Simulink[®] Design Optimization™ Getting Started Guide

R2012b

MATLAB® SIMULINK®



How to Contact MathWorks



(a)

www.mathworks.comWebcomp.soft-sys.matlabNewsgroupwww.mathworks.com/contact_TS.htmlTechnical Support

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



508-647-7001 (Fax)

508-647-7000 (Phone)

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 1993–2012 by The MathWorks, Inc.

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

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

Trademarks

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See www.mathworks.com/trademarks 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)	

Contents

Product Overview

Product Description	
What You Can Accomplish Using This Product	1-3
Optimization Support for Simulink Models Using Third-Party Applications	1-4
Speeding Up Design Optimization Using Parallel Computing	1-5
Learn More	1-6
Required and Related Products	1-7
Documentation	

Parameter Estimation

2

Supported Data	2-2
Quick Start — Estimating Model Parameters	2-3
Prepare Data for Parameter Estimation Using the GUI About This Tutorial	2-11 2-11

Configuring a Project for Parameter Estimation	2-12
Importing Data into the GUI	2-14
Analyzing Data	2-23
Selecting Data for Estimation	2 - 25
Removing Outliers	2 - 33
Filtering Data	2-37
Interpolating Missing Data	2-42
Saving the Project	2-44

Estimate Parameters from Measured Data Using the

GUI	2-47
About This Tutorial	2-47
Estimating Model Parameters Using Default Estimation	
Settings	2-51
Improving Estimation Results Using Parameter Bounds	2-64
Validating Estimated Model Parameters	2-69

Response Optimization

3

Supported Design Requirements	3-2
Optimize Controller Parameters 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
Optimize Controller Parameters 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	3-21

Optimize Controller Parameters to Track Reference	
Signal (GUI)	3-26
Model Structure	3-26
Design Requirements	3-27
Specify Reference Signal	3-27
Specify Design Variables	3-34
Optimize Model Response	3-37
Design Optimization to Meet Frequency-Domain	
Requirements (GUI, with Check Block)	3-41
	3-41 3-41
Requirements (GUI, with Check Block)	•
Requirements (GUI, with Check Block)Model Structure	3-41
Requirements (GUI, with Check Block)Model StructureDesign Requirements	3-41 3-42
Requirements (GUI, with Check Block)Model StructureDesign RequirementsSpecify Design Requirements	3-41 3-42 3-42

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	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

Examples

A		
	Getting Started	 A-2

Index

L

Product Overview

- "Product Description" on page 1-2
- "What You Can Accomplish Using This Product" on page 1-3
- "Optimization Support for Simulink Models Using Third-Party Applications" on page 1-4
- "Speeding Up Design Optimization Using Parallel Computing" on page 1-5
- "Learn More" on page 1-6
- "Required and Related Products" on page 1-7
- "Documentation" on page 1-8

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 timeand 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

What You Can Accomplish Using This Product

Simulink Design Optimization software supports parameter optimization for Simulink models that invoke third-party simulation tools or contain legacy simulation code (using S-functions).

The software uses methods from the Optimization Toolbox[™] and Global Optimization Toolbox products for design optimization. Using the software with the Parallel Computing Toolbox[™] software can speed up numerical computations during estimation and optimization.

The software provides library blocks that let you accomplish the following goals:

- Model time-varying systems as lookup tables using "Adaptive Lookup Tables"
- Optimize model parameters by graphically specifying time-domain design requirements and reference signal using Signal Constraint blocks
- Compute and optimize continuous and discrete root mean square values of signals using CRMS and DRMS blocks with Signal Constraint blocks

When the Control System Toolbox[™] software is installed, you can refine controller parameters of linear time-invariant (LTI) models and Simulink models, linearized using the Simulink Control Design software, in the SISO Design Tool.

You can work either in the Graphical User Interface (GUI) or at the command line. New users should start by using the GUI to become familiar with the product. The following operations are available only using the GUI:

- Interactive data preprocessing for parameter estimation
- Model validation using residual plot for validating estimation
- Optimization-based controller design

1

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 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.

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.
- ThePattern 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.

1

Learn More

The Simulink Design Optimization documentation provides information to use this product. Although this product employs optimization methods for parameter estimation and optimization, using the product does not require you to have a strong background in optimization theory. You may find it helpful to:

- Consult the Optimization Toolbox documentation to learn more about optimization theory, and minimizing an objective function.
- Learn how Simulink Design Optimization algorithms formulate the optimization problems. See "How the Optimization Algorithm Formulates Minimization Problems" in the User's Guide.

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.

T

Documentation

Accessing Documentation

The Simulink Design Optimization documentation contains the following components:

- Getting Started Guide Provides information for mapping your problem to the capabilities of the Simulink Design Optimization software. Step-by-step tutorials walk you through the most common tasks for estimating parameters, optimizing parameters, and designing controllers using optimization methods.
- Reference Describes commands and blocks for design optimization.
- Release Notes Describes important changes in the current product version and compatibility considerations.

If you are new to using this product, the Getting Started Guide helps you begin using this product quickly. You can follow the steps in the tutorials to perform design optimization using the graphical user interface (GUI) or the MATLAB Command Window.

You can also search or browse the documentation for information about specific design optimization tasks.

Parameter Estimation

- "Supported Data" on page 2-2
- "Quick Start Estimating Model Parameters" on page 2-3
- "Prepare Data for Parameter Estimation Using the GUI" on page 2-11
- "Estimate Parameters from Measured Data Using the GUI" on page 2-47

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 Using the GUI" on page 2-47.

Quick Start – Estimating Model Parameters

In this quick start, you get an overview of the typical tasks for estimating model parameters using the Control and Estimation Tools Manager GUI:

- **1** Start a parameter estimation task.
- 2 Import estimation and validation data sets.
- **3** Select parameters to estimate.
- **4** Estimate the parameters from the estimation data.
- **5** Validate the estimated parameters using the validation data set.

Prerequisites for parameter estimation include:

• Simulink model that contains inport or outport blocks, or signal logging

For more information, see "Model Requirements for Importing Data" in the Simulink Design Optimization User's Guide.

• Transient data in the MATLAB workspace

To estimate model parameters:

1 Start a parameter estimation task by selecting Analysis > Parameter Estimation in the Simulink model window.

	File View Help		1
New estimation task	Workspace Project - spe_engine_t Project - s	Task settings Title: Subject: Author: Company: Description:	Optional information about the estimation task
		Model: spe_engine_throttle1 Open ModelUpdate Task	
	Select the nodes below to config	ire and run estimations.	

- **2** Import the input and output data for estimating and validating model parameters.
 - a Select the Transient Data node, and click New.
 - **b** Select the **New Data** node.
 - **c** In the **Input Data** tab, select the **Data** cell corresponding to the model channel, and click **Import**. Select the variable to import in the Data Import dialog box.

fo	click	n unu	Select a click Imj import ti	port to		Select variable to import
Control and Estimation Too	ols Manager					×
File View Help						Data Import
🗃 🎦 🗳 🛃 🔳						
	nput Data Output Da Assign data to blocks-	ita 🛛 State Data				Import from: Workspace
E-un Transient Data	Block Name	Data	Time / Ts	Weight	Length	Variable Name Size Bytes Class
	spe_engine_throt	le1/Input				J 1x1 8 double
	Channel - 1			1	-1-	Kt 1x1 8 double
Escination						🗰 angle init 1x1 8 doubl 💌
						 Assign the following columns to selected channel
						C Assign the following rows to selected channel(:
	Import	Pre-proces	F	Plot Data	Clear All	Import Close Help
- New Data node has been added to	to Transient Data.					-
Select the tabbed panels to configu	ire the transient data :	set.				Import variable

- **d** Select the **Time** /**Ts** cell, and click **Import**. Select the time vector to import in the Data Import dialog box.
- In the **Output Data** tab, repeat steps c-d to import the output data and time vector.

	🙀 Control and Estimation Tools	Manager				<u> </u>
	File View Help					
	🗃 🛅 🖨 🛃 🔳					
	📣 Workspace	Input Data Outp	ut Data State D	Data		
	Project - spe_engine_throt	Assign data to blo	cks	· · · · · · · · · · · · · · · · · · ·		
	Estimation Task	Block Name	Data	Time / Ts	Weight	Length
	[iii] New Data	spe_engine_th	nrottle1/Input			
Validation _ data node	► 👫 New Data (2)	Channel - 1			1	-/-
	Variables					
	Estimation					
		1				
		Import	Pre-proce	es Plo	ot Data	Clear All
	- New Data node has been added to	Transient Data.				
	- New Data (2) node has been added	to Transient Data.				
	J	the transient date				
	Select the tabbed panels to configure	the transfert data	set.			///

f Repeat steps a-d to import the validation data set.

For more information, see "Import Data (GUI)" in the *Simulink Design Optimization User's Guide*.

3 Specify parameters to estimate by selecting the **Variables** node, and clicking **Add**. Select the parameters in the Select Parameters dialog box.

Control and Estimation Too File View Help	ols Manager		Select parameter to estimate	
Workspace Project - spe_engine_thr Estimation Task Estimation Task Wirkswards Estimation Task Wirkswards Wirkswards Estimation Waldation	Estimated Parameters Estimated States	Default settings Name: Value: Initial guess: Minimum: Maximum: Typical value: Referenced by:	Select Parameters Select additional parameters to e Name Kt angle_init angle_open C input_delay K Specify expression (e.g., s.x, a)	Size
New Data (2) node has been add	Add Delete		OK Cancel 1	Help Apply

Specify parameters

For more information, see "Specify Parameters to Estimate (GUI)" in the Simulink Design Optimization User's Guide.

