/>	Variable	Value	Minimum	Maximum	Scale
<input checked="" type="checkbox"/>	C	0.0005	1e-6	1e-3	0.0005
<input checked="" type="checkbox"/>	L	0.05	1e-3	500e-3	0.5
<input checked="" type="checkbox"/>	R	25	1	50	25

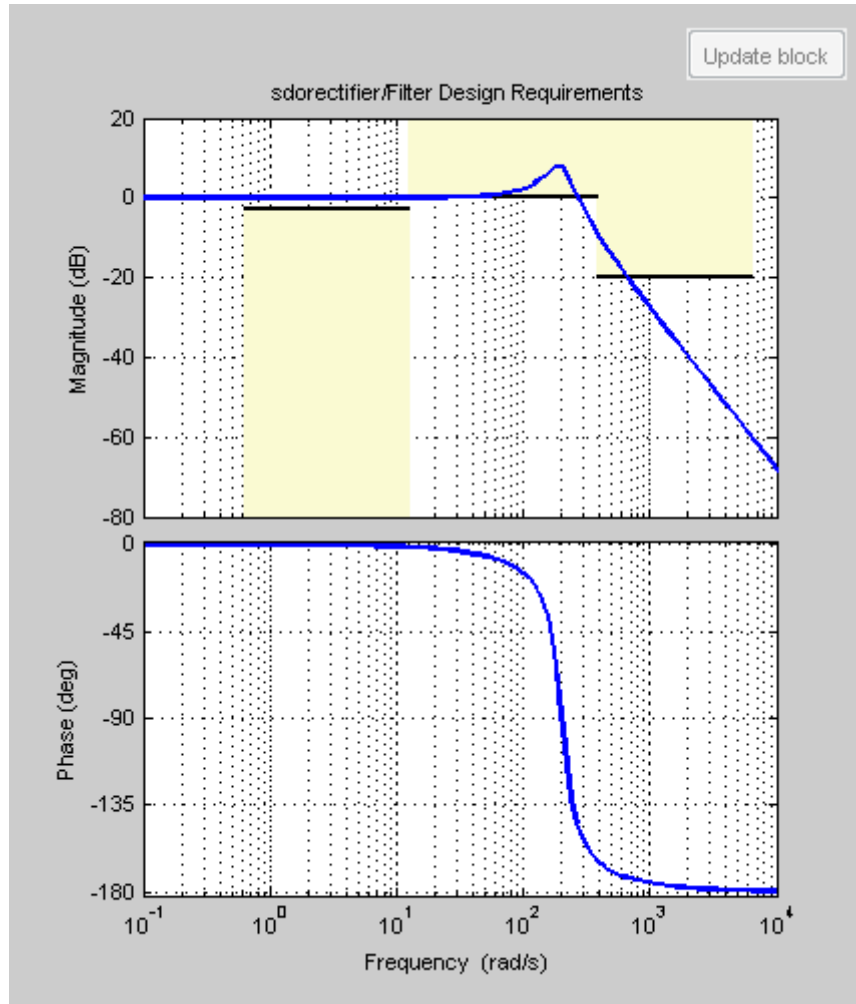
Click **OK**. A new variable `DesignVars` appears in **Design Optimization Workspace** of the Design Optimization tool.

Optimize Design

Before you begin this task, you must have already specified the design requirements and design variables, as described in “Specify Design Requirements” on page 3-41 and “Specify Design Variables” on page 3-44, respectively.

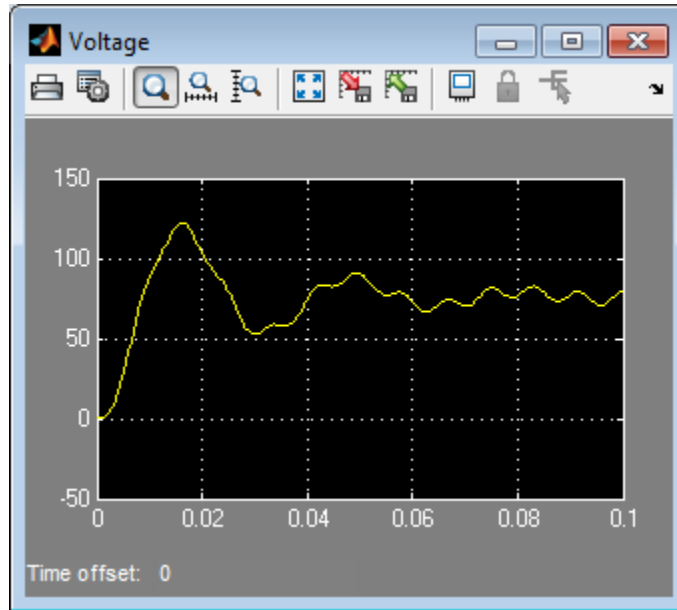
- 1 (Optional) View the current response of the model. Click



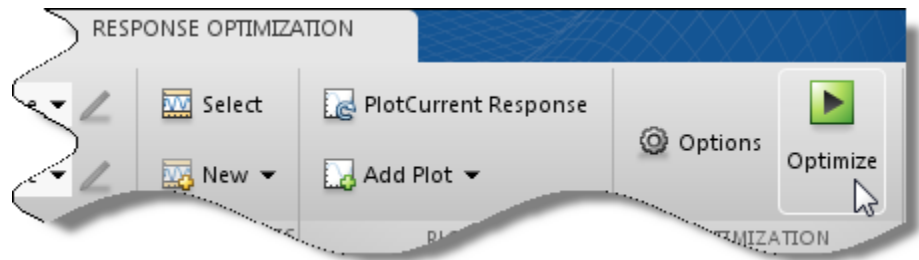


The plot shows that the model does not meet the specified gain and bandwidth requirements.

You also see that the filter voltage signal overshoots its steady-state value and contains significant harmonic content in the Voltage scope window.



2 Click **Optimize**.



An optimization progress window opens.

After the optimization completes, the optimization progress window resembles the next figure. After a few iterations, the optimization converges to satisfy the filter bandwidth requirements.

The screenshot shows a window titled "Optimization Progress" with a table of results and a log of events. The table has five columns: Iteration, F-count, MaxMinRL (min), Filter Design Requireme... (<=0), and Filter Design Requireme... (>=0). The log shows that optimization started on 08-Jan-2013 at 10:07:11 and converged at 10:07:48. The optimized variable values were written to 'DesignVars' in the Design Optimization workspace.

Iteration	F-count	MaxMinRL (min)	Filter Design Requireme... (<=0)	Filter Design Requireme... (>=0)
0	8	1.1000	8.1361	1.0067
1	16	0.3519	-0.8514	-0.8115
2	23	0.0420	0.9941	1.0000
3	31	0.0688	0.2360	0.9654
4	39	0.0794	0.0451	0.9415
5	46	0.0840	0.0056	0.9342
6	53	0.0840	0.0056	0.9342

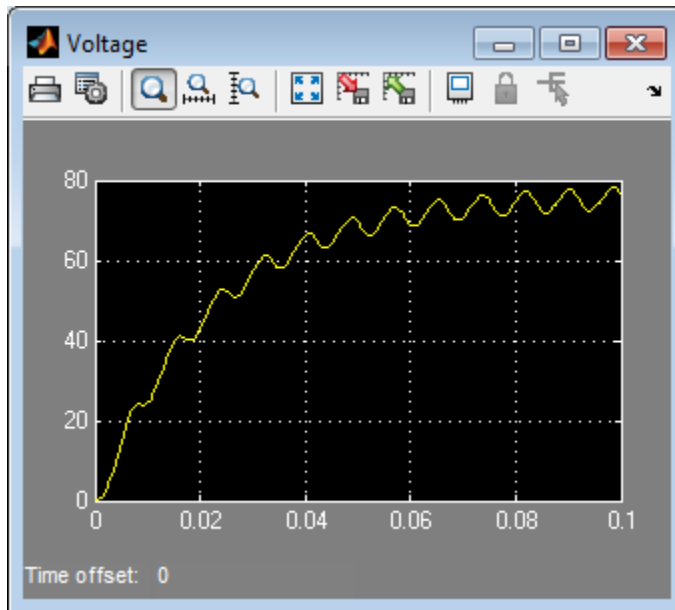
Optimization started 08-Jan-2013 10:07:11

Optimization converged, 08-Jan-2013 10:07:48

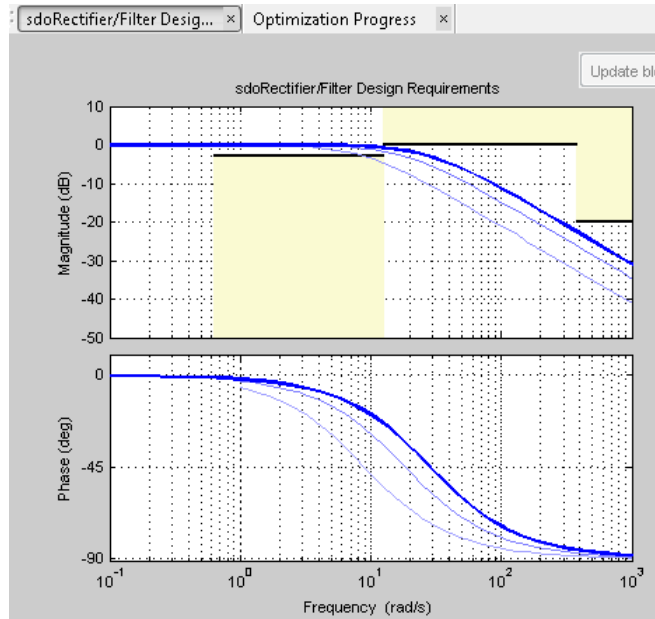
Optimized variable values written to 'DesignVars' in the Design Optimization workspace

Save Iteration... Display Options... Optimize

The filter voltage signal in the Voltage scope window also settles to within its steady-state value in around 0.08 seconds without any overshoot and the harmonic content is significantly reduced from the initial design.



3 Verify that the model meets the gain and bandwidth requirements.



The plot displays the last five iterations. The final response using the optimized parameter values appears as the thick line.

The optimized response lies in the white region bounded by the design requirement line segments and thus meets the requirements.

- 4** Click **DesignVars** in **Design Optimization Workspace** and view the updated values in the **Variable Preview** area.

The optimized values of the design variables are automatically updated in the Simulink model.

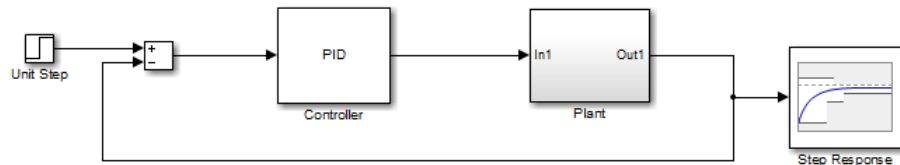
Time-Domain Model Verification

This example shows how to perform time-domain model verification using Simulink Design Optimization **Model Verification** blocks. During time-domain verification, the software monitors a signal to check if it meets time-domain characteristics such as step response characteristics and upper and lower amplitudes, or tracks a reference signal.

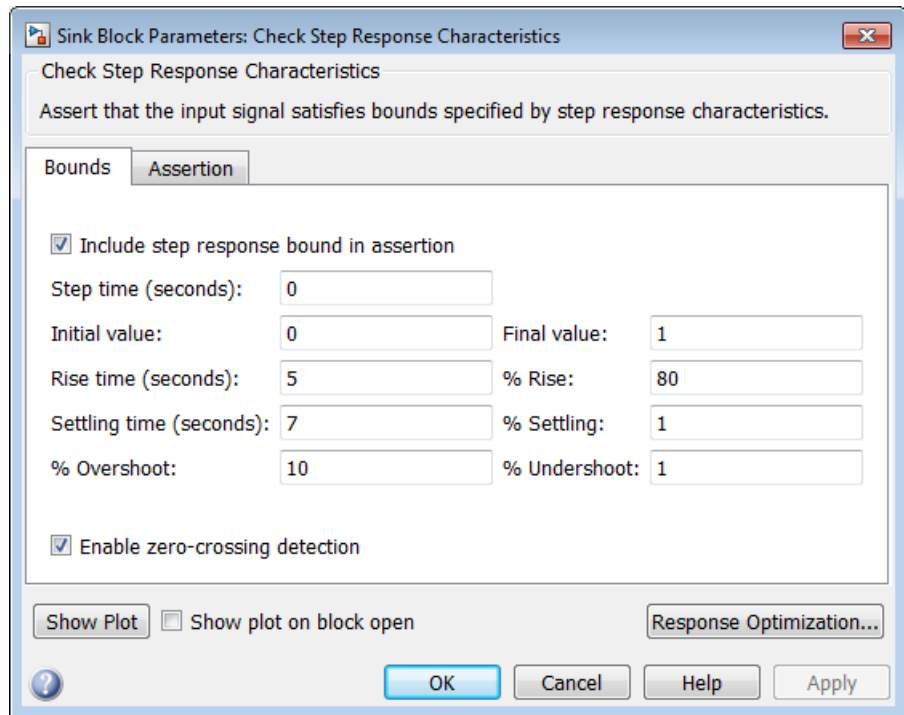
You can also use blocks from Simulink and Simulink Control Design **Model Verification** libraries to design complex assertion logic for time- and frequency-domain verification, and signal monitoring. If you have Simulink Verification and Validation™ software, you can construct simulation tests for your model using the Verification Manager tool in the Signal Builder.

1 Open Simulink model.

```
sys = 'sldo_model1_stepblk';  
open_system(sys);
```



The model includes a Step Response block which is a Check Step Response Characteristics block from the Simulink Design Optimization **Model Verification** library and has default step response bounds.



2 In the Simulink Editor, click **Simulation > Run**.

The block asserts multiple times during simulation because the signal to which the block is connected violates the specified bounds. Assertion warnings appear in the MATLAB command window.

You can optimize model parameters to satisfy the bounds and eliminate assertion warnings. See “Design Optimization to Meet Step Response Requirements (GUI)” on page 3-4.

Optimization-Based Linear Control Design

- “When to Use Optimization-Based Linear Control Design” on page 4-2
- “Types of Time- and Frequency-Domain Design Requirements for Optimization-Based Control Design” on page 4-3
- “Quick Start — Optimization-Based Linear Control Design” on page 4-4
- “Design Optimization-Based PID Controller for Linearized Simulink Model (GUI)” on page 4-15

When to Use Optimization-Based Linear Control Design

When you have Control System Toolbox software installed, you can design and optimize control systems for LTI models by optimizing controller parameters in the SISO Design Tool. To use optimization methods for linear control design, also known as *optimization-based tuning*, you must already have an initial controller. You can then use optimization-based tuning to refine the controller design to meet additional design requirements. For more information on designing controllers, see the Control System Toolbox documentation.

Note Optimization-based tuning only changes the value of the controller parameters and not the controller structure itself.

Optimization-based tuning provides flexibility in terms of specifying additional design requirements for the controller. When you have a large number of design requirements, you can first design an initial controller by selecting a subset of requirements and subsequently select additional requirements to refine the design.

Optimization-based tuning also provides flexibility in terms of selecting a subset of controller parameters to optimize, and specifying bounds on the controller parameters.

To design linear controllers for Simulink models using optimization-based tuning, you must first linearize the model using the Simulink Control Design software. For more information on linearizing Simulink models, see the Simulink Control Design documentation.

Types of Time- and Frequency-Domain Design Requirements for Optimization-Based Control Design

When you design linear controllers for LTI or Simulink models using the Simulink Design Optimization software, you can specify both time- and frequency-domain requirements on the system response. You can specify design requirements on the following plots:

- Root Locus plot
- Open-Loop and Prefilter Bode plots
- Open-Loop Nichols plot
- Step/Impulse Response plots

For more information, see “Time- and Frequency-Domain Requirements in SISO Design Tool”.

Simulink Design Optimization software uses the frequency-domain requirements to compute the frequency response of the system. It then uses optimization methods to reduce the distance between the current response and the requirements by modifying the controller parameters. The software does not change the controller structure when optimizing the controller parameters.

Quick Start – Optimization-Based Linear Control Design

In this quick start, you get an overview of the typical tasks for optimization-based linear control design using the SISO Design Tool:

- 1 Open a SISO Design Tool session.
- 2 Configure a project for optimization-based control design.
- 3 Specify the controller parameters to design.
- 4 Specify the design requirements.
- 5 Design the controller.
- 6 Evaluate the controller design.

Note The same workflow applies to optimization-based control design for LTI models created at the command line using Control System Toolbox software. To learn how to create LTI models, see “Linear (LTI) Models” in the Control System Toolbox documentation.

Prerequisites for optimization-based linear control design include:

- Simulink Compensator Design Task that contains a linearized version of the Simulink model and, optionally, any response plots you configure.

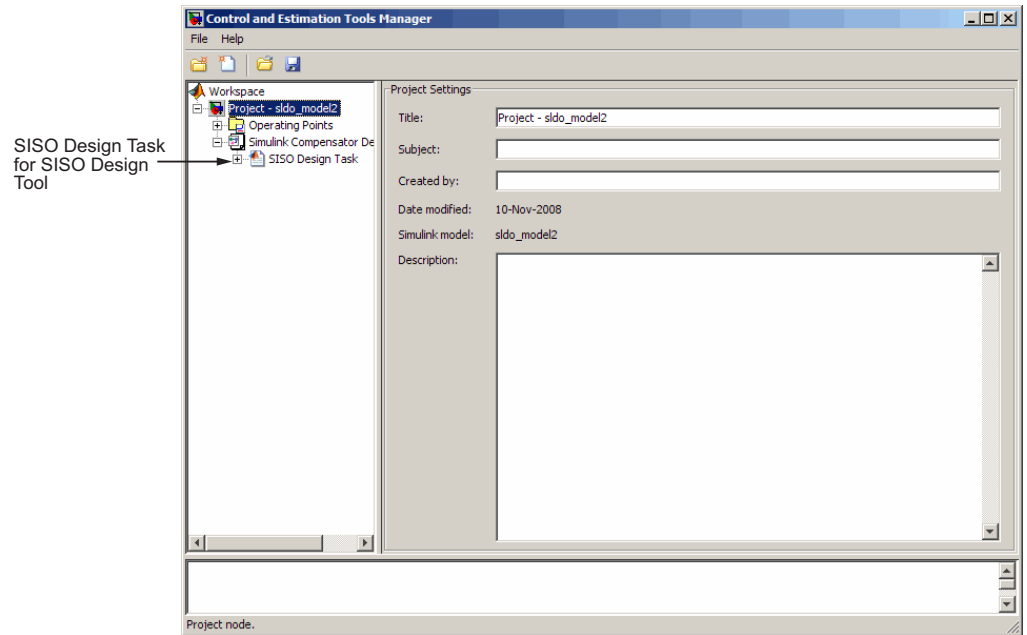
For more information on how to linearize a Simulink model for control design, see the Simulink Control Design documentation.

- Time- and frequency-domain design requirements

To design a controller using optimization methods:

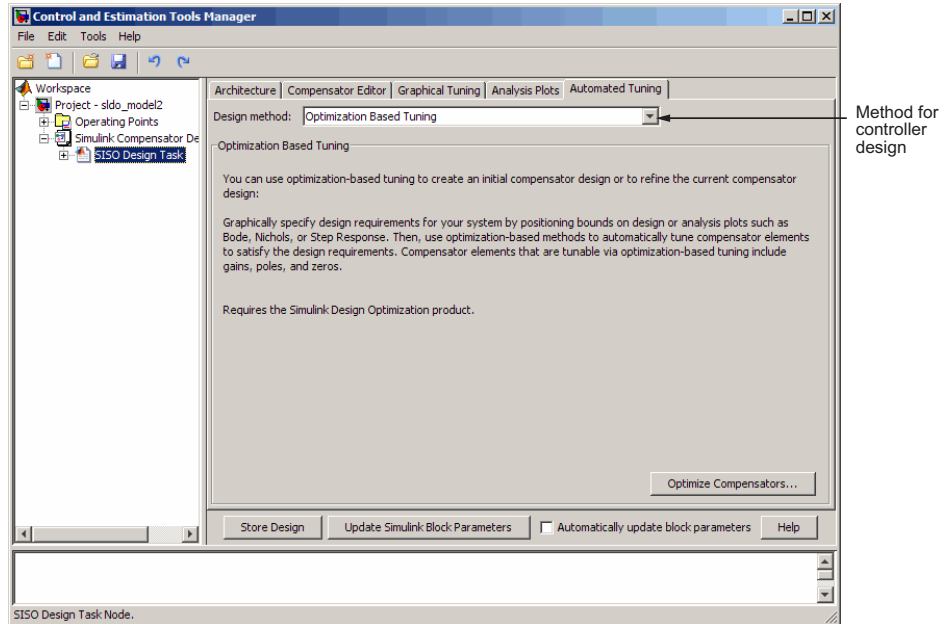
- 1 Open a SISO Design Tool session by typing the following command at the MATLAB prompt:

```
sisotool('projectname.mat')
```

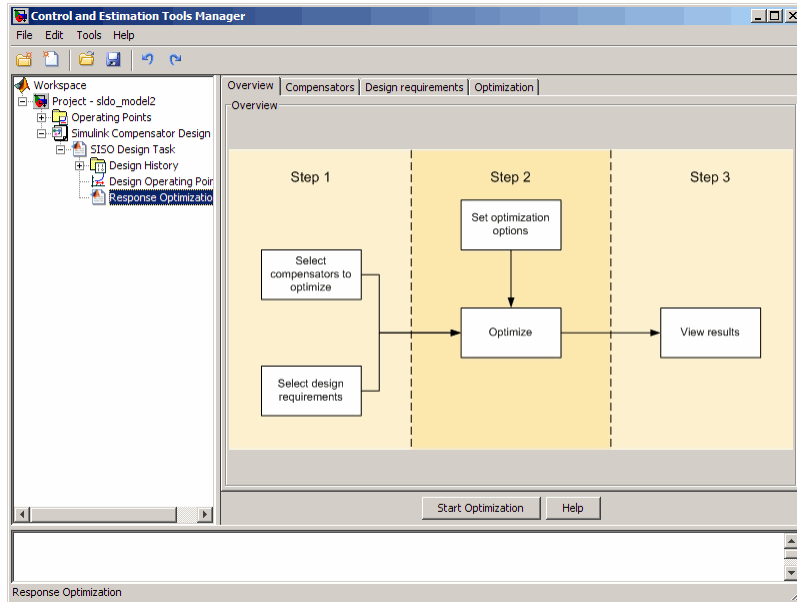


The command also opens a SISO Design for SISO Design Task window by default and any response plots you configured when you linearized the Simulink model using Simulink Control Design software.

- 2 Configure a project for optimization-based control design by clicking **Optimize Compensators** in the Automated Tuning tab of the SISO Design Task.



This action creates a new **Response Optimization** node in the Control and Estimation Tools Manager.



3 Specify the controller parameters to design in the **Compensators** tab.

Select parameters to optimize

Right-click to change controller parameters representation to Simulink block mask parameters

(Optional) Specify initial value

(Optional) Specify parameter bounds

Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical value
<input type="checkbox"/>	sldo_model2/Controller					
<input type="checkbox"/>	Gain	0.1	0.1	-Inf	Inf	0.1
<input type="checkbox"/>	Real Zero	-8.999	-8.999	-Inf	Inf	-8.999
<input type="checkbox"/>	Real Zero	-0.10102	-0.10102	-Inf	Inf	-0.10102
<input type="checkbox"/>	Real Pole	-100	-100	-Inf	Inf	-100

Right click on a compensator name to change its representation.

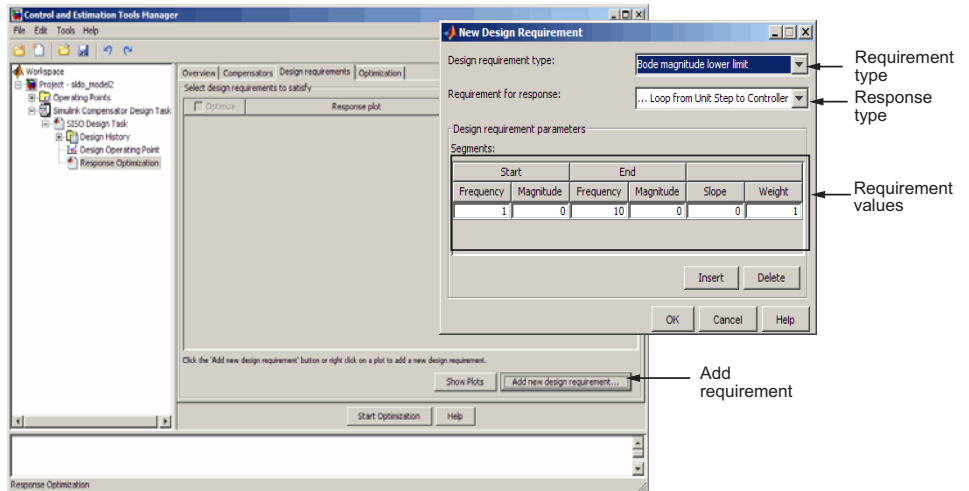
Use Value as Initial Guess

Start Optimization Help

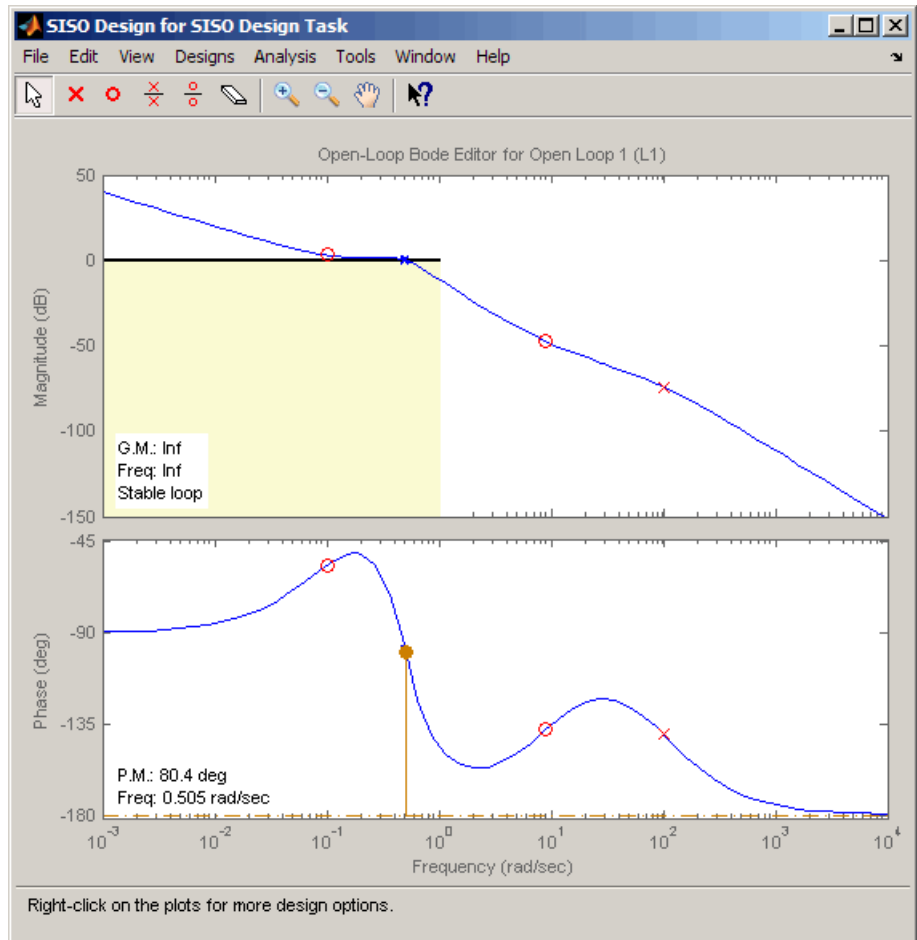
Response Optimization

4 Specify the design requirements.

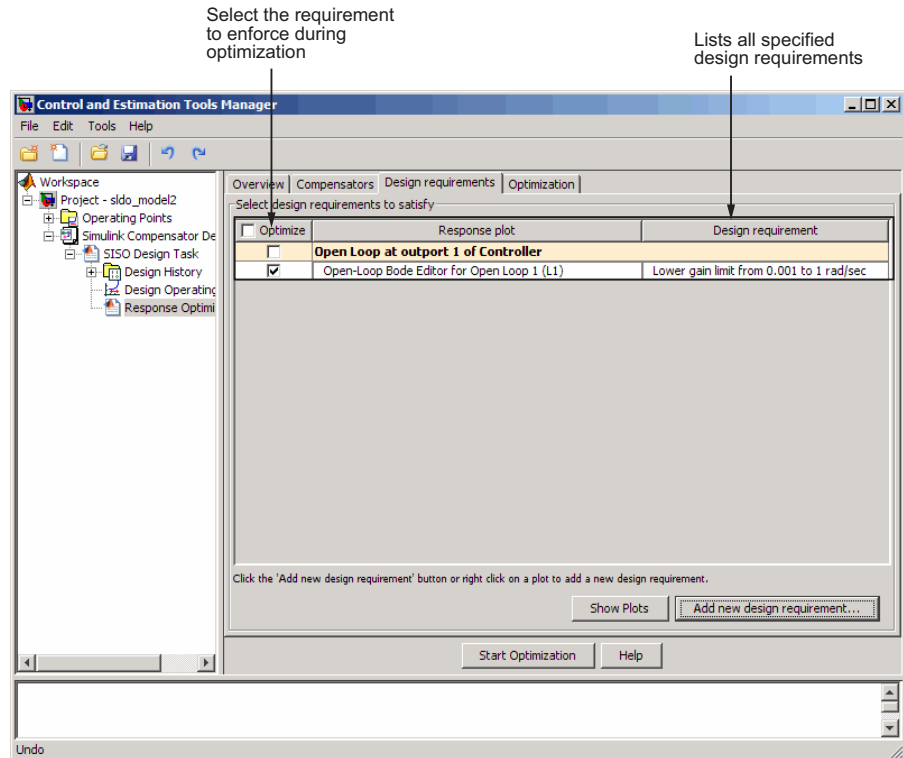
- In the **Design requirements** tab, click **Add new design requirement**. Specify the design requirements, for example Bode magnitude lower limit, in the New Design Requirement dialog box.



In the SISO Design window, the yellow region with the black line segment represents the design requirement on the response plot.



The **Design Requirements** tab also lists the design requirement.



- b** Repeat step a to specify additional time- and frequency-domain requirements.

- 5 Start the optimization to design the controller by clicking **Start Optimization** in the **Optimization** tab.