- **4** Estimate the parameters.
 - **a** Select the **Estimation** node, and click **New**.
 - **b** Select the **New Estimation** node.
 - c In the Data Sets tab, select the estimation data set.

	ists imported lata sets	Select dat for estima				
Control and Estimation Tools	Manager					
File View Help						
🖆 🎦 🗳 🛃 🔳						
📣 Workspace	Data Sets Parameters	s States Estima	tion			
Project - spe_engine_throt	Data sets used for esti	imation		-Output data wei	ghts	
Estimation Task	Transient estimation		-	Block Name	Length	Weight
- [;;;] New Data				spe_engine_t	hrottle1/Pos	ition
Itil New Data (2)	Data Set	Selected		Channel - 1	- -	1
Variables	New Data New Data (2)					
Estimation	New Data (2)					
····· 🖧 New Estimation •···· 🕞 🕞 Views			-			
	Select All	Clear All				
	Select All					
- New Estimation node has been adde	ed to Estimation.					_
						<u> </u>
Select the tab panels to configure you	ur estimation.					11.

					value (Optiona	al) Specify ter bounds	
Control and Estimation Tools	Manager						_ D ×
File View Help							
😅 🛅 🗳 🛃 🔳							
Workspace ⊡	Data Sets Para		Estimation				
Estimation Task				• • •		<u> </u>	1
🔁 🛅 Transient Data	Name	Value	Estimate	Initial Guess	Minimum	Maximum	Турі
[it] New Data	<u> </u>	0.05		J	-Inf -Inf	+Inf	II
liți New Data (2)	input_delay	40	 	c input delav	-Inf	+Inf +Inf	inpu
Variables	Linput_delay	1		hput_delay	-Inf	+100 +10f	inpu
Estimation	•	•		1			
Validation	Use Value as	s Initial Guess	Reset to	Default Settings	Save a	as Default Sett	tings
- New Estimation node has been add	ed to Estimation.						
Select the tab panels to configure yo	ur estimation.						

d In the **Parameters** tab, select the parameters to estimate.

e In the Estimation tab, begin estimation by clicking Start.

File New Help File New Help File New Help Statistics Project Statistics Project Statistics Project Statistics Project Statistics Project Statistics Project Statistics Project Statistics Statistics Project Statistics				(converge	on about ence of th n algorith	ne		
13 14 1891.3 0.1031 14 15 1891.3 0.1031 14 15 1891.3 0.1031 14 15 1891.3 0.1031 Active experiments: New Data Active aperiments: New Data Active aperiments: New Data	File View Help	Data Sets Par Etimation pro- Iteration 6 9 9 10 11 12 13 14 Herrorium Active exp Estimated Local minin Isquares references of the func-	Press Control	Cost Function 1891.4 1891.3 1891.3 1891.3 1891.3 1891.3 1891.3 1891.3 1891.3 1891.3 1891.3 1891.3 1891.3 1891.3 1891.4 1891.5 1891.4 1891.5	0.1031 0.1031 0.1031 0.1031 0.1031 0.1031 0.1031 0.1031 0.1031 0.1031 0.1031 0.1031 0.1031 0.1031 0.1031	m of		Estmation Options Display Options	Start estimation Select to view measured and simulated response plots Estimation

f In the **Parameters** tab, examine the estimated parameter values. The Simulink model also gets updated with the estimated parameter values.

		stimated barameter v	alues				
Gontrol and Estimatio	n Tools Manage	r					_ 🗆 ×
File View Help							
🗃 🛍 🖆 🛃 🔳	Ĵ						
Workspace	Data Sets Para	meters States	Estimation				
Estimation Task	Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Valu
Variables	J	0.22364	V	J	0	+Inf	J
Estimation	c	9.0575	~	c	0	+Inf	с
💀 New Es	input_delay	0.0089359	~	input_delay	0	0.1	input_dela
🖃 🛄 Views	k	19.896		k	0	+Inf	k
🛛 🗠 🔀 Validation	Use V.	alue, as Initial Gu	Jess Re	set to Default Se	ettings	Save as Defaul	L Settings
Estimation completed.	ļ						 ;
Estimation completed. Select the tab panels to conf	iqure vour estimati	n					
beleet the tab parters to com	igure your escillaci	2016					

For more information, see "Estimate Parameters (GUI)" in the *Simulink* Design Optimization User's Guide.

- **5** Validate the estimated parameters.
 - **a** Select the Validation node, and click New.
 - **b** Select the **New Validation** node.

		elect validation ot type	Select valio data set	dation	
🙀 Control and Estimation To	ols Manager				
File View Help					
🗃 🎦 😂 🛃 🔳					
📣 Workspace	Validation Setup				
Project - spe_engine_thr Engine_Estimation Task	Select plot types	+			
Estimation Task	Plot Number	Plot Type		Plot Title	
- [;;;] New Data	Plot 1 Reside				<u> </u>
[iii] New Data (2	Plot 2 (none				
Variables	Plot 3 (none Plot 4 (none				_
Estimation					
	Options		•		
🖳 🔛 New Viev	Validation data set:	New Data (2)			
Ualidation	Estimation	Plot	1		
🚱 New Validatio	New Estimation		-		
		1			
			Show Plots		
TYCH Validation Hode has been a	ace to validation.				A V
Configure validation plots.					
		0.1			///
		Select to display	- Open		
		validatio	on plot valida	ation plots	

c Configure the validation plots and the validation data set.

For more information, see "Validate Parameters (GUI)" in the *Simulink Design Optimization User's Guide*.

See Also: "Estimate Parameters from Measured Data Using the GUI" on page 2-47

Prepare Data for Parameter Estimation Using the GUI

In this section
"About This Tutorial" on page 2-11
"Configuring a Project for Parameter Estimation" on page 2-12
"Importing Data into the GUI" on page 2-14
"Analyzing Data" on page 2-23
"Selecting Data for Estimation" on page 2-25
"Removing Outliers" on page 2-33
"Filtering Data" on page 2-37
"Interpolating Missing Data" on page 2-42
"Saving the Project" on page 2-44

About This Tutorial

- "Objectives" on page 2-11
- "About the Sample Data" on page 2-12

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.mdl contains the Simulink model of the engine throttle system.

Configuring a Project for Parameter Estimation

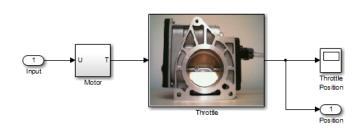
To perform parameter estimation, you must first configure a Control and Estimation Tools Manager project.

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.

Engine Throttle Model



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.

🙀 Control and Estimation Tools Man	ager			
File View Help				
🋅 🗅 🗳 🖬 🗐				
📣 Workspace	-Task settings			
Project - spe_engine_throttle1	Title:			
Estimation Task	Subject:			
Variables	Author:	[
Estimation	Company:			
-	Description:			
	Description			<u> </u>
				-
		,		
	Model:	spe_engine_throttle1	Open Model	Update Task
				<u> </u>
				•
Select the nodes below to configure and ru	n estimations.			11.

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 2-15

• "Importing Output Data and Time Vector" on page 2-20

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 2-13. 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 2-12.

To import input data and time vector into the Control and Estimation Tools Manager GUI: 1 In the Control and Estimation Tools Manger GUI, select **Transient Data** under the **Estimation Task** node, and click **New**.

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

Control and Estimation Tools File View Help	1anager		<u>_ </u>
Workspace Project - spe_engine_throttle Estimation Task Transient Data	-Transient data sets Name New Data	Properties	
Variables			
			•
	Description:		4
× >		Delete Edit	T
- New Data node has bee	n added to Transient Data.		•
, Transient data sets are stored in this f	older. Press 'New' to create a new data set		- //.

2 Select the **New Data** node.

Gontrol and Estimation Tools	Manager				
File View Help					
😁 🗅 🚅 🖥 🔳					
📣 Workspace	Input Data Output Data	a State Data			
Project - spe_engine_throttle	Assign data to blocks				
Estimation Task	Block Name	Data	Time / Ts	Weight	Length
	spe_engine_throttle	e1/Input			
Variables	Channel - 1			1	-1-
Validation					
		Import	Pre-process	Plot Data	Clear All
	L				
- New Data node has bee	n added to Trans	sient Data.			
					-
Select the tabbed panels to configure	the transient data set.				

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

This action opens the Data Import dialog box.

🙀 Data Import				×			
Import from: Workspace							
Variable Name	Size	Bytes	Class				
🗮 in1	201x1	1608	double	-			
🗰 input1	701x1	5608	double				
🗰 input2	500x1	4000	double				
🗰 input3	500x1	4000	double	•			
Assign the following columns to selected channel(s): 1 Assign the following rows to selected channel(s): [1:701]							
Limport Close Help							

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**.

🙀 Control and Estimation Tools	Manager				
File View Help					
😁 🗅 🗳 🖬 🔟					
📣 Workspace	Input Data Output Data State Data				
Project - spe_engine_throttl	Assign data to blocks				
Escillation Task	Block Name	Data	Time / Ts	Weight	Length
[iii] New Data	spe_engine_throttle1/Ir				
Variables	Channel - 1	input1(:,1)		1	701/-
Estimation					
		Import	Pre-process	Plot Data	Clear All
- New Data node has be		nt Data.			×
Select the tabbed panels to configure	the transient data set.				

- **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**.

🙀 Control and Estimation Tools	Manager				
File View Help					
🗚 🗅 😂 🖬 🔳					
Workspace Input Data Output Data State Data					
Project - spe_engine_throttle E- E Estimation Task	Assign data to blocks				
Escination Task	Block Name	Data	Time / Ts	Weight	Length
🗔 🛄 New Data	spe_engine_throttle1/input				
Variables	Channel - 1	input1(:,1)	time1(:,1)	1	701/701
A C Validation					
		Import	Pre-process	Plot Data	Clear All
- New Data node has been added to Transient Data.					
					•
Select the tabbed panels to configure the transient data set.					

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 Manger GUI, select the **Output Data** tab of the **New Data** node.

🙀 Control and Estimation Tools	Manager				
File View Help					
🚓 🗅 👒 🖪 🗐					
📣 Workspace	Input Data Output Data	3 State Data			
🖻 💽 Project - spe_engine_throttle	oject - spe_engine_throttleAssign data to blocks				
Estimation Task	Block Name	Data	Time / Ts	Weight	Length
Lis Halson Data	spe_engine_throttle1/Position				
Variables	Channel - 1			1	-1-
Estimation					
		Import	Pre-process	Plot Data	Clear All
- New Data node has bee	en added to Trans	sient Data.			<u> </u>
					•
J Select the tabbed panels to configure	the transient data set.				

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

This action opens the Data Import dialog box.

🙀 Data Import				×
Import from: Workspace 💌				
Variable Name	Size	Bytes	Class	
🗮 kexp	1x1	8	double	-
position1	701x1	5608	double	
position2	500x1	4000	double	
position3	500x1	4000	double	-
 Assign the following columns to Assign the following rows to s): 1	11
	port	Close	Help	

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**.