The screenshot displays the 'Control and Estimation Tools Manager' window. The 'Optimization' tab is active, showing a table of optimization progress and a status window. The table has the following data:

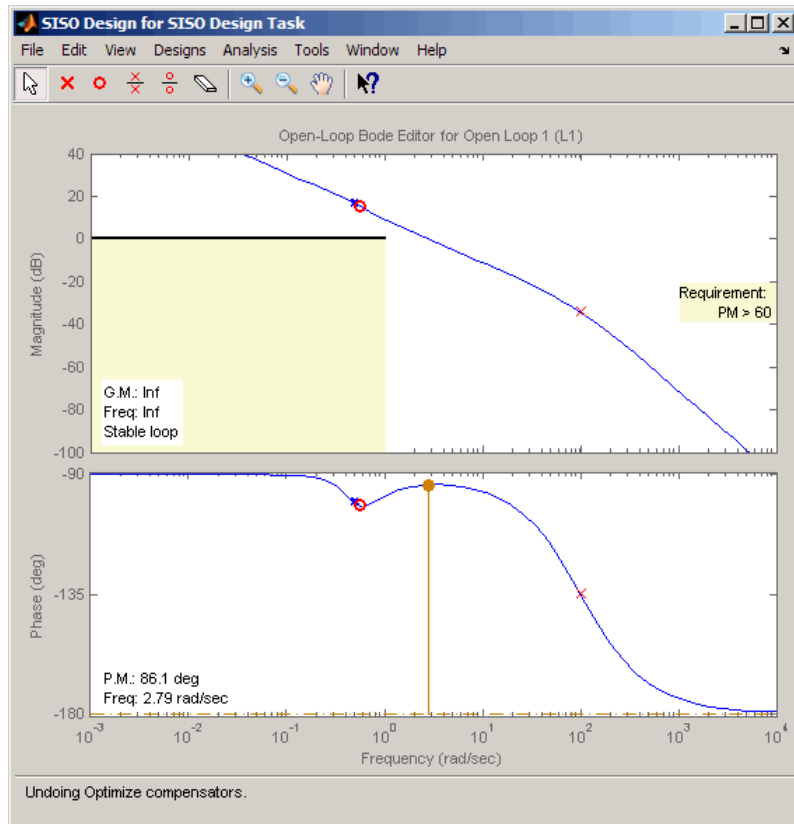
Iteration	Eval-Count	Cost function	Constraint...	Step size	Procedure
0	7	0	57.12		
1	14	0	1.777	8.36	
2	21	0	1.697	12.1	Hessian m...
3	28	0	0	21.7	

The status window at the bottom contains the following text:

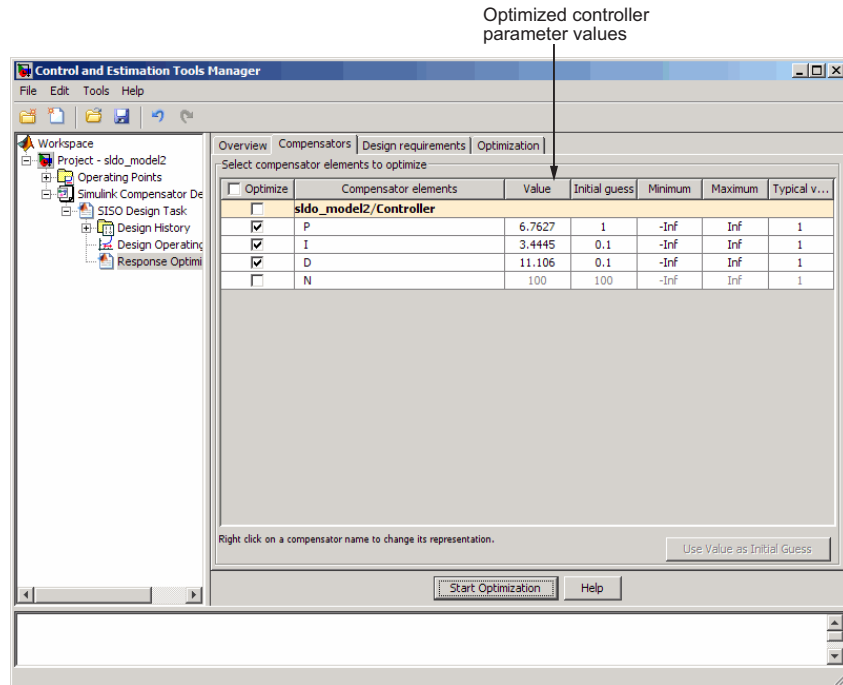
```
Constructing optimization problem...
Optimization started 10-Nov-2008 14:01:11
Optimization finished 10-Nov-2008 14:01:21
Successful termination. Found a feasible or optimal solution within the specified
tolerances.
```

Arrows on the right side of the image point to the table and the status window, labeled 'Optimization progress' and 'Optimization status' respectively. The 'Start Optimization' button is visible at the bottom of the window.

- 6 Evaluate the controller design.
 - a Examine the system's response in the response plot, for example the Bode plot, to see if it meets the requirements. The system's response must lie in the white region in order to meet the design requirement.



- b** Examine the controller parameter values in the **Compensator** tab.



- 7** Write the controller parameter values into the Simulink model. To do so, click **Update Simulink Block Parameters** in the **SISO Design Task** node.

See Also: “Design Optimization-Based PID Controller for Linearized Simulink Model (GUI)” on page 4-15.

Design Optimization-Based PID Controller for Linearized Simulink Model (GUI)

In this section...

“About This Tutorial” on page 4-15

“Configuring a Project for Optimization-Based Control Design” on page 4-17

“Designing an Initial PID Controller to Meet Bode Magnitude and Phase Margins Requirements” on page 4-22

“Refining the Controller Design to Meet Controller Output Bounds” on page 4-43

“Saving the Project” on page 4-59

About This Tutorial

- “Objectives” on page 4-15
- “About the Model” on page 4-15
- “Design Requirements” on page 4-16

Objectives

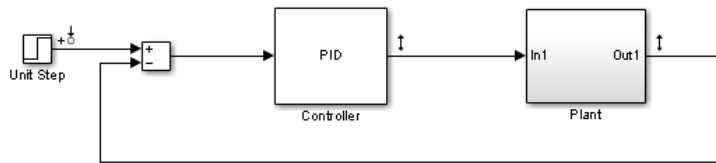
In this tutorial, you learn to use optimization methods to design a PID controller to meet frequency-domain design requirements on a system’s response.

You accomplish the following tasks using the GUI:

- Specify frequency-domain Bode magnitude and phase margin requirements.
- Design an initial controller to meet the frequency-domain requirements.
- Refine the initial controller design to limit the controller’s output signal.

About the Model

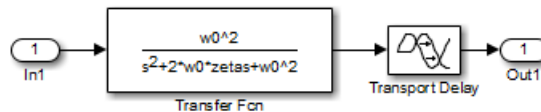
In this tutorial, you use the Simulink model named `s1do_model12`, as shown in the next figure.



The model contains a `Controller` block, which is a PID Controller. This block controls the output of the `Plant` subsystem.

Using the Simulink Control Design software has linearized the Simulink model at the operating point specified in the model. The `sldo_model2.mat` file contains a preconfigured SISO Design Tool session saved after linearizing the model. To learn more about linearizing Simulink models for control design, see “Control Design”.

Double-click the `Plant` subsystem to open it. The plant is modeled as a second-order system with delay. It contains `Transfer Function` and `Transport Delay` blocks, as shown in the next figure.



To learn more about the blocks, see `Transfer Fcn` and `Transport Delay` block reference pages.

Design Requirements

The compensator you design in this tutorial must meet the following design requirements:

- Bode lower magnitude bound of 0 in the frequency range 1e-3 to 1 rad/sec
- Phase margin greater than 60 degrees
- Controller output bounds in the range [-250 550]

Configuring a Project for Optimization-Based Control Design

To design a linear controller for a Simulink model, you must first configure a Control and Estimation Tools Manager project.

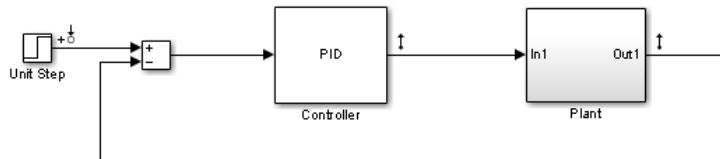
- 1 Open a SISO Design Tool session for the linearized Simulink model by typing the following command at the MATLAB prompt:

```
sisotool('sldo_model2.mat')
```

Note sldo_model2.mat file contains a preconfigured SISO Design Tool session. This session was saved after Simulink Control Design software linearized the sldo_model2 Simulink model.

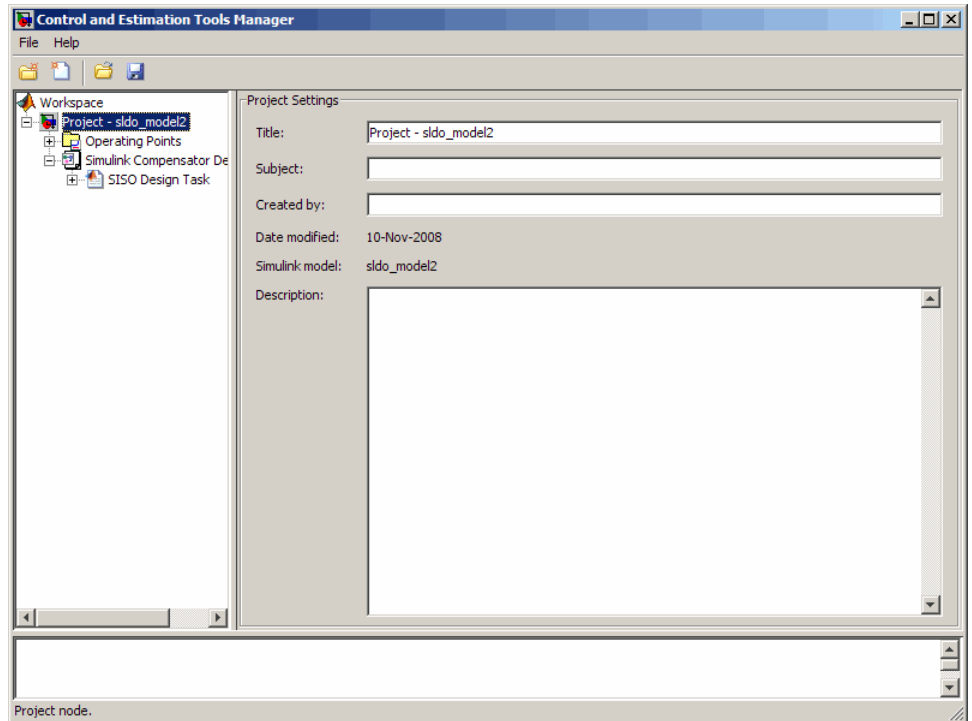
This command opens the following windows:

- Simulink model

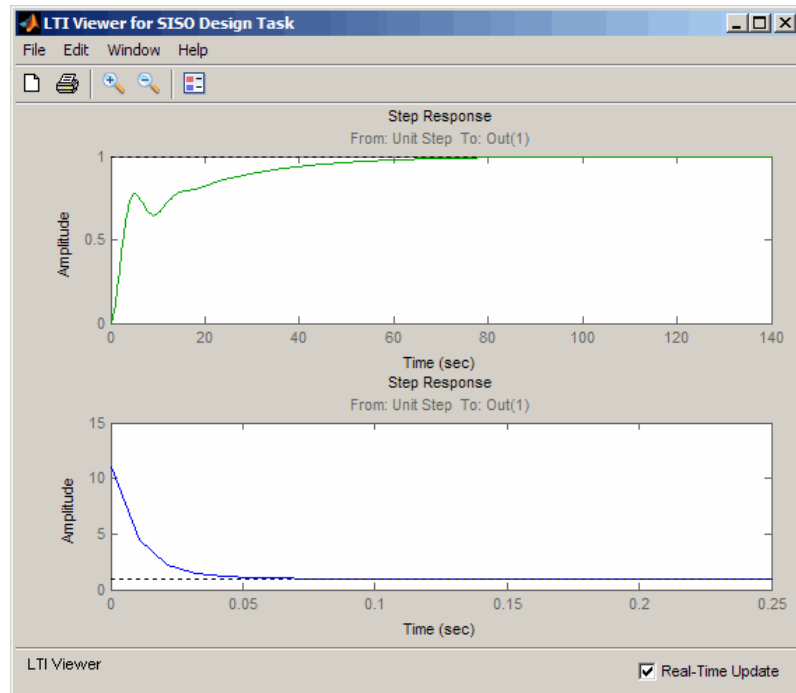


To learn more about the model, see “About the Model” on page 4-15.

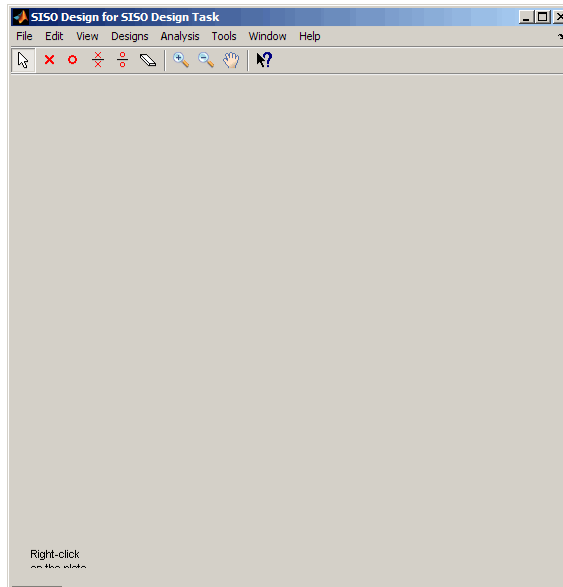
- Control and Estimation Tools Manager GUI, which contains a **SISO Design Task** node under the **Simulink Compensator Design Task** node.



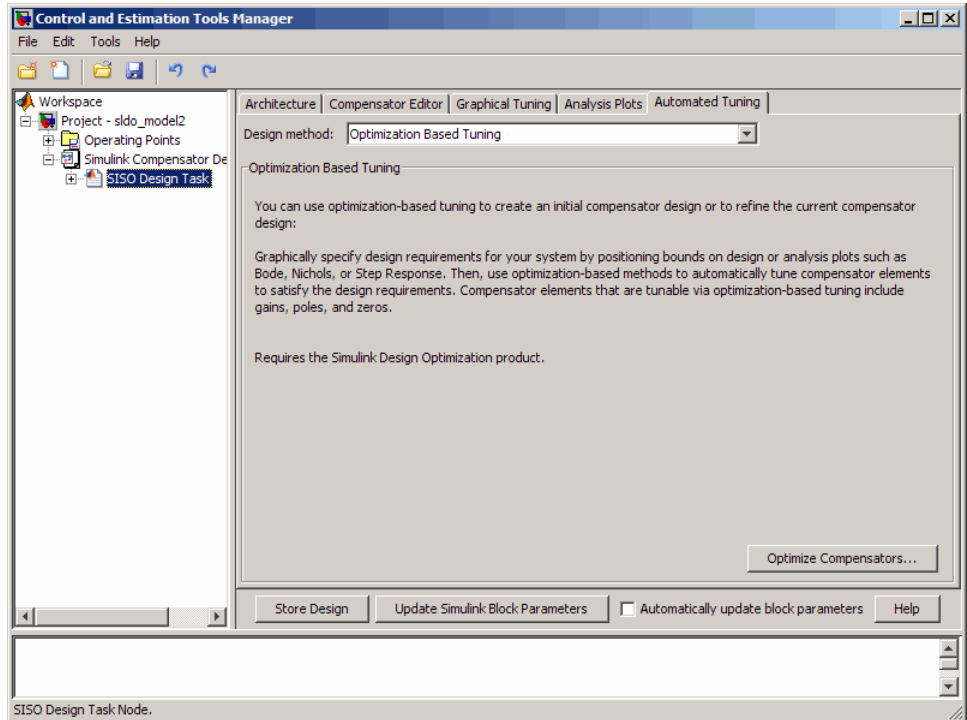
- LTI Viewer for SISO Design Task window, which contains the following plots:
 - Closed-loop step response of the system in the top plot
 - Output of the Controller block in the bottom plot



- A blank SISO Design for SISO Design Task window

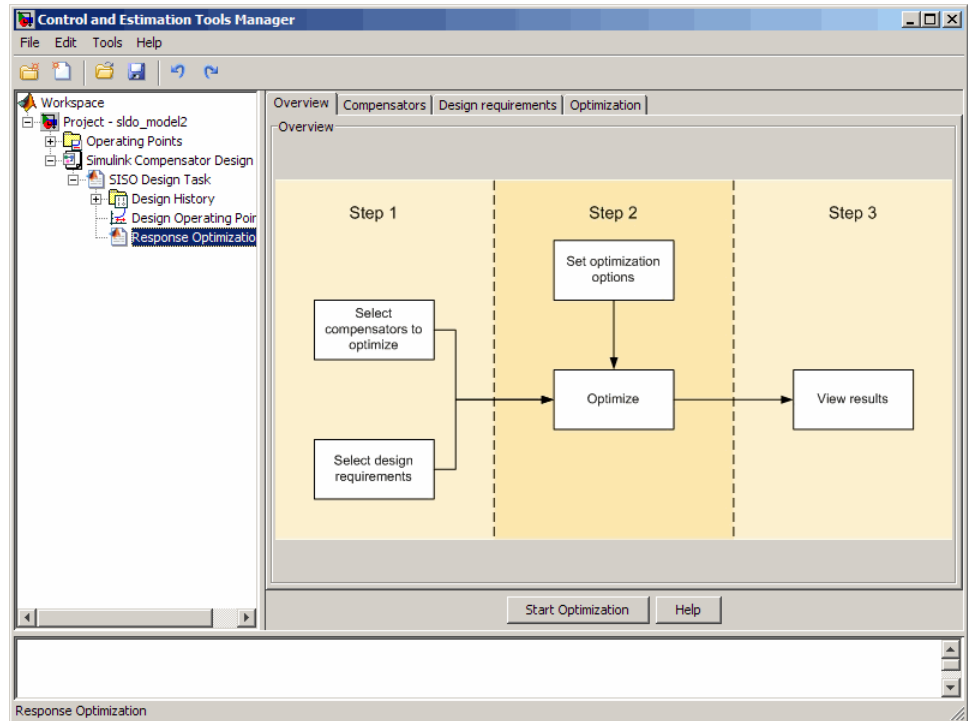


- 2 In the Control and Estimation Tools Manager GUI, select the **Automated Tuning** tab in the **SISO Design Task** node.