🙀 Control and Estimation Tools	Manager			
File View Help				
😁 🗅 🚅 🖬 🔳				
📣 Workspace	Input Data Output Data State Data			
Project - spe_engine_throttl E- E E Stimation Task	Assign data to blocks			
Esumation Task	Block Name Data	Time / Ts	Weight	Length
[iii] New Data	spe_engine_throttle1/Position			
Variables	Channel - 1 position1(:	,1)	1	701/-
⊕ G Estimation				
		Pre-process	Plot Data	Clear All
- New Data node has be	en added to Transient Data.			
Select the tabbed panels to configure	the transient data set.			

- 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**.

🙀 Control and Estimation Tools	Manager				_ _ _ _ ×
File View Help					
📸 🗅 🛸 🖬 🗐					
📣 Workspace	Input Data Output Data	State Data			
Project - spe_engine_throttle E- E E E	Assign data to blocks				
Escination Task	Block Name	Data	Time / Ts	Weight	Length
[iii] New Data	spe_engine_throttle				
Variables	Channel - 1	position1(:,1)	time1(:,1)	1	701/701
Estimation				Plot Data	Clear All
•		Import	Pre-process		
- New Data node has bee	en added to Trans	sient Data.			
Load projects					///

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.

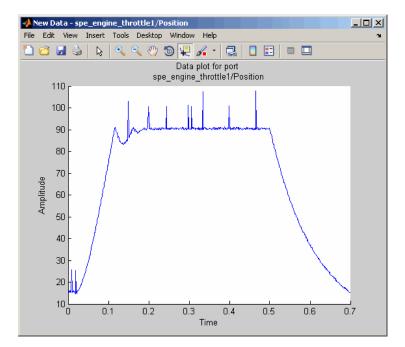
Analyzing Data

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 2-14. 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**.

🙀 Control and Estimation Tools	Manager					- D ×
File View Help						
🗚 🗅 😂 🖬 🔳						
📣 Workspace	Input Data Output Data	^a State Data				
Project - spe_engine_throttle	Assign data to blocks					
Estimation rask	Block Name	Data	Time / Ts	Weight	Length	
Iti New Data	spe_engine_throttle					
Variables	Channel - 1	position1(:,1)	time1(:,1)	1	701/701	
Estimation						
۲		Import	Pre-process	Plot Data	Clear #	1
- New Data node has bee	en added to Trans	sient Data.				▲ ▼
Load projects						11.



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 2-12. 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 2-25 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 2-33, and "Filtering Data" on page 2-37 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 2-25
- "Selecting Input Data" on page 2-31

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 2-23, 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 2-14. If you have not imported the data, click here.

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

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.

🙀 Data Preprocessing Tool f	or Dataset: New Data	×
Modify data fromthrottle1/Po	write results to:	C existing dataset w Data 💌 💿 new dataset Dataset1
Data Editing		
Raw data Modified data		Exclusion Rules Detrend/Filtering
Time (seconds)	position1(:,1)	
0.0	15.116	F Bounds
1.0e-3	15.964	
2.0e-3	15.241	Exclude X: <= 💌 Exclude X: >= 💌
3.0e-3	15.993	
4.0e-3	15.376	Exclude Y: <=
5.0e-3	15.997	
6.0e-3	15.380	_
7.0e-3	15.100	Outliers
	15.649	
	25.592	Window length: 10
1.0e-2	15.944	windowicigen.
	15.238	Confidence Italia (W.)
1.200e-2	15.817	Confidence limits (%): 95
1.300e-2	15.385	
, <u>, </u>		MATLAB expression abs(x)>1
Grey back Excluded by Red text Manually exc	Exclude Graphically	Flatines Window: 5
Missing Data Handling		
Prissing Data ndi Iulii Iy		
Remove rows where	data is excluded or missing	Tinterpolate missing values using interpolation method zoh
		Add Close Help

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.

😺 Data Preprocessing Tool	for Dataset: New Data	X
Modify data fromthrottle1/F		existing dataset
Data Editing		
Raw data Modified data		Exclusion Rules Detrend/Filtering
Time (seconds)	position1(:,1)	
0.0	15.116	✓ Bounds
1.0e-3	15.964	
2.0e-3	15.241	Exclude X: <=
3.0e-3	15.993	
4.0e-3	15.376	Exclude Y: <=
5.0e-3	15.997	
6.0e-3	15.380	
7.0e-3	15,100	Cutliers
8.0e-3	15.649	
9.0e-3	25.592	
1.0e-2	15.944	Window length: 10
1.100e-2	15.238	
1.200e-2	15.817	Confidence limits (%): 95
1.300e-2	15.385	
		MATLAB expression abs(x)>1
Grey back Excluded by Red text Manually ex	Exclude Graphically	Flatlines Window: 5
Missing Data Handling		
Remove rows where all	data is excluded or missing	Interpolate missing values using interpolation method zoh
		Add Close Help

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 position1(:,1)* in the **Output Data** tab.

🙀 Control and Estimation Tools	Ma	nager					
File View Help							
🛎 🗅 📂 🖬 🔳							
K Workspace	I	nput Data Output Data	State Data				
Project - spe_engine_throttle	Ir'	Assign data to blocks					
E-		Block Name	Data	Time / Ts	Weight	Length	
[iii] New Data		spe_engine_throttle					
[#] Dataset1		Channel - 1	position1(:,1)*	time1(:,1)*	1	701/701	
Horrables Estimation Ref Validation							
			Import	Pre-process	Plot Data	Clear Al	
Select the tabbed panels to configure	the	transient data set					▲ ▼
select the cabbed panels to configure	the	transient data set.					//

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.

Data Preprocessing Tool I		existing dataset Dataset V • new dataset Dataset2
-Data Editing		
Raw data Modified data		Exclusion Rules Detrend/Filtering
Time (seconds)	position1(:,1)*	
0.497	90.331	Bounds
0.498	90.323	
0.499	90.816	Exclude X: <=
0.500	90.890	
0.501	NaN	Exclude Y: <=
0.502	NaN	
0.503	NaN	
0.504	NaN	Coutliers
0.505	NaN	
0.506	NaN	US- day las atta
0.507	NaN	Window length: 10
0.508	NaN	
0.509	NaN	Confidence limits (%): 95
0.510	NaN	
		MATLAB expression abs(x)>1
Grey back Excluded by Red text Manually ex	Exclude Graphically	Flatines Window: 5
Missing Data Handling		
Remove rows where	data is excluded or missing	Interpolate missing values using interpolation method zoh
		Add Close Help

- 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 all a data is excluded or missing check box.

Data Preprocessing Too Modify data fromthrottle1 Data Editing		C existing dataset Dataset1 V C new dataset Dataset2
Raw data Modified data	,	Exclusion Rules Detrend/Filtering
Time (seconds)	position1(:,1)**	E punt
0.487	90.604	Bounds
0.488	90.533	
0.489	90.105	Exclude X: <=
0.490	90.592	
0.491	90.72	Exclude Y: <=
0.492	90.648	
0.493	90.772	
0.494	90.432	Cutliers
0.495	90.562	
0.496	90.206	Window length: 10
0.497	90.331	Window length: 10
0.498	90.323	
0.499	90.816	Confidence limits (%): 95
0.500	90.890	
		MATLAB expression abs(x)>1
Grey back Excluded Red text Manually of	Exclude Graphically	Flatines Window: 5
Missing Data Handling	al 💌 data is excluded or missing	Therpolate missing values using interpolation method
		AddCloseHelp

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

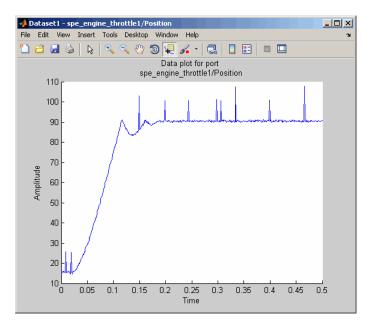
In the Write results to existing dataset Dataset					
Modify data fromthrottle1/Position 💌	Write results to:	 existing dataset 	Dataset1 💌	C new dataset	Dataset2

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 2-25, 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 Manger 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 Dataset option remains selected.
 - d Click Add.

This action adds the modified data 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 input1(:,1)* with NaNs.

- 4 To remove the NaNs:
 - a In the Control and Estimation Tools Manager GUI, select the input1(:,1)* cell in the Input Data tab of the Dataset1 node, and click Pre-process.

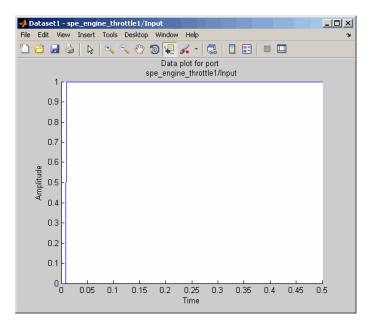
This action updates the Data Preprocessing Tool GUI with the selected data input1(:,1)*.

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

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 2-33
- "How to Remove Outliers" on page 2-34

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 2-42.

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 2-25. 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.

🙀 Data Preprocessing Tool	for Dataset: Dataset1	×
Modify data from throttle1/I	Position 🔻 Write results to: (existing dataset Dataset1 Onew dataset Dataset2
,		
Data Editing		
Raw data Modified data		Exclusion Rules Detrend/Filtering
Time (seconds)	position1(:,1)**	
8,0e-3	15.649	E Bounds
9.0e-3	NaN	
1.0e-2	15.944	Exclude X: <=
1.100e-2	15.238	
1.200e-2	15.817	Exclude Y: <=
1.300e-2	15.385	
1.400e-2	16.11	
1.500e-2	15.879	✓ Outliers
1.600e-2	15.748	
1.700e-2	15.260	Window length: 10
1.800e-2	14.688	
1.900e-2	NaN	Confidence limits (%): 95
2.0e-2	15.391	
2.100e-2	14.742	
		MATLAB expression abs(x)>1
Grey back Excluded b	y a rule Exclude Graphically	
Red text Manually e		Flatlines Window: 5
Missing Data Handling		
Remove rows where all	data is excluded or missing	Therpolate missing values using interpolation method zoh
		Add Close Help

Note The data samples containing outliers are replaced with NaNs.