3 Click **Optimize Compensators**.

This action creates a new **Response Optimization** node.



Designing an Initial PID Controller to Meet Bode Magnitude and Phase Margins Requirements

- “Specifying the Controller Parameters” on page 4-22
- “Specifying Bode Magnitude and Phase Margin Design Requirements” on page 4-27
- “Designing the Controller” on page 4-37

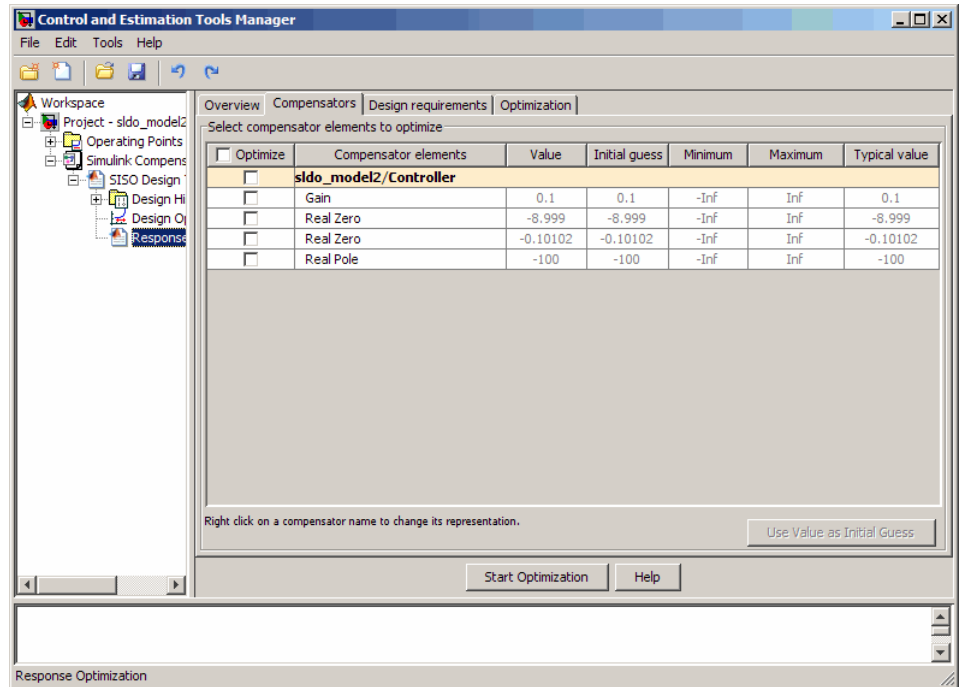
Specifying the Controller Parameters

In this portion of the tutorial, you specify the controller parameters to design.

You must have already configured a project for control design, as described in “Configuring a Project for Optimization-Based Control Design” on page 4-17.

To specify the controller parameters to design:

- 1 In the Control and Estimation Tools Manager GUI, select the **Compensators** tab in the **Response Optimization** node.

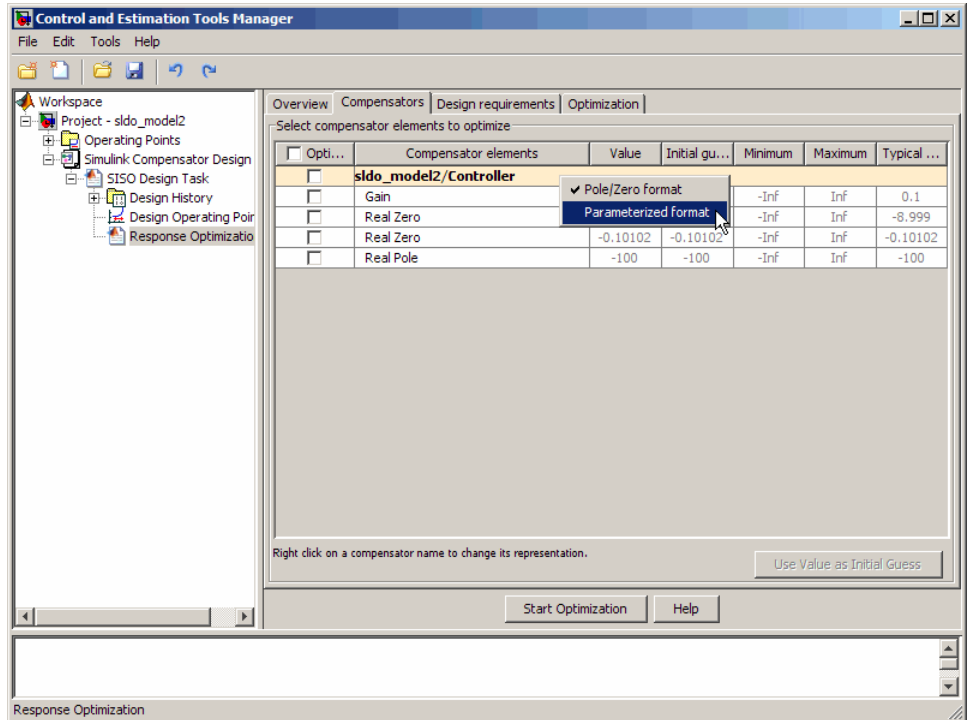


The controller parameters appear as poles and zeros in the **Compensator elements** column and represent the following:

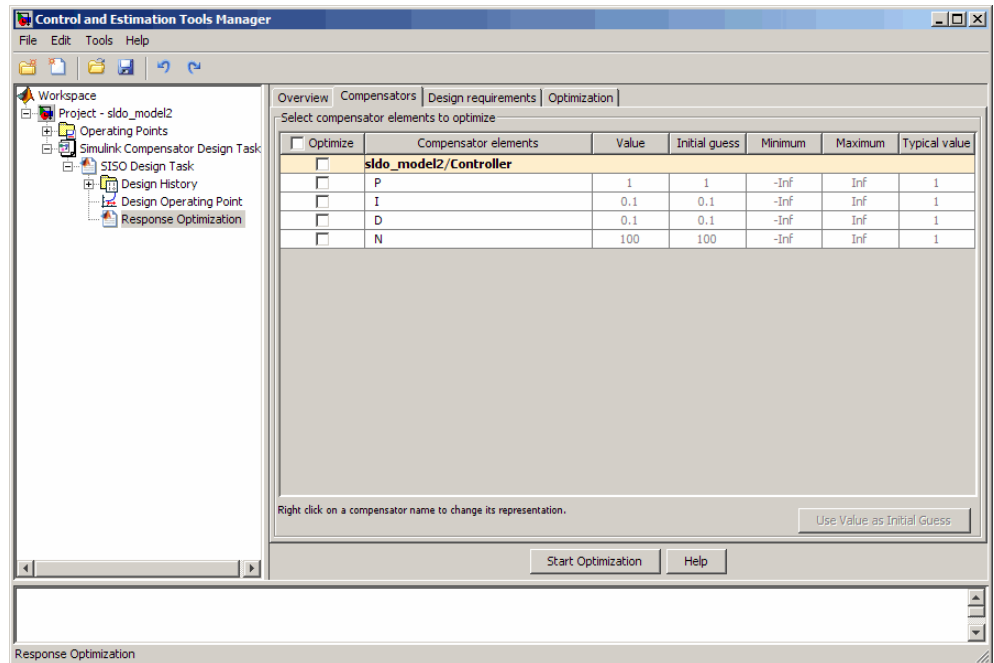
- Gain — Overall gain of the controller
- Real zeros — Zeros resulting from the differentiator and integrator
- Real pole — Pole resulting from the low-pass filter of the differentiator

Tip To view the structure of the Controller block, right-click the block and select **Mask > Look Under Mask**.

- 2 For convenience, change the PID controller parameters to Simulink block mask parameters format. To do so, right-click the **sldo_model2/Controller** column, and select **Parameterized format**.



This action displays the controller parameters as Simulink block mask parameters P, I, and D, as shown in the next figure. To learn more about mask parameters, see “Mask Parameters” in the Simulink documentation.

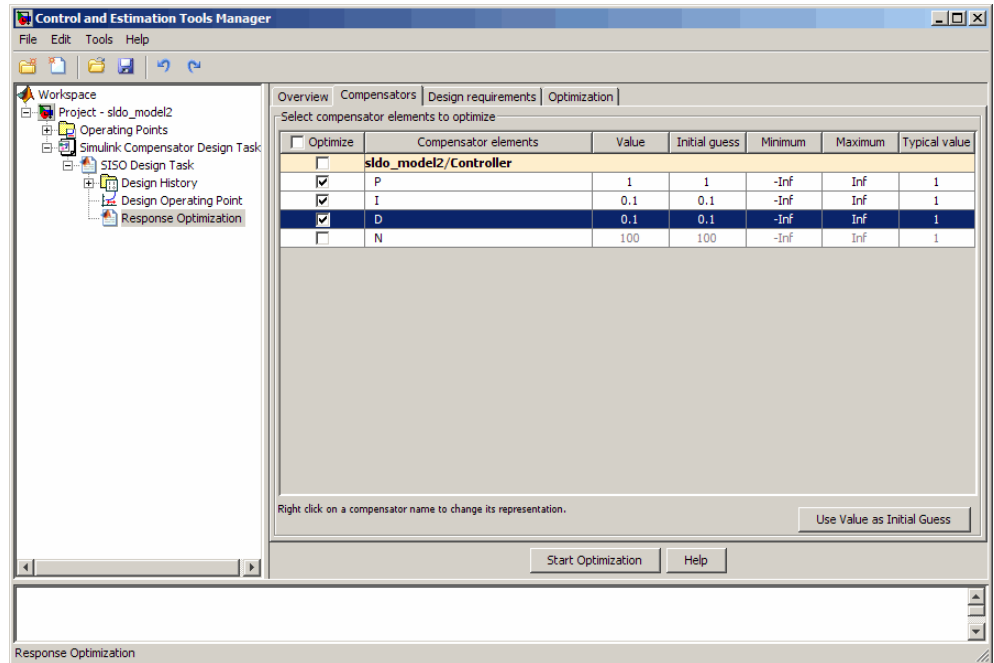


The **Compensators** tab displays the following parameter settings:

- **Value** — Current controller parameter value
- **Initial Guess** — Initial controller parameter value
- **Minimum** and **Maximum** — Controller parameter bounds
- **Typical Value** — Scaling factor for the controller parameter

Note Compensator elements or parameters cannot have uncertainty when used with frequency-domain based response optimization.

3 In the **Optimize** column, select the check boxes for P, I, and D.



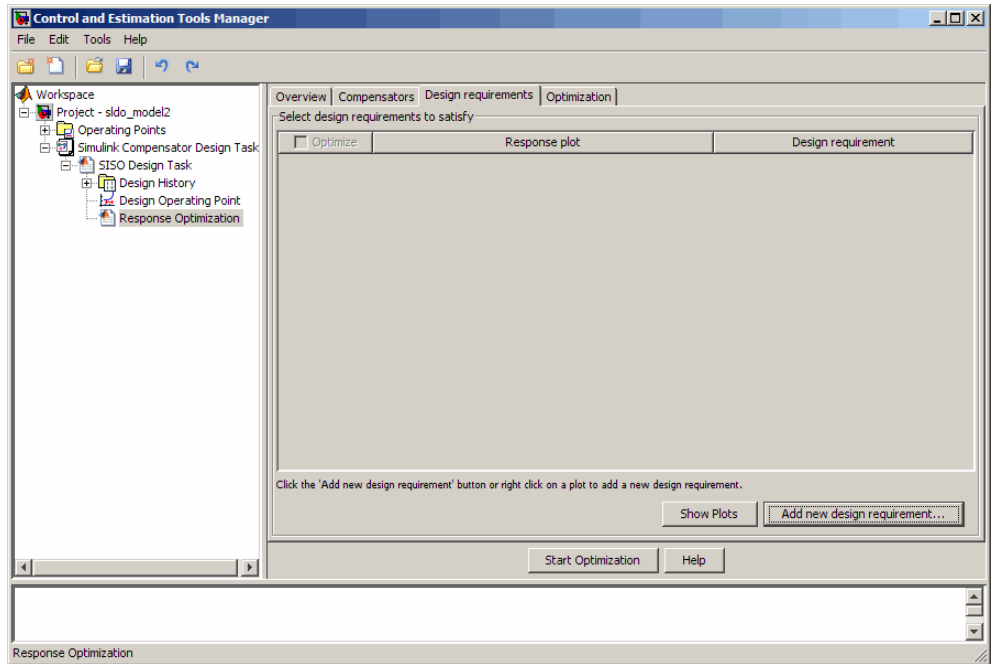
Specifying Bode Magnitude and Phase Margin Design Requirements

In this portion of the tutorial, you specify the Bode magnitude and phase margin requirements that the controller must satisfy.

Before you specify the design requirements, you can specify the controller parameters to design, as described in “Specifying the Controller Parameters” on page 4-22.

To specify the Bode design requirements:

- 1 In the **Response Optimization** node of the Control and Estimation Tools Manager GUI, select the **Design requirements** tab.



2 Click Add new design requirement.

This action opens the New Design Requirement dialog box.

New Design Requirement

Design requirement type: **Step response bounds**

Requirement for response: **Closed Loop from Unit Step to Plant**

Design requirement parameters

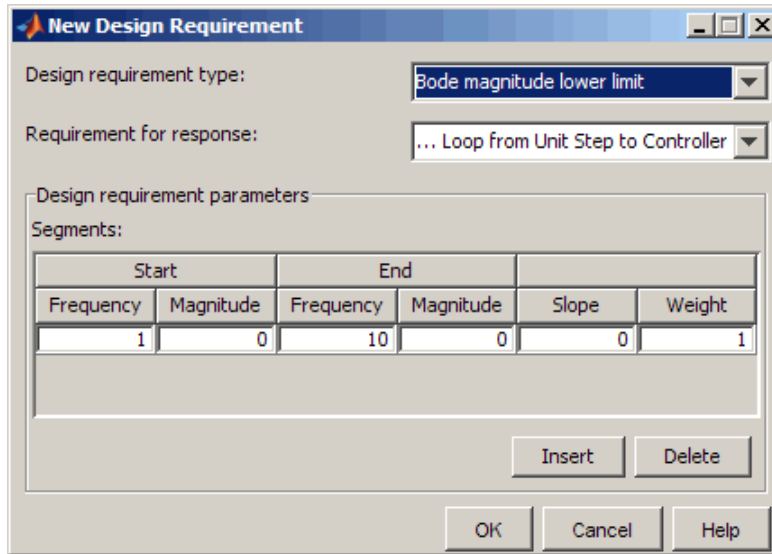
Initial value:	<input type="text" value="0"/>	Final value:	<input type="text" value="1"/>
Step time:	<input type="text" value="0"/>		
Rise time:	<input type="text" value="5"/>	% Rise:	<input type="text" value="80"/>
Settling time:	<input type="text" value="10"/>	% Settling:	<input type="text" value="1.0000"/>
% Overshoot:	<input type="text" value="10.0000"/>	% Undershoot:	<input type="text" value="1"/>

OK Cancel Help

3 Specify the Bode magnitude lower limit requirement:

- a In the New Design Requirement dialog box, select Bode magnitude lower limit from the **Design requirement type** drop-down list.