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

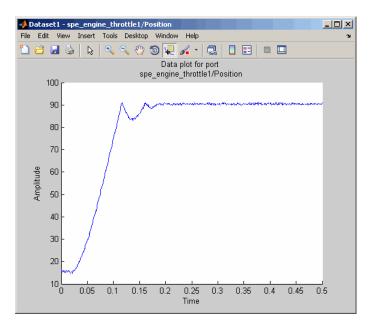
2

4 Click Add.

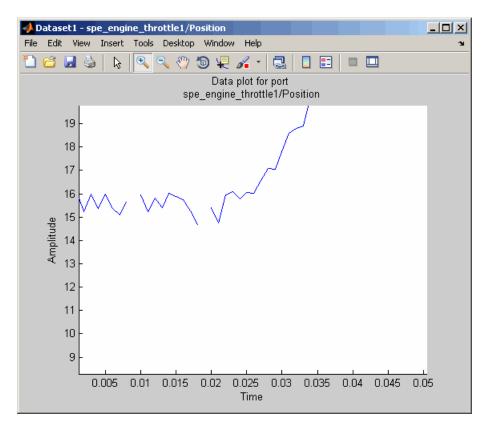
The Update table data dialog box appears. Click **Yes** to overwrite the position1(:,1)* data set with the modified data.

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 2-38
- "Filtering Input Data" on page 2-41

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 2-33. 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.

당 Data Preprocessing Tool I	for Dataset: Dataset1	×
Modify data fromthrottle1/P	osition 💌 Write results to:	🔿 existing dataset Dataset1 🔽 💿 new dataset Dataset2
Data Editing Raw data Modified data		Exclusion Rules Detrend/Filtering
Time (seconds)	position1(:,1)*	Detrending © Constant
	15.964	C Straight Line
2.0e-3	15.241	 Selengine and
	15.993	
	15.376	Filtering Select filter type First order
	15.997	
	15.380	
	15.649	First order filter with time constant 0.001
	NaN	
	15.944	
1.100e-2	15.238	
1.200e-2	15.817	
1.300e-2	15.385	
Grey back Red text Manually ex	Exclude Graphically	
Missing Data Handling		
	data is excluded or missing	Interpolate missing values using interpolation method zoh
		Add Close Help

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

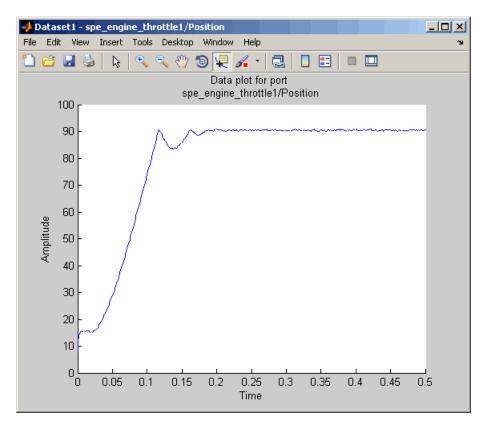
2

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 2-38, 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.

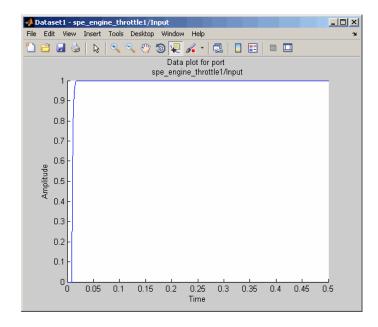
This action updates the Data Preprocessing Tool GUI with the selected data input1(:,1)*.

- 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 Dataset1 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 2-33. 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 2-37. 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)*.

2-42

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.

😺 Data Preprocessing Tool	for Dataset: Dataset1	×			
Modify data fromthrottle1/F	Position Vite results to:	existing dataset Dataset1 C new dataset Dataset2			
Data Editing					
Raw data Modified data		Exclusion Rules Detrend/Filtering			
Time (seconds)	position1(:,1)**				
0.0	15.116	Bounds			
1.0e-3	15.964				
2.0e-3	15.241	Exclude X: <= Exclude X: >=			
3.0e-3	15.993				
4.0e-3	15.376	Exclude Y: <= Exclude Y: >=			
5.0e-3	15.997				
6.0e-3	15.380				
7.0e-3	15.100	Coutliers			
8.0e-3	15.649				
9.0e-3	15.649				
1.0e-2	15.944	Window length: 10			
1.100e-2	15.238				
1.200e-2	15.817	Confidence limits (%): 95			
1.300e-2	15.385				
<u></u>		MATLAB expression abs(x)>1			
Grey back Excluded by a rule Exclude Graphically Red text Manually excluded Exclude Graphically		Flatines Window: 5			
Missing Data Handling					
missing baca nanuling					
Remove rows where all 💌 data is excluded or missing 🔽 Interpolate missing values using interpolation method] zoh 💌					
		Add Close Help			

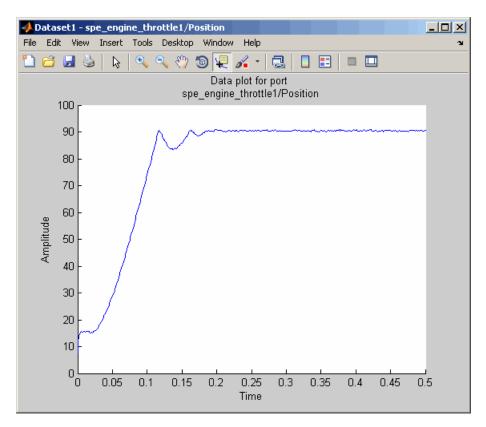
Tip You can view the interpolated data samples in the Modified data tab.

- 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.

🙀 Control and Estimation Tools Mana	ger				_ 🗆 ×		
File View Help							
😁 🗅 🚅 🖬 📋							
Workspace	Input Data Output Data State Data						
Estimation Task	Assign data to blocks						
🔁 🛅 Transient Data	Block Name	Data	Time / Ts	Weight	Length		
lii Estim_Data_Prep	spe_engine_throttl						
Unitables ⊡⊡⊡∰ Estimation	Channel - 1	input1(:,1)*	time1(:,1)***	1	501/501		
명· 물 Estimation 로 · 물 Validation							
		Import	Pre-process	Plot Data	Clear All		
					,		
1					A		
]					_		
Select the tabbed panels to configure the tra	ansient data set.				1		

- 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**.

📣 Save Projects	
Projects:	
Project - spe_engine_throttle1	
	_
OK Cancel	Help
OK Cancel	▼ Help

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 Using the GUI" on page 2-47.

Estimate Parameters from Measured Data Using the GUI

In this section...

"About This Tutorial" on page 2-47

"Estimating Model Parameters Using Default Estimation Settings" on page 2-51

"Improving Estimation Results Using Parameter Bounds" on page 2-64

"Validating Estimated Model Parameters" on page 2-69

About This Tutorial

- "Objectives" on page 2-47
- "About the Model" on page 2-48

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.

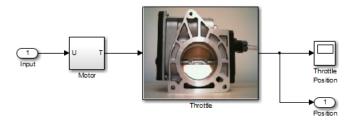
You learn to perform the following tasks using the GUI:

- 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.

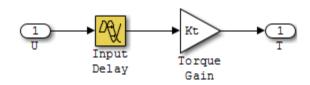
Engine Throttle Model



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 2-49, and "Throttle Subsystem" on page 2-49.

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).

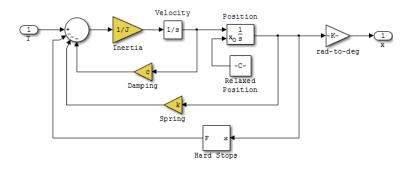
Motor Subsystem. The Motor subsystem contains the DC motor model. To open the model, double-click the corresponding block.



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

Variables	<i>U</i> 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.
	${\tt t}_{\tt d}$ 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	Т

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

2-50

Equations	The mathematical system for the butterfly value is described in the following equation: $J\ddot{\theta} + c\dot{\theta} + k\theta = T + T_{hardstop}$ where $15^{\circ} \le \theta \le 90^{\circ}$, 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: $T_{hardstop} = \begin{cases} 0, & 15^{\circ} \le \theta \le 90^{\circ} \\ K(90^{\circ} - \theta), \theta > 90^{\circ} \\ K(15^{\circ} - \theta), \theta < 15^{\circ} \end{cases}$ where K is the gain of the Hard Stops block.
Input	Т
Output	θ

Estimating Model Parameters Using Default Estimation Settings

- "Overview of the Estimation Process" on page 2-52
- "Specifying Parameters and Estimation Data" on page 2-53
- "Validating Model Parameters" on page 2-57

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 2-53
- "Validating Model Parameters" on page 2-57

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.

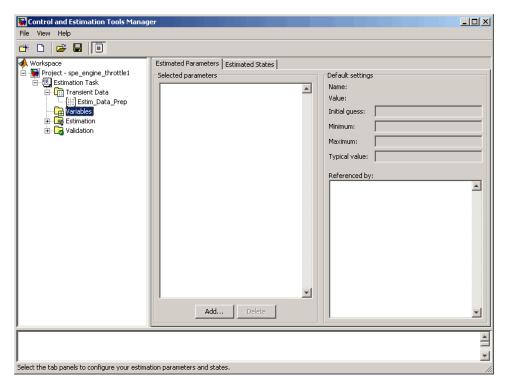
This MAT-file contains the estimation data. To learn more about the data and how to prepare it, see "Prepare Data for Parameter Estimation Using the GUI" on page 2-11.

To load the preconfigured project:

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

🙀 Control and Estimation Tools Mana	ger							
File View Help								
😁 🗅 🚅 🖬 🔟								
📣 Workspace	Input Data Output Data State Data							
Project - spe_engine_throttle1	Assign data to blocks							
Estimation Task	Block Name	Length						
Estim_Data_Prep	spe_engine_thrott	spe_engine_throttle1/Input						
Variables	Channel - 1	input1(:,1)*	time1(:,1)***	1	501/501			
B- Carlination B- Carlination								
		Import	Pre-process	Plot Data	Clear All			
p								
					v			
Select the tabbed panels to configure the tra	nsient data set.				11.			

- 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.

Select Parameters		2
Select additional parameters to est	imate	
Name	Size	
3	1×1	
Kt	1×1	
angle_init	1×1	
angle_open	1×1	
c	1×1	
input_delay	1×1	
k	1×1	
		-
, Specify expression (e.g., s.x, a(3)), b{2}):	
OK N Cancel H	Help	Apply

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.

2

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

This action specifies the selected parameters for estimation.

🙀 Control and Estimation Tools Man	ager							l ×
File View Help								
🍅 🗅 😂 🖬 📋	_							
Workspace Project - spe_engine_throttle1	Data Sets Para Estimation para Name J	Value 0.05	Estimate	Initial Guess J	Minimum -Inf	Maximum +Inf	Typical Value J	1
Variables	input delay	40	\[\begin{bmatrix} 	c input delay	-Inf -Inf	+Inf +Inf	c input delay	
Estimation	k	1		k k	-Inf	+Inf	input_uelay k	
		Use Value as	Initial Guess	Reset to D	efault Settings	Save a	s Default Settings	
- New Estimation node has								
- New Estimation node has	been added	to Estima	tion.					-
Place the tab papels to configure your esti-	mation							

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 -Inf, and +Inf, respectively. This means that the optimization method searches for a solution in the range [-Inf, +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.

🙀 Control and Estimation Tools Manager				<u>_ ×</u>
File View Help				
🚓 🗅 📚 🖪 📳				
Workspace Data Sets Parameters States Estimation	1			
Data sets used for estimation	Outp	put data weighl	ts	
Estimation Task	- E	Block Name	Length	Weight
Lifetim Data Pren			rottle1/Position	
Data Set Selected	Ch	hannel - 1	501/501	1
Estimulation	Y			
- New Estimation node has been added to Estimation.				
- New Estimation node has been added to Estimation.				-
J Load projects				

Validating Model Parameters

After you specify parameters and data for estimation, as described in "Specifying Parameters and Estimation Data" on page 2-53, 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.

Control and Estimation Tools	Manager	_ 🗆 🗵
Workspace Project - spe_engine_throttle Estimation Task Fift] Estim_Data_Pre Variables Estimation Views Views Views Validation	Plot Number Plot Type Plot Title	
I b	Show Plots	
- New View node has bee	has been added to Estimation. en added to Views.	
Configure dynamic views.		11.

File View Help	1anager		
Workspace Project - spe_engine_throttle Estimation Task Transient Data Hill Estim_Data_Pre Variables Estimation Views Views Views Validation	View Setup Select plot types Plot Number Plot 1 Plot 2 Plot 3 Plot 4 Plot 5 Plot 6 Options Estimation New Estimation	Plot Type Measured and simulated Cost function (none) (none) (none) (none) Plot 1 Plot 1 Plot 1	Plot Title
<		Show Plots	
 New Estimation node h New View node has bee Configure dynamic views. 			

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.

5 🗋 🧉 🔚 🗐 👘							
Workspace	Data Sets	Parameters	States Estimation				
Project - spe_engine_throttle	Estimation	n progress					
🗄 🗐 Estimation Task	Itera	tion Function	on Count Cost Fun	ction Step Size	Procedure	T	Estimation Options
[iii] Estim_Data_Prep	2	3	21105	1.3125		- .	
Variables	3	4	8372.9	0.746			Display Options
	4	5	5818.6	0.48151			
New Estimation	5	6	2150	0.79956			
	6	7	860.5	0.41954			
C New View	7	8	459.68	0.32374			Start
	8	9	414.72	0.18391			Julia
	9	10	414.72	9.7901			Show progress views
	10	11	414.72	2.2376			j onow progress views
	11	12	412.69	0.19945			
	12	13	411.31	0.00062433		-	
	Estima Local r Isqnoni	nted paramet ninimum poss lin stopped be		lelay, k the current step is	less than the	T	
						_	
	Ľ						
ration 12 complete							

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.

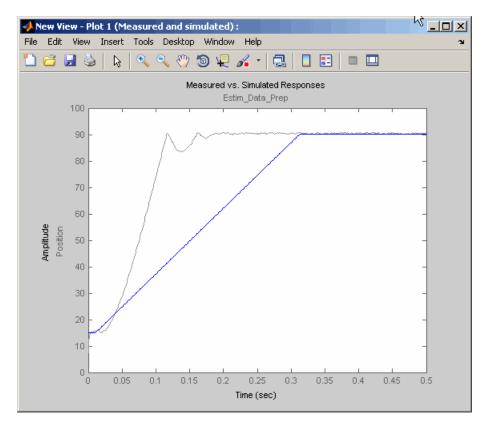
- **Step Size**: Displacement of the optimization method in the current search direction when minimizing the cost function. A final value close to zero indicates that the method has found a local minimum. As shown in the previous figure, the final step size in the last iteration is 5.96e-7.
- **3** View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

🙀 Control and Estimation Tools I	Manager							_ 🗆 ×
File View Help								
😅 🗋 😂 🛃 🗐								
Workspace	Data Sets Para Estimation paran		Estimation					
Estimation Task	Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Value	
Transient Data	3	0.049531		J	-Inf	+Inf]	
Variables	c	40		c	-Inf	+Inf	c	
	input delay	0.001085	<u> </u>	input delay	-Inf	+Inf	input_delay	
New Estimation	k	0.99999	V	k	-Inf	+Inf	k	
E-C Views								
-								
		Use Valu	ue as Initial Gue	ss Reset	to Default Set	tings S	ave as Default Se	ttings
racionation to complete								
Iteration 14 complete Estimation completed.								
Esumation completed.								-
Select the tab panels to configure your	r estimation.							

2

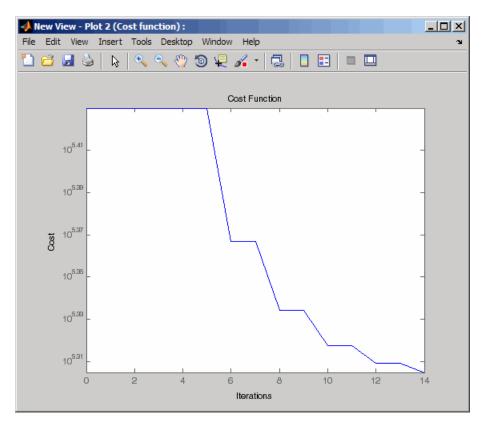
4 Examine the simulated response plot to see how well the simulated output matches the measured output.

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 2-64.

Improving Estimation Results Using Parameter Bounds

- "Strategy for Improving the Estimation Results" on page 2-64
- "How to Specify Parameter Bounds" on page 2-64

Strategy for Improving the Estimation Results

Analyzing the Measured and Simulated and Cost function plots, as described in "Validating Model Parameters" on page 2-57, 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 ${\tt Measured}$ 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 2-51.

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.

🙀 Control and Estimation Tools	Manager							_ 🗆 ×
File View Help								
📸 🗅 😂 🖬 🗐								
📣 Workspace	Data Sets Para	meters States	Estimation					
Project - spe_engine_throttl	Estimation parar	neters						
Estimation Task	Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Value	
Estim_Data_Pre	J	0.0017013	V	J	0	+Inf	J	
Variables	c	39.994	<u> </u>	c	0	+Inf	c	
	input_delay	0.00016981 0.99889	v	input_delay k	0	0.1 +Inf	input_delay k	
New Estimation	ĸ	0.99669	✓	К	U	+107	К	
New View								
E- 🔀 Validation								
-								
		Line Link	ie as Initial Gue		to Default Set		ave as Default Se	things 1
		Use valu	ie as micial Gue	SS Reset	, to berault set	ungs Sa	ave as perault se	adings
Iteration 37 complete								
Estimation 37 complete								<u> </u>
Estimation completed.								-
J Select the tab panels to configure you	r estimation.							

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.

<u> </u>	_							
Workspace		Data Sets Param	eters States E	stimation				
Project - spe_engine_throttle 	F	Estimation progre	SS					
Estimation Task		Iteration	Function Count	Cost Function	Step Size	Procedure		Estimation Options
[iii] Estim_Data_Prep		2	3	21105	1.3125			-
Variables		3	4	8372.9	0.746			Display Options
		4	5	5818.6	0.48151			
- By New Estimation		5	6	2150	0.79956			
		6	7	860.5	0.41954			
New View		7	8	459.68	0.32374			Start
		8	9	414.72	0.18391			Julia
		9	10	414.72	9.7901			Show progress views
		10	11	414.72	2.2376			1 Show progress views
		11	12	412.69	0.19945			
		12	13	411.31	0.00062433		-	
		, 					_	
		Performing (transient estir	nation				
		Active exper	iments: Estim	Data Prep				
			rameters: J,		ŀ			
		Lotimateu pa	nameters. 5, (c, input_domy,	, к			
		Local minimu	m possible.					
		Isononlin ston	ned because th	e size of the c	urrent step is le	ss than the		
			of the step siz					
		Selected value	of the step siz	e tolerance.				
		1					_	

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

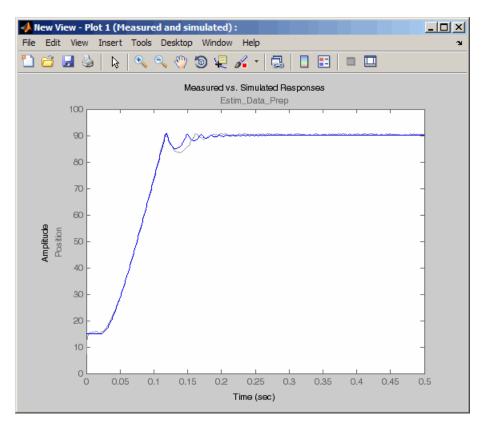
The cost function decreases by 99% from 1.95e5 to 239.94. The final step size 8.75e-7 is close to 0, which indicates that the optimization method has found a local minimum.

- 😽 Control and Estimation Tools Manager _ 🗆 🗙 File View Help 🗃 🛅 🗳 🛃 🔳 Workspace
 Project - spe_engine_throttle
 ⊡ ⊡ ⊡ ⊡ ⊡ Estimation Task Data Sets Parameters States Estimation Estimation parameters Name Initial Guess Minimum Maximum Typical Value Transient Data Value Estimate Intrinsient Data
 Intr **v** 0.19637 J 0 +Inf 1 10.274 ~ 0 +Inf c с input_delay 0.010574 2 0 0.1 input_delay input_delay 1 New Estimation 0.010497 k 0 +Inf k C New View 🗄 🛃 Validation compier * Iteration 12 complete Estimation completed. Select the tab panels to configure your estimation.
- 5 View the estimated parameter values in the Value column of the Parameters tab.