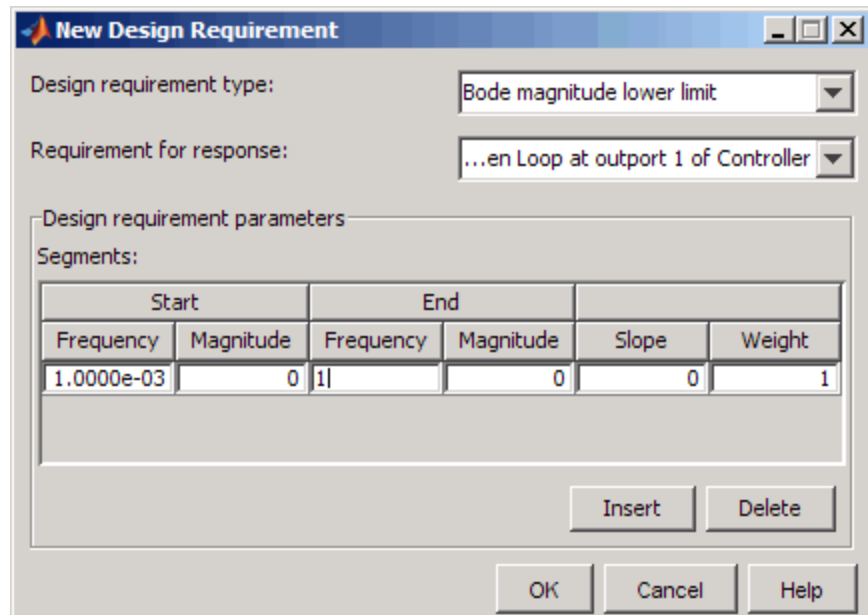
This action updates the New Design Requirement dialog box, as shown in the next figure.



- b Select Open Loop at output 1 of Controller from the **Requirement for response** drop-down list.
- c In the **Frequency** field of the **Start** column, enter $1e-3$.

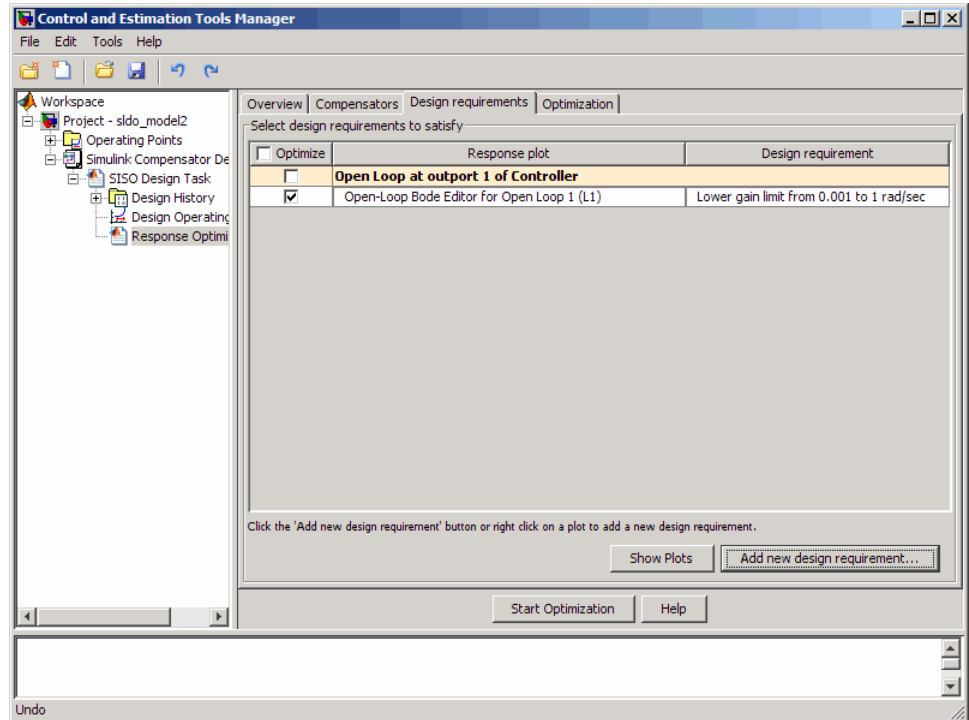
- d** In the **Frequency** field of the **End** column, enter 1.

The New Design Requirement dialog box resembles the following figure.

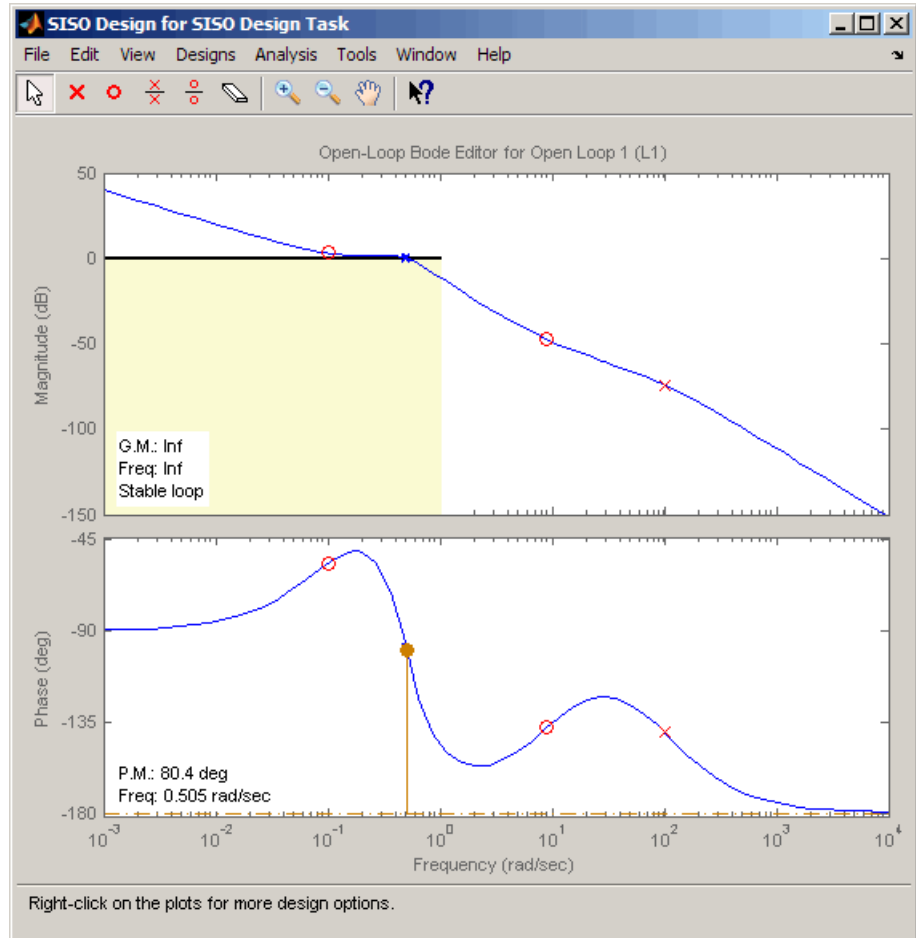


- e** Click **OK** to close the New Design Requirement dialog box.

The Bode lower magnitude limit is added to the **Design requirements** tab, as shown in the next figure.

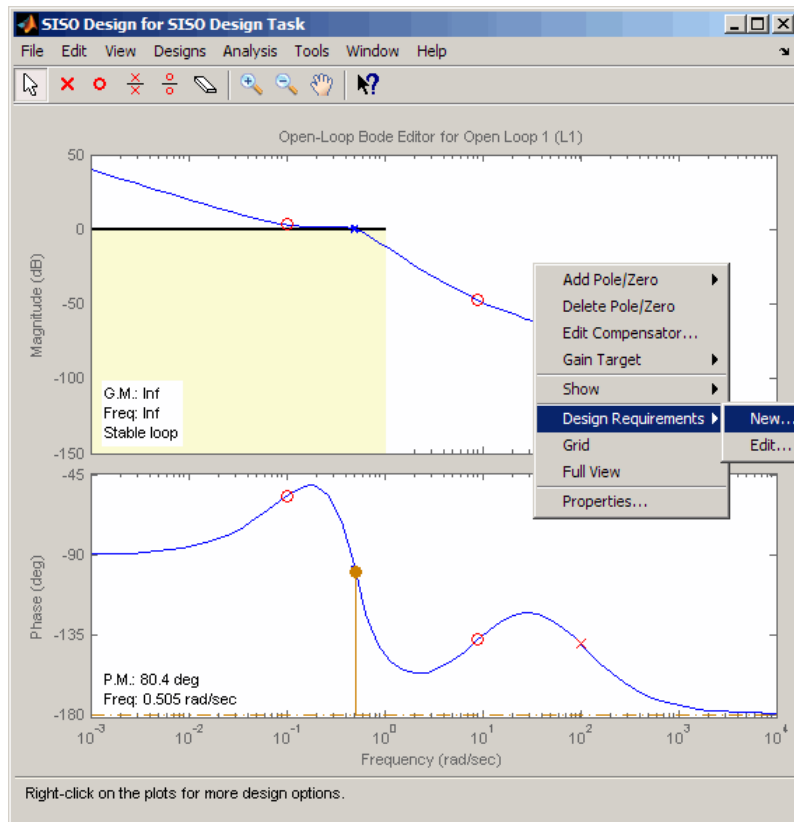


The SISO Design for SISO Design task window also updates to show the Bode plot with the design requirement displayed as the black line segment.

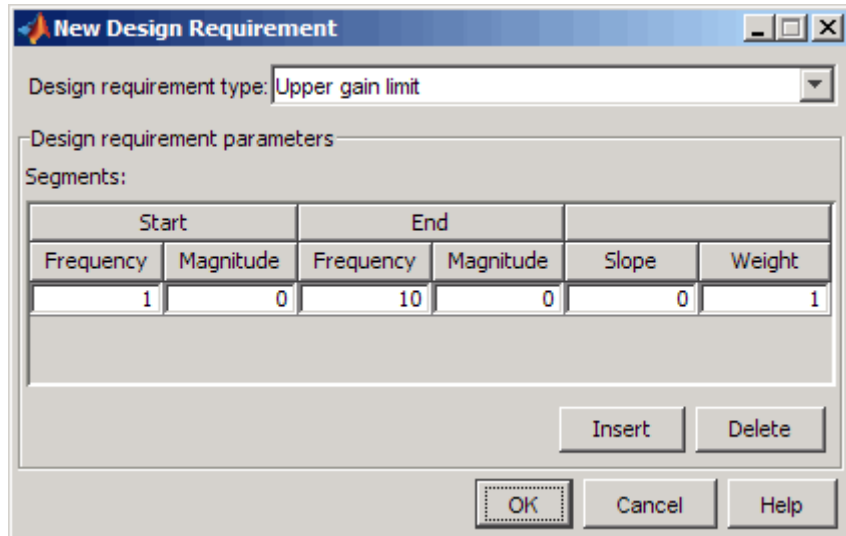


4 Add the phase margin requirement:

- In the SISO Design window, right-click the white area on the top plot, and select **Design requirement** > **New**.

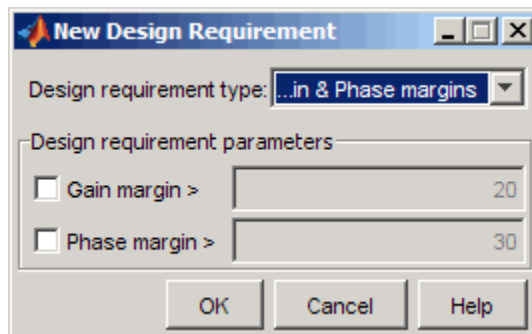


This action opens the New Design Requirement dialog box, as shown in the next figure.

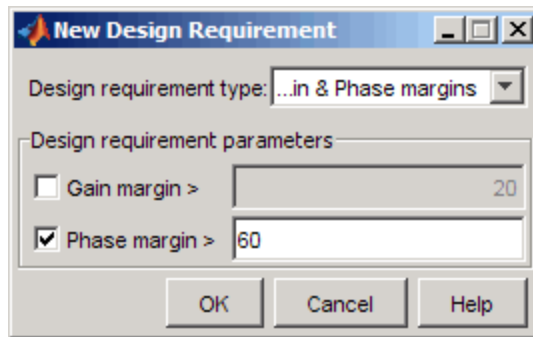


- b** In the New Design Requirement dialog box, select **Gain & phase margins** from the **Design requirement type** drop-down list.

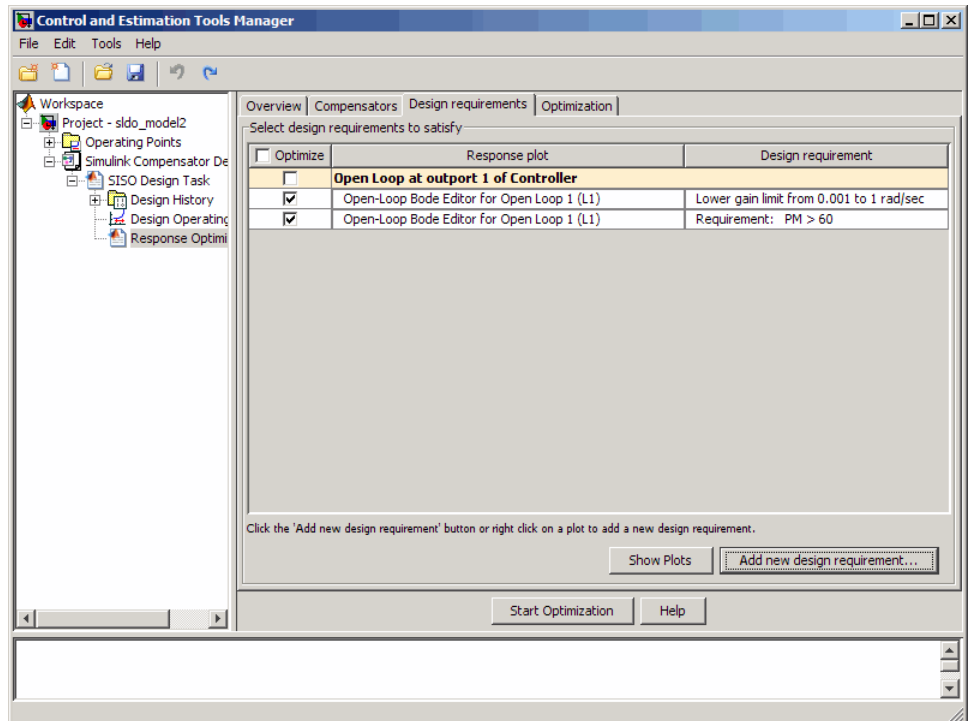
The New Design Requirement dialog box updates to display the **Gain Margin >** and **Phase Margin >** options, as shown in the next figure.



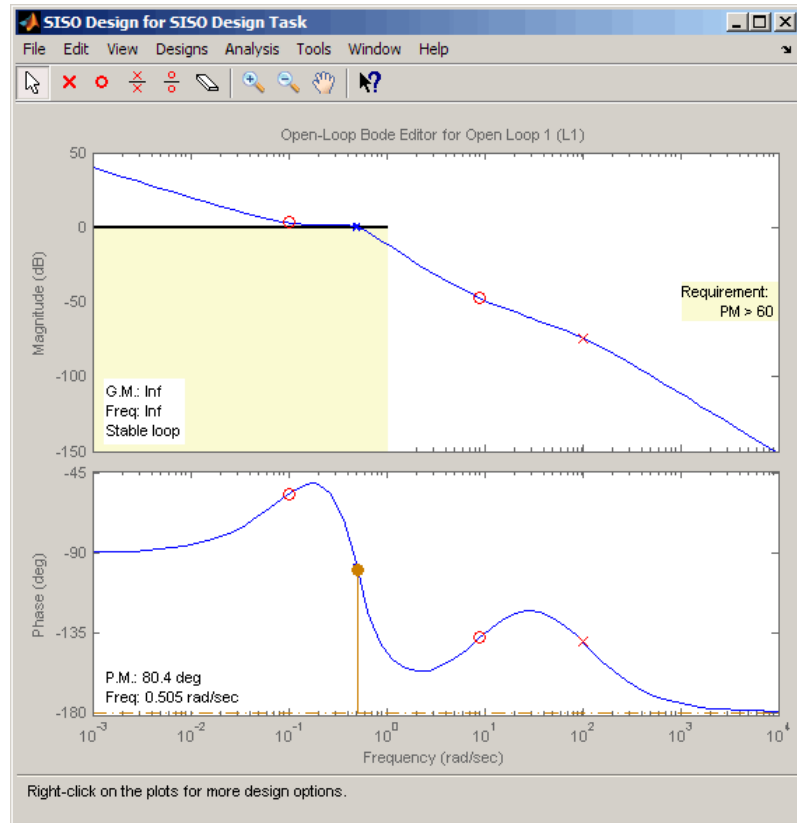
- c** Select the **Phase margin >** check box, and enter 60 in the adjacent field. Then, click **OK** to close the dialog box.