2

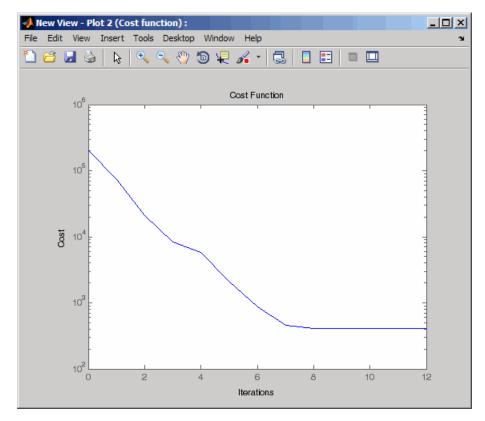
6 Examine the simulated response plot to see how well the simulated output matches the measured output.

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 2-64, 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:

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 2-64.

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 2-14.

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.

Workspace	Input Data Outpu Assign data to blo	it Data State Data -ks			
Estimation Task	Block Name	Data	Time / Ts	Weight	Length
المعالم المعالم المعالم المعالم	spe_engine_th	rottle1/Input			
- [∰] Validation Data1 	Channel - 1	input2(:,1)	time2(:,1)	1	500/500
	Impo	Pre-p	process	Plot Data	Clear All

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.
- 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.

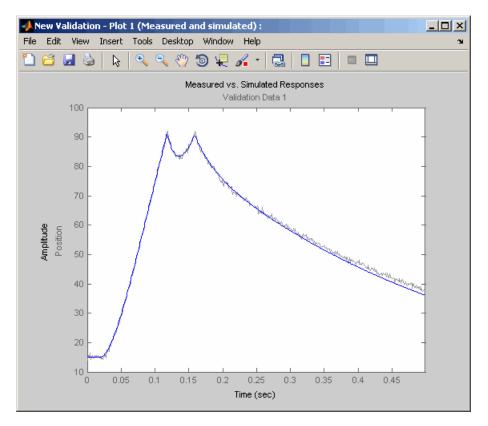
Control and Estimation Tools Manager	r				_ 🗆 ×
Workspace Project - spe_engine_throttle1 Estimation Task Transient Data Fill Stim_Data_Prep Waidation Data 1 Variables Variables Views Views Views Validation New Validation	Plot 2 Re Plot 3 (nc Plot 4 (nc Plot 5 (nc Plot 6 (nc	Plot Type asured and simulated siduals one) one) one) one) Validation Data 1 Validation Data 1 Validation Data 1	Plot 2	Plot Title	
- New Data node has been adde			Show Plots		
 New Validation node has been configure validation plots. 	en added to Valid	ation.			•

f Click Show Plots.

This action opens the following figures:

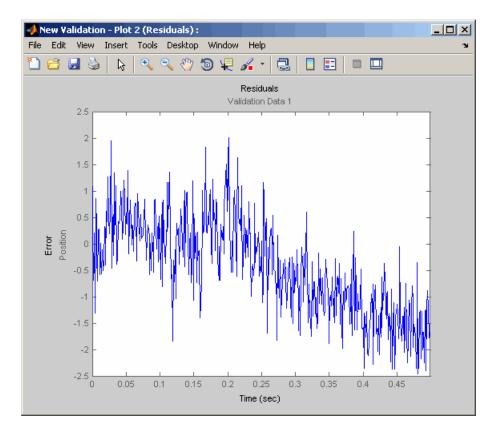
- New Validation Plot 1 (Measured and simulated): Displays the validation data and simulated response.
- New Validation Plot 2(Residuals): Displays the difference between the measured data and simulated response.
- **3** Examine the simulated response plot to see how well the simulated output matches 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**.
- 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
- "Optimize Controller Parameters to Meet Step Response Requirements (GUI)" on page 3-4
- "Optimize Controller Parameters to Meet Step Response Requirements (Code)" on page 3-18
- "Optimize Controller Parameters to Track Reference Signal (GUI)" on page 3-26
- "Design Optimization to Meet Frequency-Domain Requirements (GUI, with Check Block)" on page 3-41
- "Time-Domain Model Verification" on page 3-53

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

"Optimize Controller Parameters to Meet Step Response Requirements (GUI)" on page 3-4

"Optimize Controller Parameters to Meet Step Response Requirements (Code)" on page 3-18

"Optimize Controller Parameters to Track Reference Signal (GUI)" on page 3-26

"Design Optimization to Meet Frequency-Domain Requirements"

"Design Optimization to Meet Frequency-Domain Requirements (GUI, with Check Block)" on page 3-41

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

More About

"How the Optimization Algorithm Formulates Minimization Problems"

"Specifying Time Domain Design Requirements (GUI)"

"Specifying Frequency Domain Design Requirements (GUI)"

Optimize Controller Parameters to Meet Step Response Requirements (GUI)

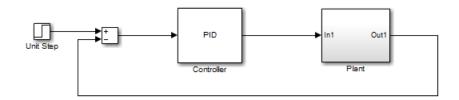
In this section	
-----------------	--

"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

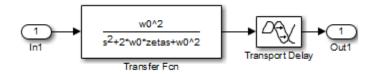
The model sldo_model1 includes the following blocks:



- **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

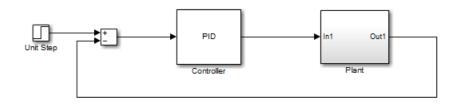
The plant output must meet the following step response requirements:

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

Specify Step Response Requirements

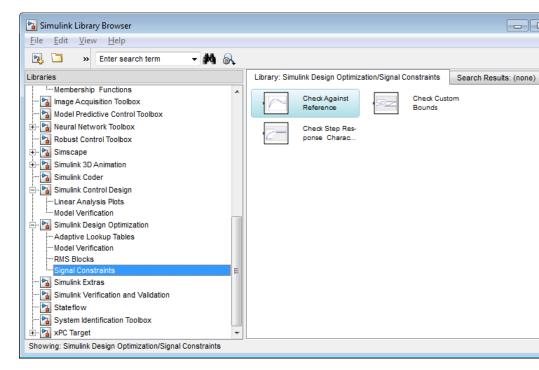
```
1 Open Simulink model.
```

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

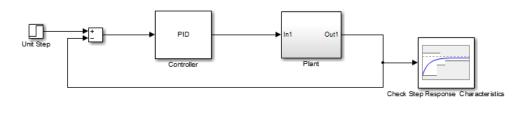


To learn more about the model, see "Model Structure" on page 3-4.

- 2 Add a Check Step Response Characteristics block to the model.
 - **a** In the Simulink model window, select **View > Library Browser**.
 - **b** In the **Libraries** pane, expand the **Simulink Design Optimization** node and select **Signal Constraints**.



- 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.

🚹 Sink Block Parameters: Ch	eck Step Response Chara	cteristics		×				
Check Step Response Characteristics								
Assert that the input signal satisfies bounds specified by step response characteristics.								
Bounds Assertion								
Include step response	e bound in assertion							
Step time (seconds):	0]						
Initial value:	0	Final value:	1					
Rise time (seconds):	5	% Rise:	80					
Settling time (seconds):	7	% Settling:	1					
% Overshoot:	10	% Undershoot:	1					
Enable zero-crossing detection								
Show Plot Show plo	t on block open		Response Optimizat	tion				
0	ОК	Cancel	Help Ap	ply				

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

Sink Block Parameters: Check Step Response Characteristics							
Assert that the input signal satisfies bounds specified by step response characteristics.							
	Assert that the input signal satisfies bounds specified by step response characteristics.						
Bounds Assertion							
✓ Include step response	e bound in assertion	_					
Step time (seconds):	0						
Initial value:	0	Final value:	1				
Rise time (seconds):	2.5	% Rise:	80				
Settling time (seconds):	30	% Settling:	1				
% Overshoot:	5	% Undershoot:	1				
Enable zero-crossing detection							
Show Plot Show plot on block open Response Optimization							
0	OK Cancel Help Apply						

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 3-27.

Specify Design Variables

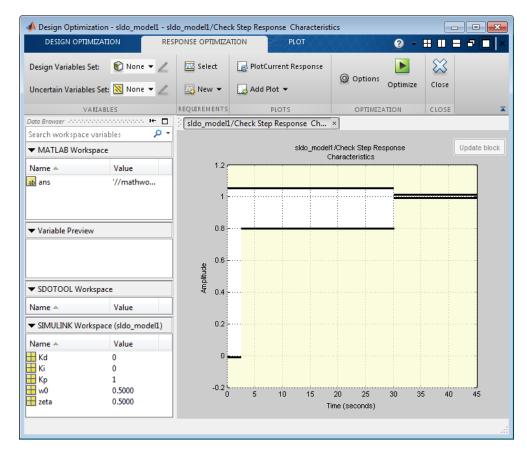
Before you begin this task, specify the design requirements as described in "Specify Step Response Requirements" on page 3-5.

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 **Tools > 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.

DESIGN OPTIMIZATI	RES	PONSE OPTIMIZ		
Design Variables Set:	😭 Non		🔤 Select	Report Plot Current Response
Uncertain Variables Set:	Ne	w	of Design ¥ari	ables Add Plot ▼
VARIABL			variables used k model to optimi	ize PLOTS

A window opens where you specify design variables.

	Variable	Value	Min	Max	Scale	-	Variable	Current value	Used By
						_	Kd	0	sldo model1/Cont
						-	Ki	0	sido model1/Cont
							Кр	1	sldo model1/Contr
							w0	0.5	
		Update m	odel var	iable va	lues		zeta	0.5	sido model1/Plant/
ariable	Detail						Specify expre	ession (e.g., s.x or a(3))	

- **3** Click Kd, Ki and Kp to select them.
- 4 Click to add the selected parameters to a design variables set.

reate D	esign Variables set: DesignVars				
V	Variable	Value	Minimum	Maximum	Scale
1	Kd	0	-Inf	Inf	1
1	Ki	0	-Inf	Inf	1
1	Кр	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 D	esign Variables set: DesignVars				
V	Variable	Value	Minimum	Maximum	Scale
	Kd	0	0	Inf	1
V	Ki	0	0	Inf	1
V	Кр	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	
Design Vars		<3x1 para	
▼ Variable Previ			
	600		
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.