The **Design requirements** tab in the Control and Estimation Tools Manager GUI updates. It now displays the phase margin requirement, as shown in the next figure.



The SISO Design window also updates to display the phase margin requirement.



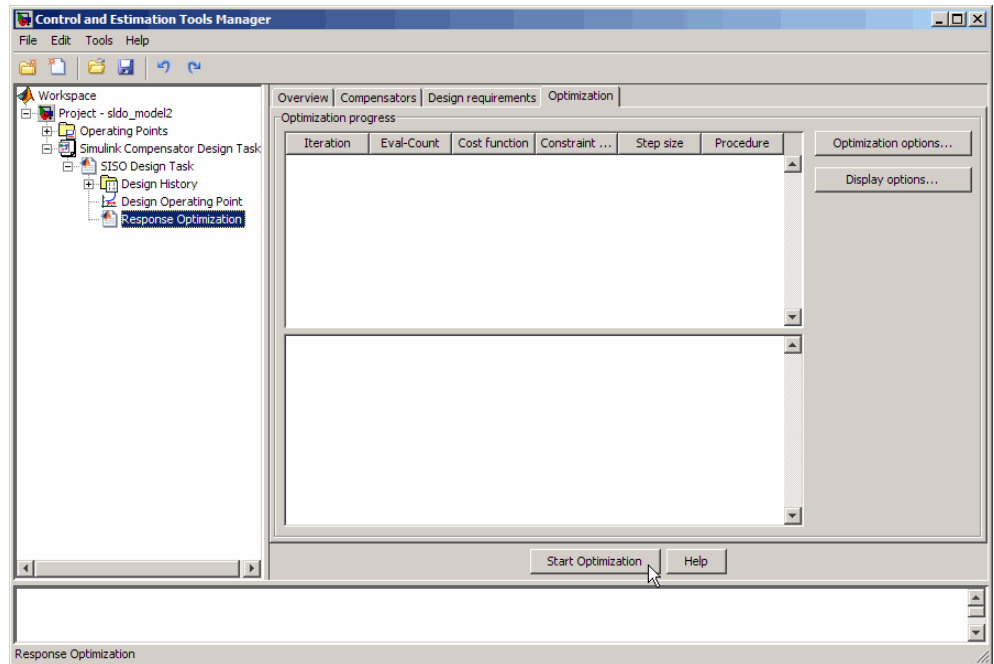
Designing the Controller

In this portion of the tutorial, you design the controller to meet the Bode magnitude and phase requirements.

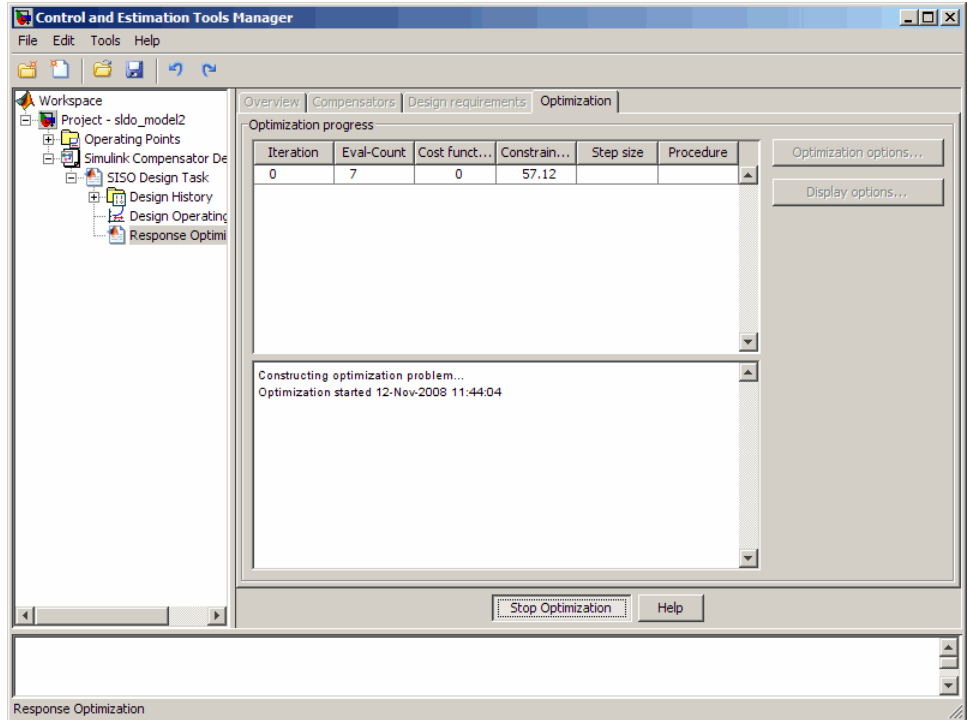
You must have already specified the controller parameters to design, as described in “Specifying the Controller Parameters” on page 4-22, and the design requirements, as described in “Specifying Bode Magnitude and Phase Margin Design Requirements” on page 4-27.

To design the controller:

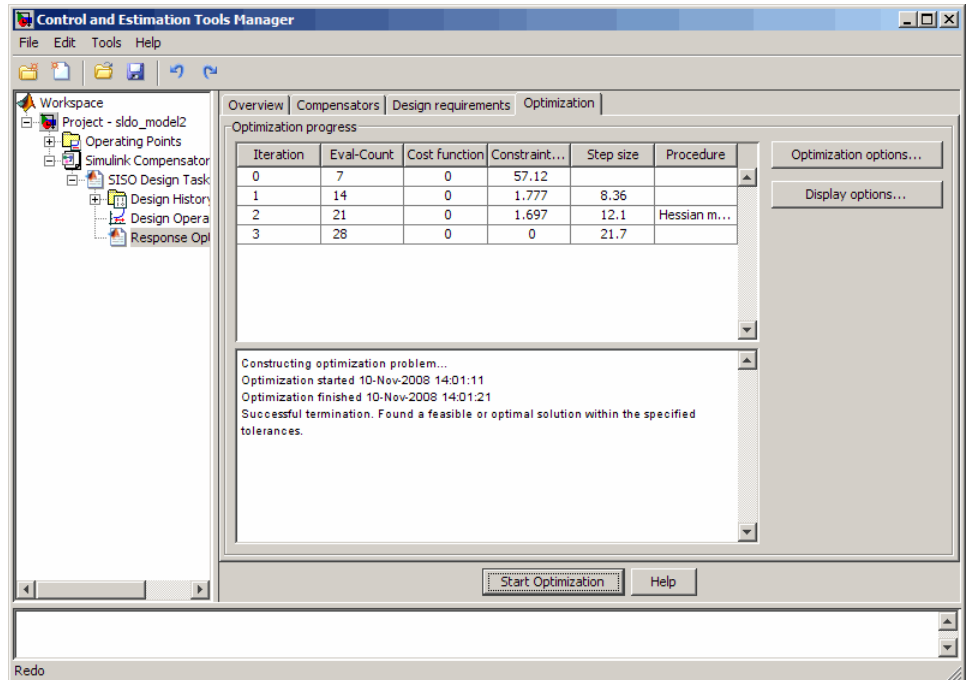
- 1 In the **Optimization** tab, click **Start Optimization**.



The **Optimization** tab updates as shown in the next figure.

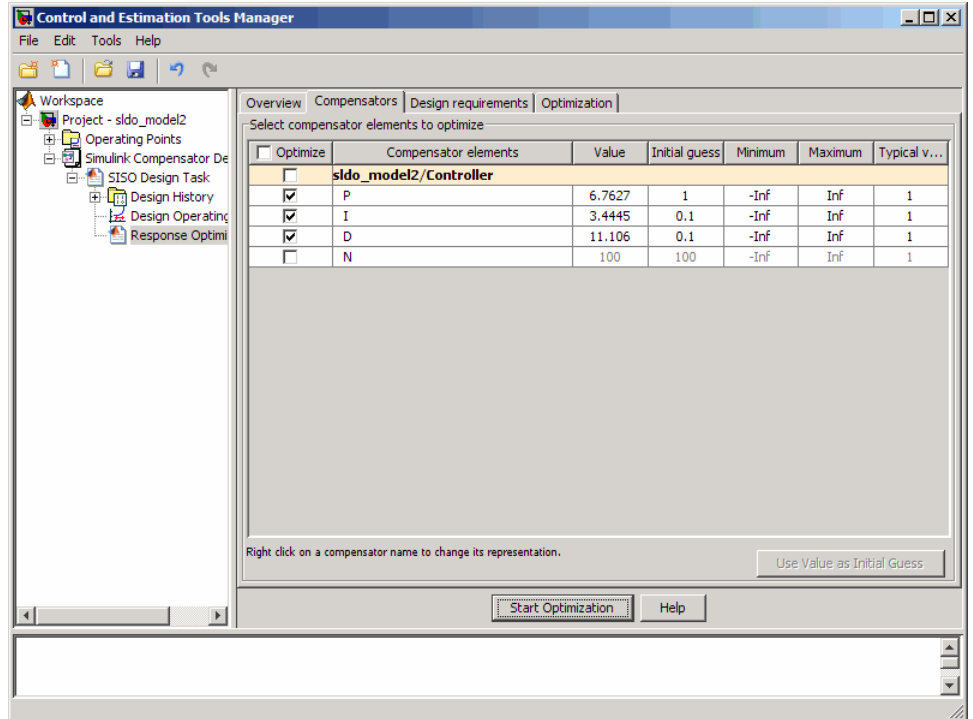


- At every optimization iteration, the default optimization method **Gradient descent** reduces the distance between the current response and the magnitude requirement line segment by modifying the controller parameters. Simultaneously, the software also computes the phase margin and reduces the distance between the current response and the phase margin. To learn more about the optimization method, see “Selecting Optimization Methods”.
- After the optimization completes, the **Optimization** tab displays the optimization iterations and status, as shown in the next figure.



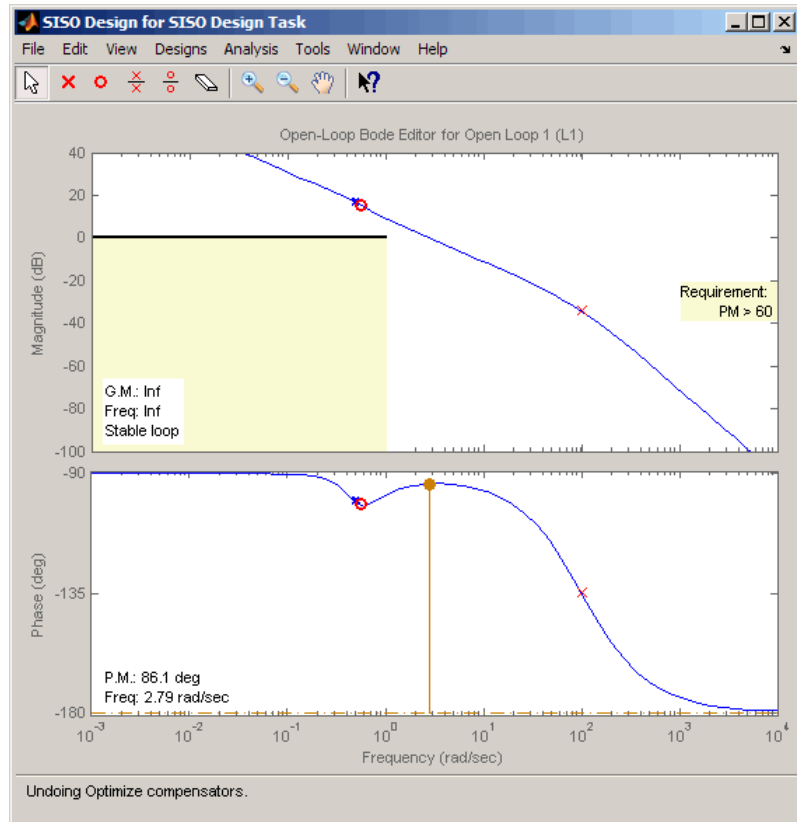
The message `Successful termination` indicates that the optimization method found a solution that meets the design requirements. For more information about the outputs displayed in the **Optimization progress** table, see “Iterative Display” in the Optimization Toolbox documentation.

- 2 Examine the controller parameters and the system's response:
- a View the optimized parameter values in the **Value** field of the **Compensator** tab.



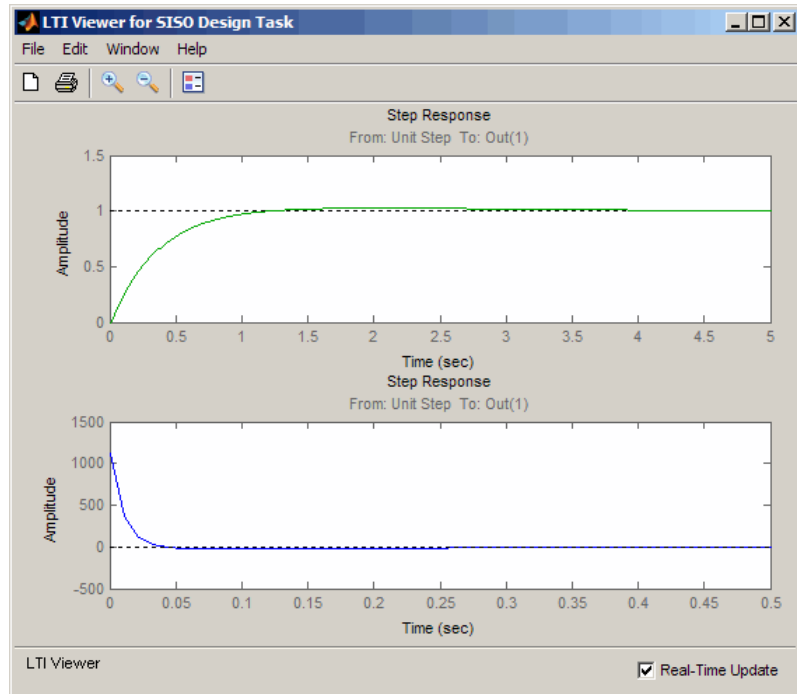
- b Examine the system's response on the following plots:

- The SISO Design window



- The top plot shows that the magnitude of the system, displayed as the blue curve, lies outside the yellow region. This plot indicates that the system has met the Bode magnitude requirement.
- The bottom plot displays the phase margin (P.M.) value of 86.1 degrees. This indicates that the system has met the phase margin design requirement of >60 degrees.

- The LTI Viewer



- The top plot shows that the closed-loop response of the system is stable. The system with the designed controller thus meets both the magnitude and phase margin requirements.
- The bottom plot in the LTI Viewer shows that the peak value of the controller's output is 1000, which is very large and can cause damage to the plant. To limit the controller output, you apply lower and upper bounds on the signal, as described in "Refining the Controller Design to Meet Controller Output Bounds" on page 4-43.

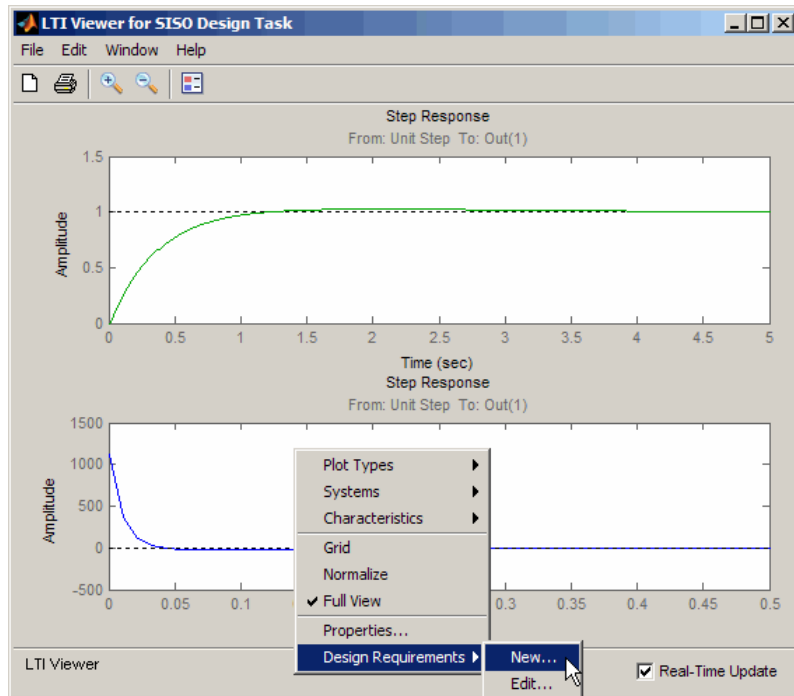
Refining the Controller Design to Meet Controller Output Bounds

In this portion of the tutorial, you refine the controller to satisfy bounds on the controller's output.