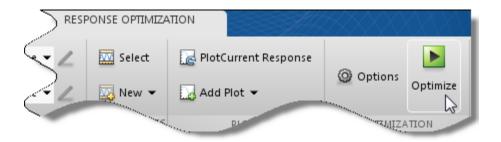
1 (Optional) View the current response of the model. Click

Update block sldo_model1/Check Step Response Characteristics 1.2 1 0.8 0.6 Amplitude 0.4 0.2 0 -0.2 -5 10 15 20 30 35 40 45 50 25 Time (seconds)

🗽 Plot Current Response

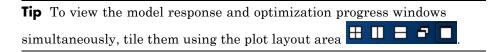
The plot shows that the model output does not meet the specified step response requirements.

2 Click Optimize.



An optimization progress window opens.

Iter	F-count	f(x)	max constraint
timizatio	n started 31-May-2011 17	:14:15	



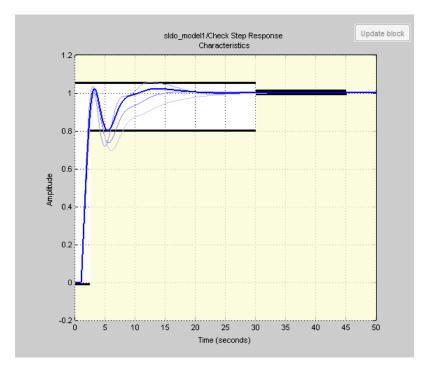
At each optimization iteration, the software simulates the model, and the default optimization solver Gradient descent (fmincon) modifies the design variables to reduce the distance between the simulated response and the design requirement line segments. For more information, see "Selecting Optimization Methods" and "How the Optimization Algorithm Formulates Minimization Problems".

Iter	F-count	f(x)	max
			constraint
0	5	0	431.956
1	12	0	264.342
2	19	0	6.372
3	26	0	1.027
4	33	0	0.133
5	40	0	0.104
6	48	0	0.080
7	55	0	0.003
8	62	0	-1.1153e-0
	started 31-May-2011 16		
timization (converged, 31-May-20	11 17:03:36	
ptimized var	iable values written to	DesignVars' in SDOTOOL workspace	

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.

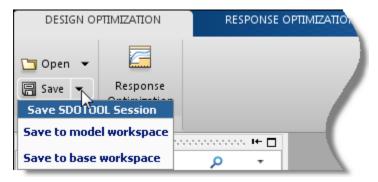
4 View the optimized parameter values. Click DesignVars in Design Optimization Workspace and view the updated values in the Variable Preview area.

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.

	^
Save to Simulink model workspa	ce
	Existing Session Variables
Session: sldo_model1_sdosessi	
	OK Cancel Help

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

- "Optimize Controller Parameters to Meet Step Response Requirements (Code)" on page 3-18
- "Optimize Controller Parameters to Track Reference Signal (GUI)" on page 3-26

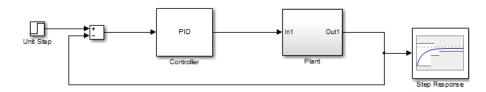
Optimize Controller Parameters to Meet Step Response Requirements (Code)

"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 3-21

This example shows how to optimize controller parameters to meet step response requirements using sdo.optimize. You programmatically modify the default step response requirements specified in a Check Step Response Characteristics block.

Model Structure

The model sldo_model1_stepblk 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.



• The Step Response block is a Check Step Response Characteristics block from the Simulink Design Optimization Signal Constraints library with default step response requirements.

Design Requirements

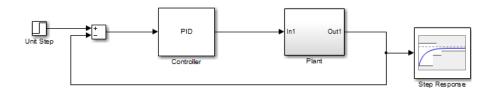
The plant output must meet the following step response requirements:

- 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_stepblk';
open_system(sys);
```



To learn more about the model, see "Model Structure" on page 3-18.

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 = 'sldo_model1_stepblk/Plant';
PlantOutput.OutputPortIndex = 1;
PlantOutput.LoggingInfo.NameMode = 1;
PlantOutput.LoggingInfo.LoggingName = 'PlantOutput';
```

3 Store the logging information.

```
simulator = sdo.SimulationTest('sldo_model1_stepblk');
simulator.LoggingInfo.Signals = PlantOutput;
```

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

4 Get the default requirements specified in Step Response block.

```
allBlkReq = getbounds('sldo_model1_stepblk/Step Response');
StepResp = allBlkReq{1};
```

StepResp is a sdo.requirements.StepResponseEnvelope object.

Note You do not need Check blocks in the Simulink model to specify design requirements. You can specify requirements directly using requirement objects such as sdo.requirements.BodeMagnitude.

5 Specify step response requirements.

```
StepResp.RiseTime = 2.5;
StepResp.SettlingTime = 30;
StepResp.PercentOvershoot = 5;
```

Changing the values does not update them in the block.

Specify Design Variables

Before you begin this task, specify the design requirements as described in "Specify Step Response Requirements" on page 3-19.

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('sldo_model1_stepblk',{'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

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 3-19 and "Specify Design Variables" on page 3-20, respectively.

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 type 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 =
-0.2578
-0.4968
-0.5028
-1.0000
0.7297
0.5095
0.4964
202.3590
```

Some Cleq values are positive which indicates that the response using the current parameter values does not satisfy the design requirements.

3 Specify Simulink model to optimize.

```
opt = sdo.OptimizeOptions;
opt.OptimizedModel = 'sldo_model1_stepblk';
```

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-countf(x)constraintSteplengthderivativeoptimalityProcedure050202.4

1	14	0	200.7	0.341	0	0	infeasible
2	21	0	105	2.24	0	7.51	
3	29	0	117.2	1.57	0	0.521	
4	36	0	79	3.34	0	0	infeasible
5	43	0	27.55	0.632	0	0.152	
6	50	0	50.85	1.06	0	1.14	
7	57	0	11.58	3.52	0	9.5	
8	64	0	3.124	2.48	0	13.9	
9	71	0	0.3022	1.57	0	2.91	
10	78	0	0.1289	0.715	0	7.12	
11	85	0	0.02747	0.489	0	0.199	
12	92	0	0.005082	0.184	0	0.0389	
13	99	0	0.0004325	0.0513	0	0.0092	Hessian modified
1 0 0 0 1		found that a	atiofica the	oonotnointo			

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 the parameter bounds. For more information about the outputs displayed during the optimization, see "Iterative Display" in the Optimization Toolbox documentation.

5 Examine the optimization termination information. This information helps you verify that the response meets the step response requirements.

optInfo

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 are negative 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.

p0pt

```
pOpt(1,1) =
       Name: 'Kp'
      Value: 1.2425
    Minimum: 0
    Maximum: Inf
       Free: 1
      Scale: 1
       Info: [1x1 struct]
pOpt(2,1) =
       Name: 'Ki'
      Value: 0.4037
    Minimum: 0
    Maximum: Inf
       Free: 1
      Scale: 1
       Info: [1x1 struct]
```

pOpt(3,1) =

```
Name: 'Kd'
Value: 2.6710
Minimum: O
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('sldo_model1_stepblk',pOpt);

b Simulate the model.

sim('sldo_model1_stepblk')

Although the requirements are satisfied, the model throws assertion warnings at the MATLAB prompt because the signal does not lies within the assertion tolerance which are tighter than the optimization tolerances. Try the following to eliminate the warnings:

• Continue to search for solutions that are typically located further inside the constraint region.

opt.StopIfFeasible = 'off'

• Increase the optimization solver tolerance.

opt.MethodOptions.TolX = 1.5e-3

Related Examples

- "Optimize Controller Parameters to Meet Step Response Requirements (GUI)" on page 3-4
- "Optimize Controller Parameters to Track Reference Signal (GUI)" on page 3-26

Optimize Controller Parameters to Track Reference Signal (GUI)

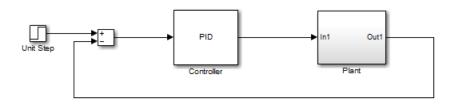
In this section	
-----------------	--

"Model Structure" on page 3-26
"Design Requirements" on page 3-27
"Specify Reference Signal" on page 3-27
"Specify Design Variables" on page 3-34
"Optimize Model Response" on page 3-37

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.



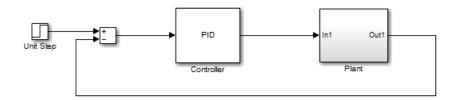
Design Requirements

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.

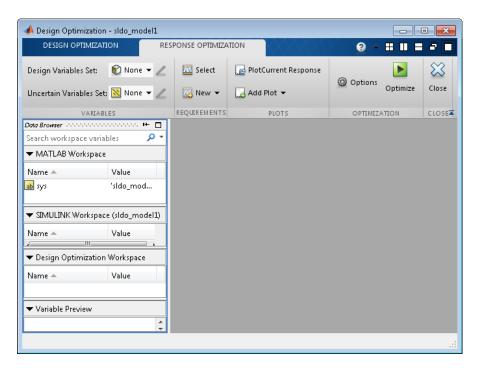
```
sys = 'sldo_model1';
open_system(sys);
```



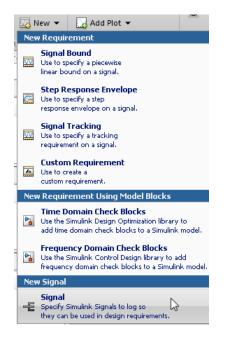
To learn more about the model, see "Model Structure" on page 3-26.

2 In the Simulink model window, select Tools > 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.



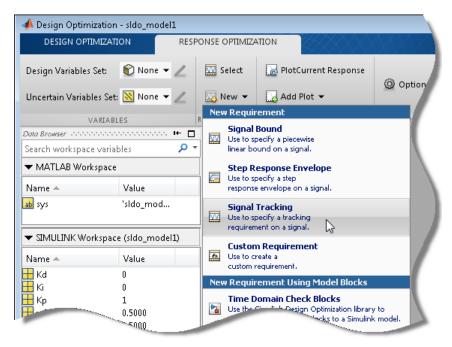
b In the Simulink model window, click the output of the Plant block.The window updates to display the selected signal.

Signal set: Sig	
Currently selected signals	Signal
	Remove Signal
	OK Cancel Help
	.::
elect the signal and click	to add it to the signal set.
	to add it to the signal set.
	to add it to the signal set.
	Signal

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.