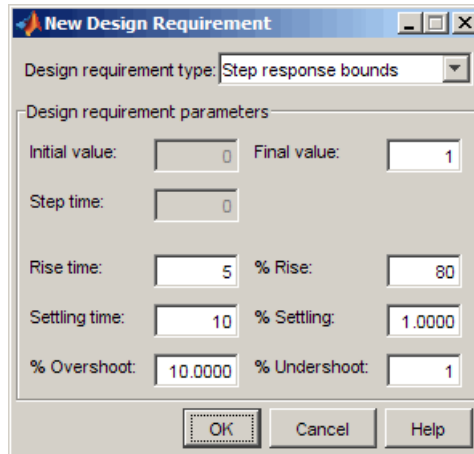
You must have already designed an initial controller, as described in “Designing an Initial PID Controller to Meet Bode Magnitude and Phase Margins Requirements” on page 4-22.

To tune the compensator parameters to meet the bounds on the controller's signal:

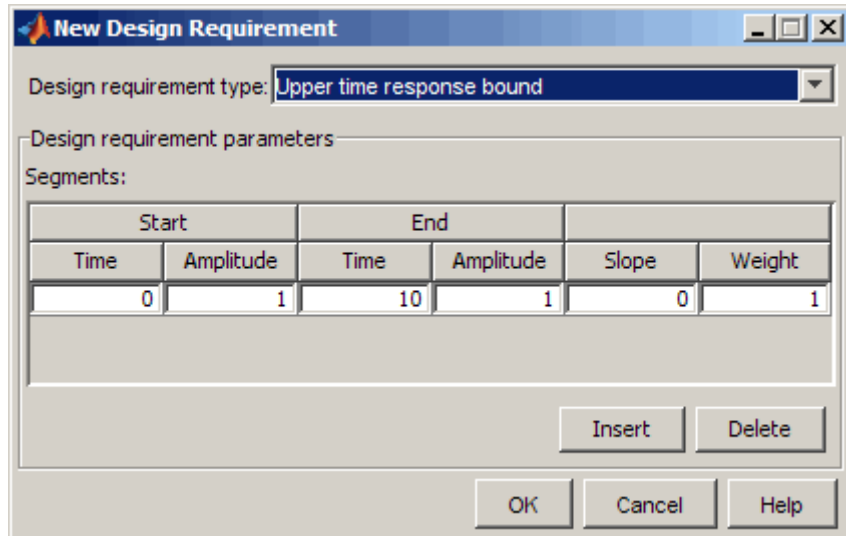
- 1 Add the upper-bound on the controller's output:
 - a In the LTI Viewer, right-click the white area on the bottom plot, and select **Design requirement** > **New**.



This action opens the New Design Requirement dialog box.



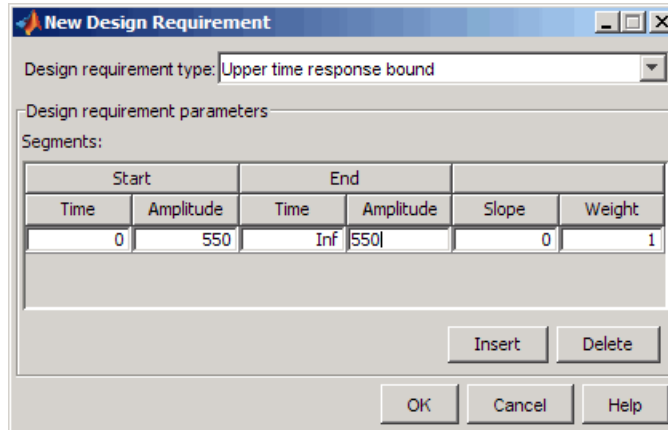
- b** In the New Design Requirement dialog box, select Upper time response bound from the **Design requirement type** drop-down list.



- c** In the **Time** field of the **End** column, enter Inf.
d In the **Amplitude** field of the **Start** column, enter 550.

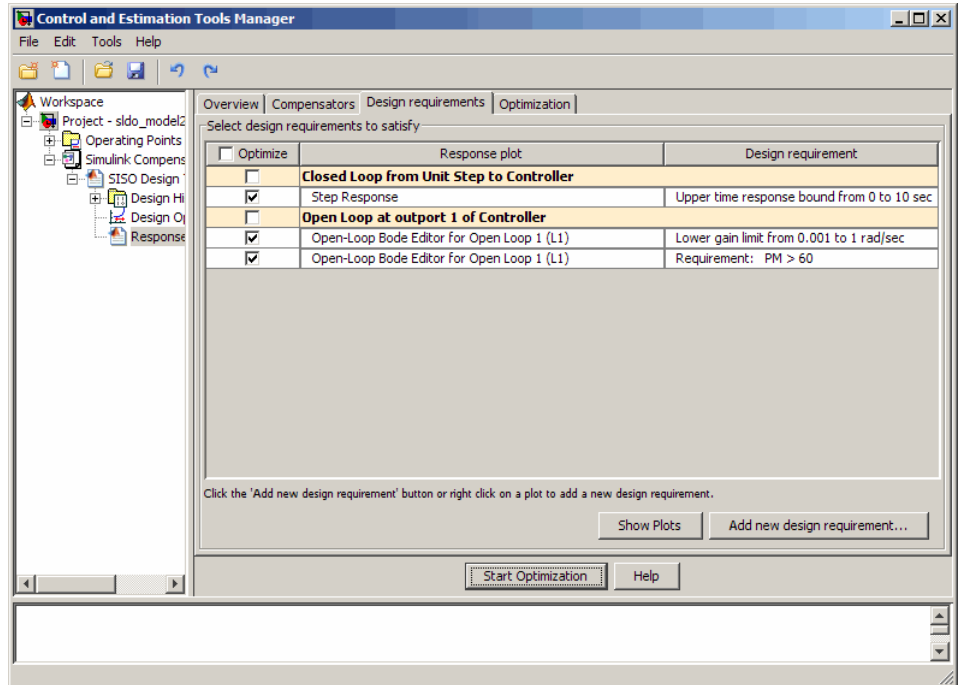
- e In the **Amplitude** field of the **End** column, enter 550.

The New Design Requirements dialog box resembles the following figure.

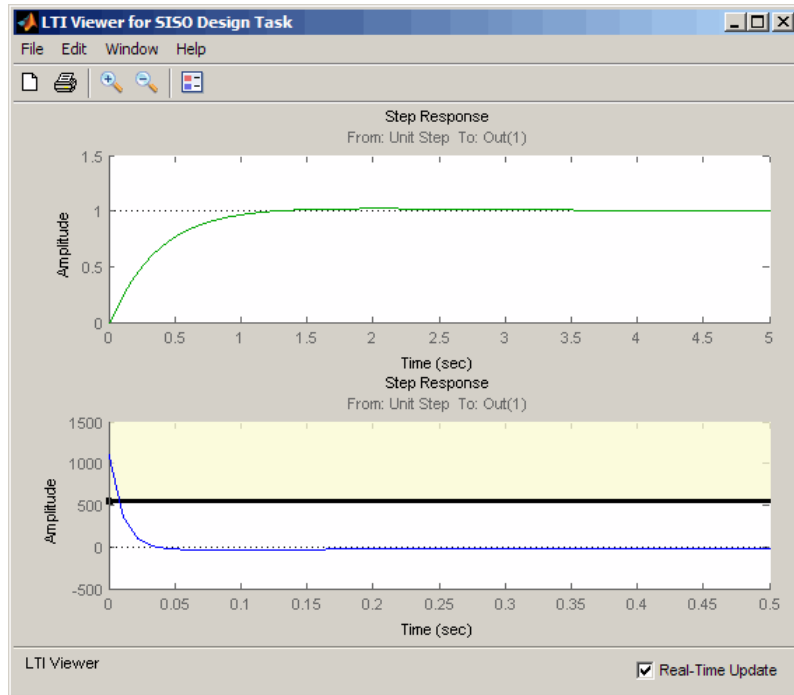


- f Click **OK** to close the dialog box.

The **Design requirements** tab in the Control and Estimation Tools Manager GUI updates. It now displays the upper-bound requirement, as shown in the next figure.



The LTI Viewer also updates to show the design requirement, as shown in the next figure.



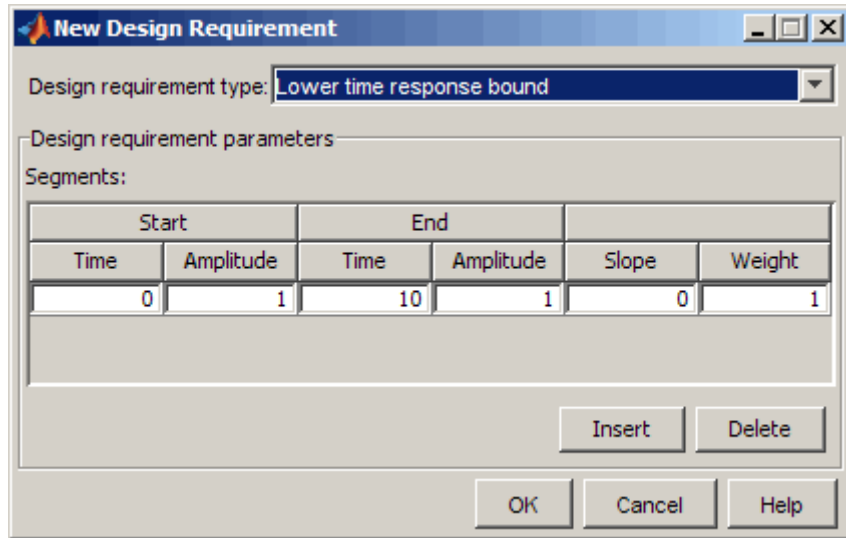
2 Add the lower-bound on the controller's output:

- Right-click the white area in the bottom plot in the LTI Viewer, and select **Design requirement > New**.

This action opens the New Design Requirement dialog box.

- b** Select Lower time response bound from the **Design requirement type** drop-down list.

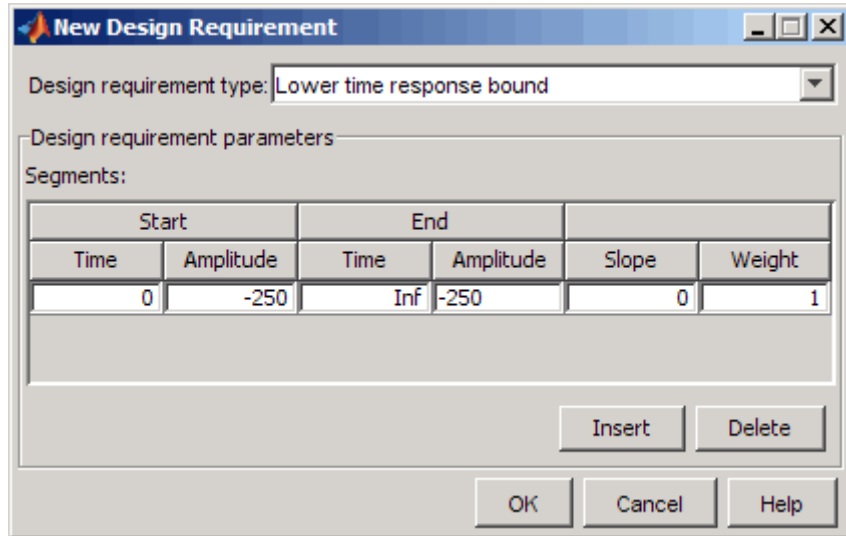
The New Design Requirement dialog box updates to show the lower bound, as shown in the next figure.



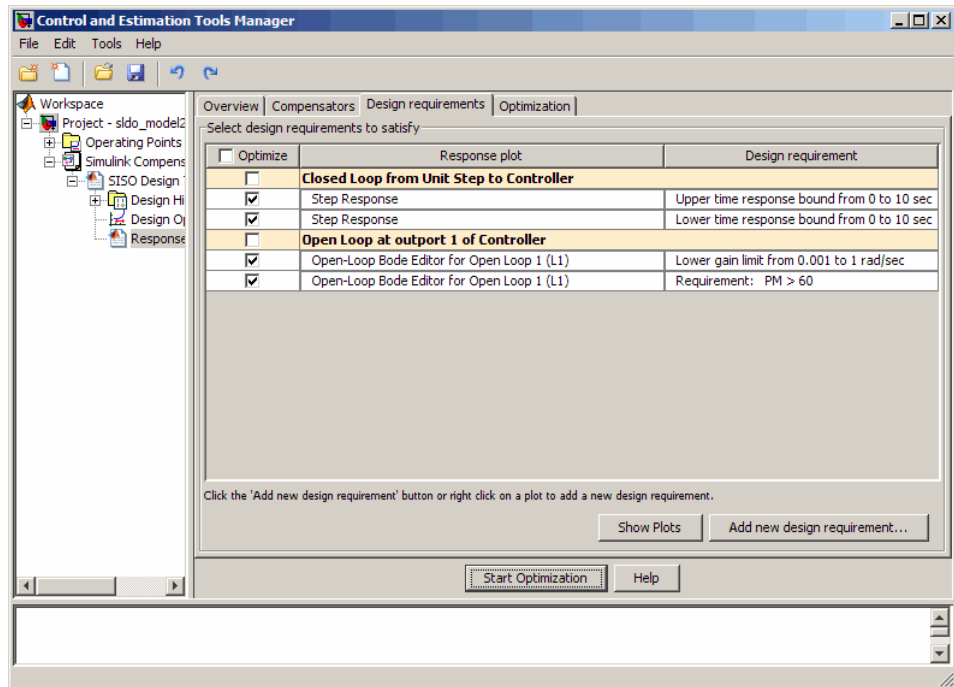
- c** In the **Time** field of the **End** column, enter Inf.
- d** In the **Amplitude** field of the **Start** column, enter -250.

- e In the **Amplitude** field of the **End** column, enter -250.

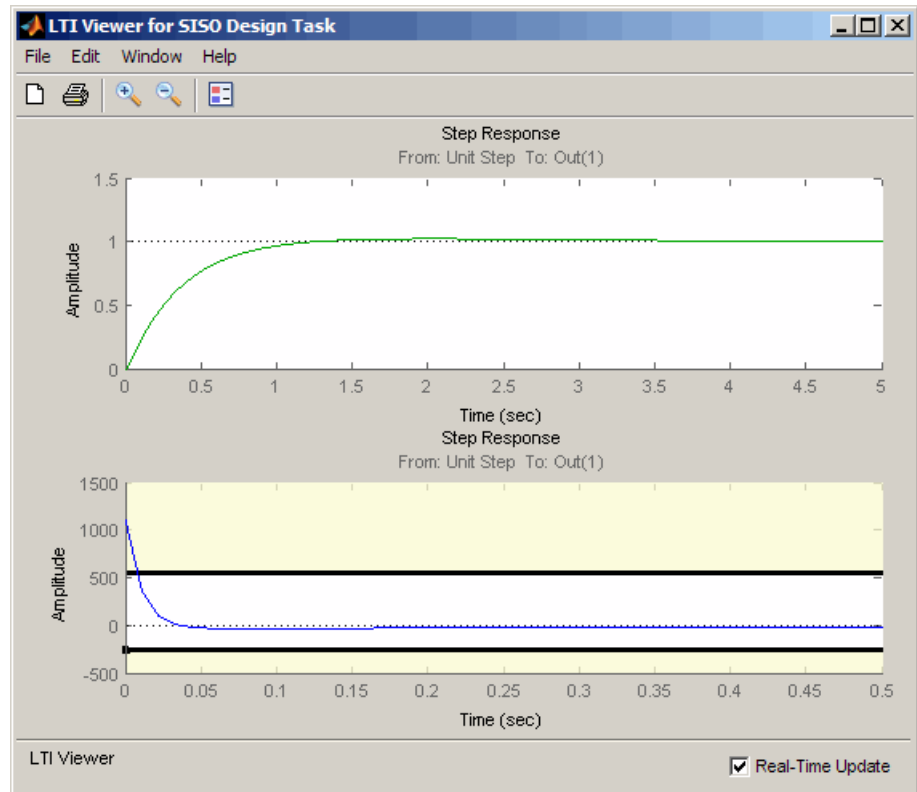
The New Design Requirement dialog box resembles the next figure. Click **OK** to close the dialog box.



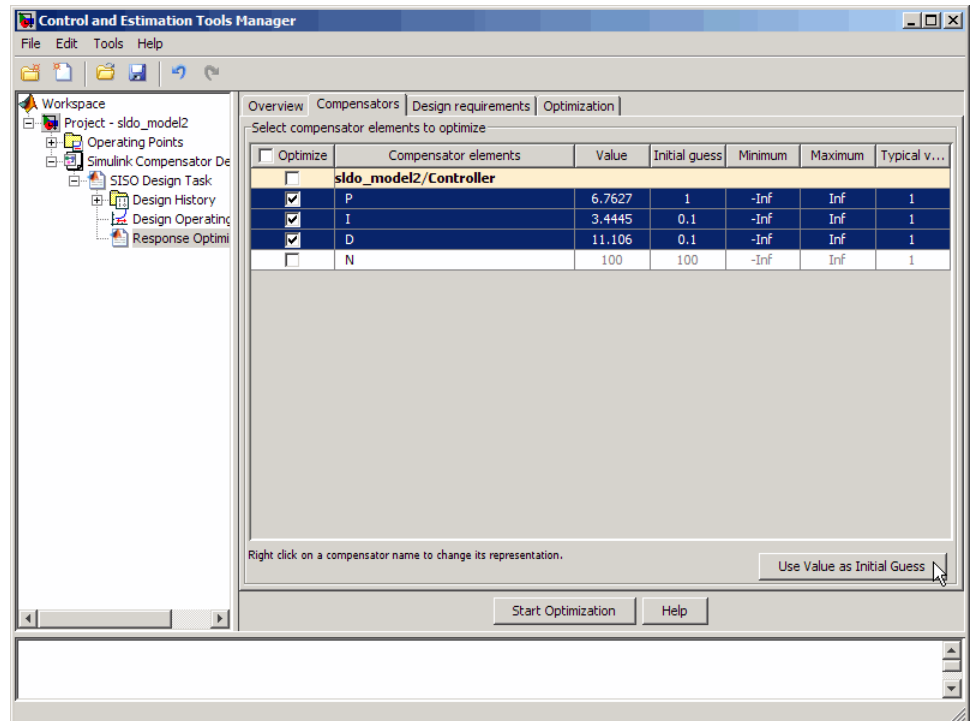
The **Design requirements** tab in the Control and Estimation Tools Manager GUI updates. It now displays the lower-limit requirement, as shown in the next figure.



The LTI Viewer also updates to show the lower-bound on the controller's output, as shown in the next figure.

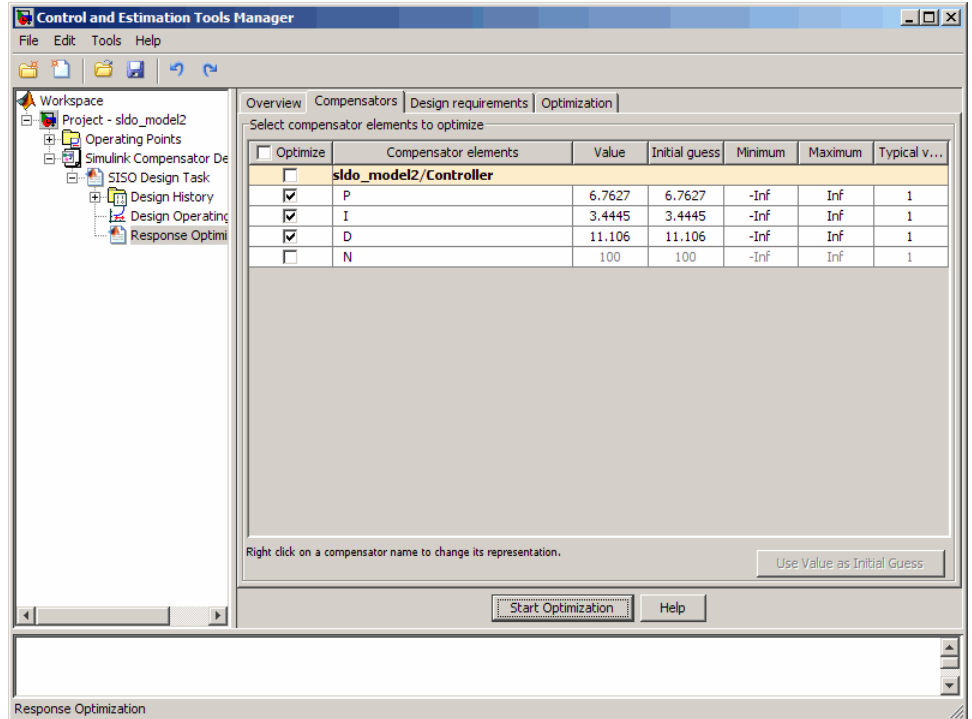


- 3 Optimize the parameters to meet the design requirements on the controller output:
 - a In the **Compensators** tab, select the rows containing P, I, and D, and click **Use Value as Initial Guess**.



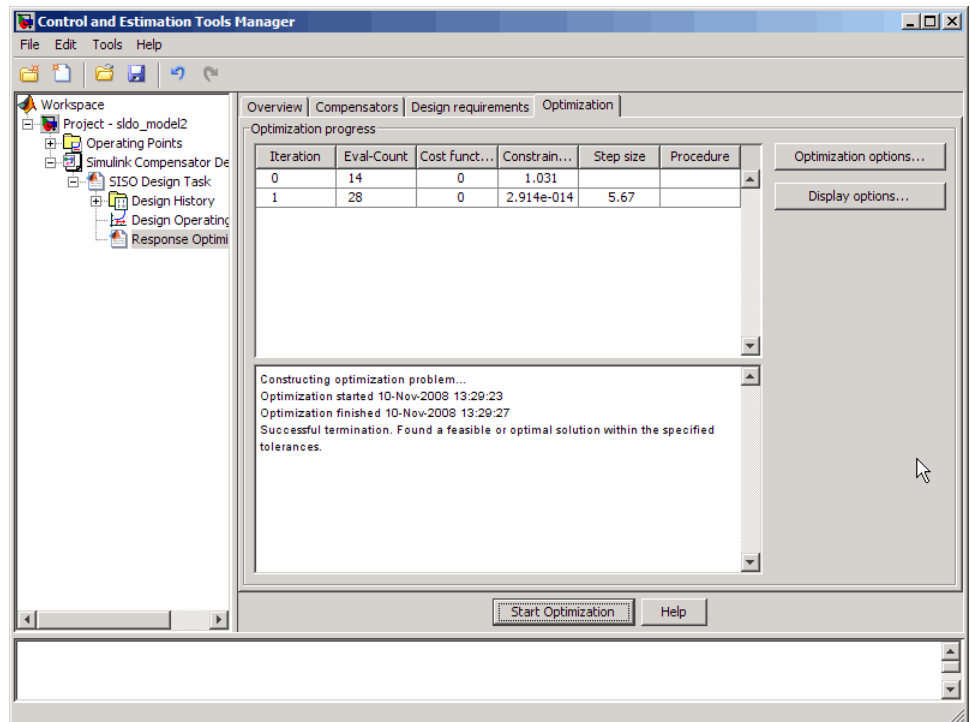
When you run the optimization again, the optimization method uses the updated parameter values as the starting point for refining the values.

Clicking **Use Value as Initial Guess** updates the values in the **Initial Guess** column, as shown in the next figure.

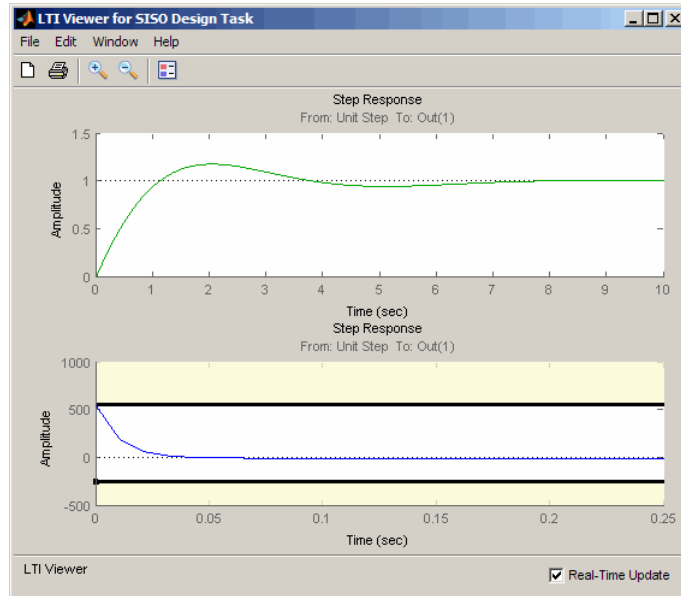


b In the **Optimization** tab, click **Start Optimization**.

- At every optimization iteration, the optimization method reduces the distance between the current response and the upper and lower bounds on the signal.
- After the optimization completes, the **Optimization** tab resembles the next figure.



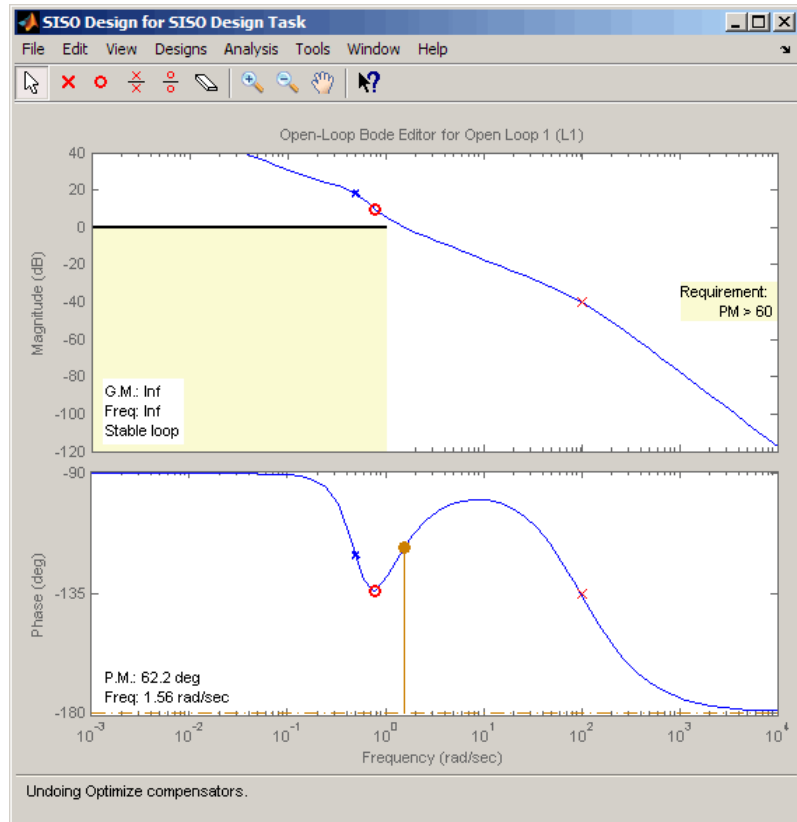
- 4 Examine the parameter values refined controller and the system's response:
 - a Examine the following response plots:
 - The LTI viewer



The bottom plot shows that the controller output lies between 550 and -250, and thus meets the design requirement on the controller's output.

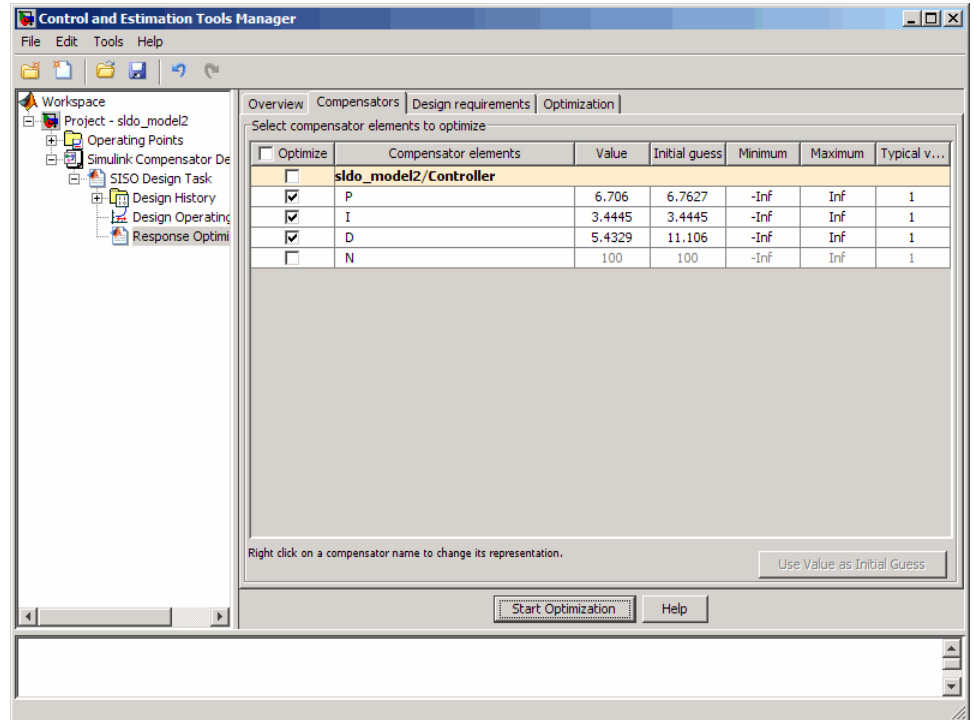
Note You must also check that the closed-loop response, shown in the top plot, remains stable after refining the controller design.

- The SISO Design window



The plots show that after refining the design, the system continues to meet the magnitude and phase margin requirements specified in “Design Requirements” on page 4-16.

- b** View the optimized controller parameter values in the **Value** field in the **Compensators** tab.



- 5** Select the **SISO Design Task** node, and click **Update Simulink Block Parameters**.

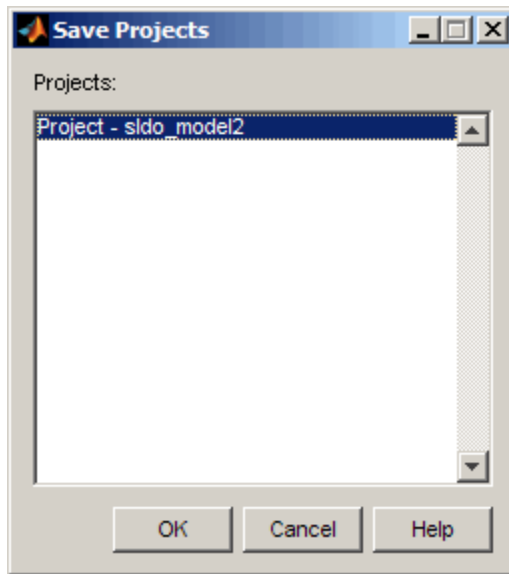
This action writes the optimized controller parameter values to the Controller block in the Simulink model.

Saving the Project

To save a project with the optimized controller parameters:

- 1** In Control and Estimation Tools Manager GUI, select **File > Save**.

This action opens the Save Projects dialog box.



- 2** In the Save Projects dialog box, select **Project - sldo_model2**, and click **OK** to open the Save Projects window .
- 3** In the Save Projects window, enter `sldo_model2_optimized.mat` in the **File name** field, and click **Save**.

The action saves the project as a MAT-file.

Tip You can reload this project by typing `sisotool('sldo_model2_optimized')` at the MATLAB prompt.