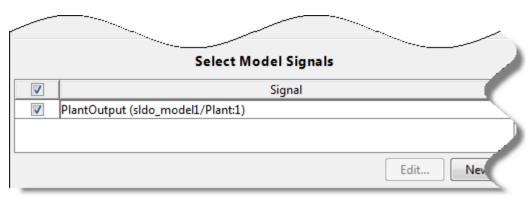
		×					
	Signal Tracking						
	Use to specify a tracking requireme	nt on a signal.					
Requirement Name	: <mark>Req</mark>						
Time vector:	[0;1;2;3;4;5;6;7;8;9;10]						
Amplitude:	[0;0.632120558828558;0.864664716763	387;0.950212931632136;0.981684361:					
		Update reference signal data					
Tracking Method:	Tracking Method: SSE 👻						
	Select Model Signa	als					
	Signal						
PlantOutp	ut (sldo_model1/Plant:1)						
		Edit New					
📝 Create Plot		OK Cancel Help					

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

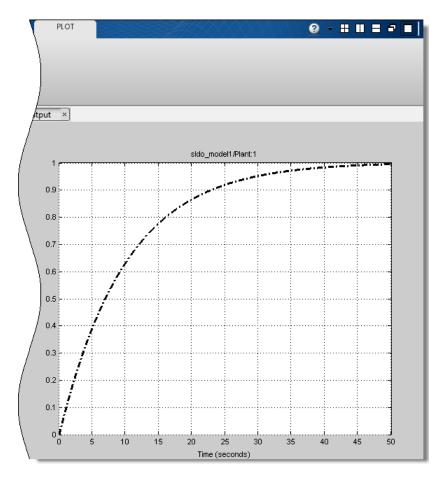
	Signal Tracking	
	Use to specify a tracking requirement o	n a signal.
Requirement Name:	ref_sig	
Time vector:	linspace(0,50,200)	
Amplitude:	1-exp(-0.3*linspace(0,50,200))	
Tracking Method:	SSE 🔻	Update reference signal dat
-	a second s	

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 **Select Model Signals** 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 3-27.

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.

DESIGN OPTIMIZATI	RES	PONSE OPTIMIZ	ATION	
Design Variables Set:	🐑 Non		🚾 Select	RIOTCurrent Response
Uncertain Variables Set:	Ne	w	of Design Varia	ables Add Plot 👻
VARIABL	by	e to specify the Simulin	variables used k model to optimiz	PLOTS

A window opens where you specify design variables.

	Variable	Value	Min	Max Scale		Variable	Current value	Used By
						Kd	0	sldo model1/Cont
					-	Ki	0	sido model1/Cont
						Кр	1	sido model1/Contr
						w0	0.5	
		Update mo	odel var	iable values]	zeta	0.5	sido model1/Plant/
Variable (Detail					Specify expre	ssion (e.g., s.x or a(3))	

- 2 Click Kd, Ki and Kp to select them.
- **3** Click to add the selected parameters to a design variables set.

Create Design Variables set: DesignVars									
V	Variable	Value	Minimum	Maximum	Scale				
1	Kd	0	-Inf	Inf	1				
1	Ki	0	-Inf	Inf	1				
1	Кр	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.

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**.

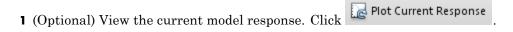
Create Design Variables set: DesignVars									
V	Variable	Value	Minimum	Maximum	Scale				
	Kd	0	0	Inf	1				
V	Ki	0	0	Inf	1				
V	Кр	1	0	Inf	1				

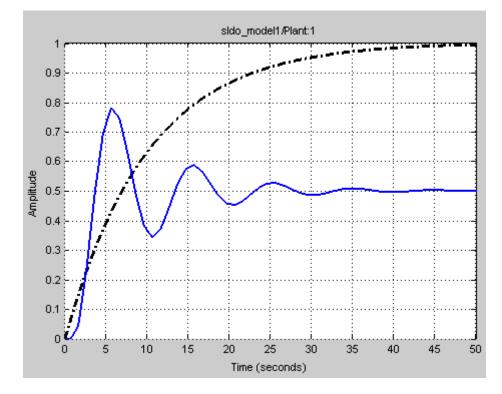
Click **OK**. A new variable **DesignVars** appears in **Design Optimization Workspace** of the Design Optimization tool. You can click the variable to view its contents in the **Variable Preview** area.

🕶 Design Optim	ization	Workspace
Name 🔺		Value
DesignVars		<3x1 para
🕶 Variable Previ	ew	
DesignVars(1	,1) =	
Name:	'Kd'	
Value:	0	
Minimum:	0	
Maximum:	Inf	
Free:	1	
Scale:	1	
Info:	[1x1	struct]

Optimize Model Response

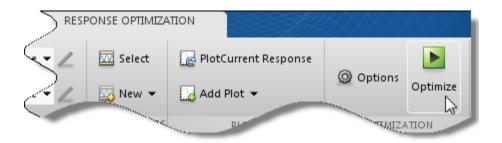
Before you begin this task, you must have specified the reference signal to track and design variables, as described in "Specify Reference Signal" on page 3-27 and "Specify Design Variables" on page 3-34, respectively.





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

2 Click Optimize.



An optimization progress window opens.

	To view the model response and optimization progress windows
sim	ultaneously, tile them using the plot layout area 🖽 🔲 🚍 🗖 🔲.

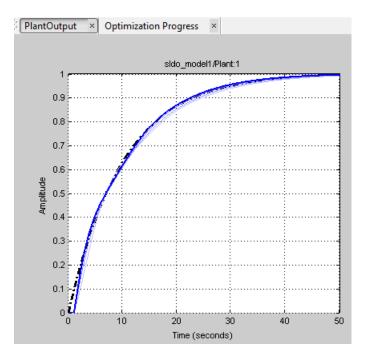
At each iteration, the optimization solver Gradient descent (fmincon) modifies the controller parameters to minimize the error between the simulated response and the reference signal. To learn more, see "How the Optimization Algorithm Formulates Minimization Problems".

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

Iter	F-count	f(x)	max constraint
0	5	30.1990	
1	17	13.9555	
2	25	12.4308	
3	32	7.2900	
4	41	0.6087	-0.066
5	51	0.4931	-0.082
6	59	0.2773	-0.087
7	67	0.2552	-0.094
8	75	0.2477	-0.093
9	83	0.2379	-0.093
10	91	0.2023	-0.091
11	99	0.1455	-0.091
12	107	0.0748	-0.094
13	115	0.0464	-0.097
14	123	0.0422	-0.099
15	131	0.0417	-0.099
		-2011 15:17:42 Jun-2011 15:33:30 witten to 'DesignVars' in SDOTOOL w	[

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

- "Optimize Controller Parameters to Meet Step Response Requirements (GUI)" on page 3-4
- "Optimize Controller Parameters to Meet Step Response Requirements (Code)" on page 3-18

Design Optimization to Meet Frequency-Domain Requirements (GUI, with Check Block)

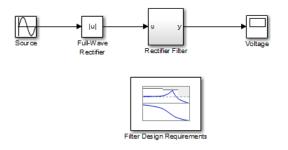
In this section
"Model Structure" on page 3-41
"Design Requirements" on page 3-42
"Specify Design Requirements" on page 3-42
"Specify Design Variables" on page 3-45
"Optimize Design" on page 3-47

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 sdorectifier 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:

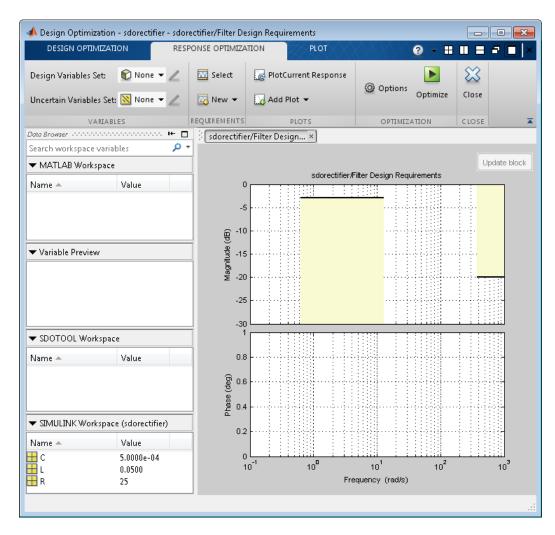
- Have a -3db bandwidth of at least 2Hz
- Limit the gain across the frequency range 2Hz—60Hz to at most 0db
- Limit the gain above 60Hz to at most -20db
- Maximize the filter resistance R
- Minimize the filter inductance L

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')
```



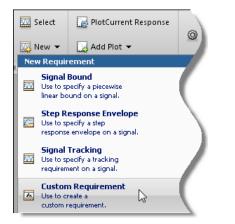
The plot shows the gain and bandwidth requirements specified in the Filter Design Requirements block. See their values in the **Bounds** tab of the Block Parameters dialog box.

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.

Custom R	equirement
during optimization passing	ptimizer will call the specified function handle a structure with fields containing Id the selected simulation signal values.
Requirement Name: Req	
Requirement function: @myCustomRequire	ment Open
Requirement type: <=	

- **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

	Custom Requirement
during	m requirement. The optimizer will call the specified function handle optimization passing a structure with fields containing sign variable values and the selected simulation signal values. xMinRL
Requirement function: Requirement type:	sdorectifier_cost
	Select Model Signals
	Signal

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 3-42.

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.

DESIGN OPTIMIZATI	ом	RES	PONSE OPTIMIZA	ATION
Design Variables Set:	🐑 None		Select	RotCurrent Response
Uncertain Variables Set:		Néw Set w	of Design Varia	Add Plot 👻
VARIABL			variables used k model to optimiz	e PLOTS

A window opens where you specify design variables.

					>
Create Design Variables set: D					
Variable	Value Min Max S		Variable	Current value	Used By
			С	0.0005	sdorectifier/Rectifier
			L	0.05	sdorectifier/Rectifier
		-	R	25	sdorectifier/Rectifier
	Update model variable valu	es			
Variable Detail			Specify expre	ession (e.g., s.x or a(3))	
				ОК	Cancel Help

 ${\bf 2}$ 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 1-500 mH
 - C in the range 1µF-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: DesignVars

			Maximum	Scale
✓ C	0.0005	1e-6	1e-3	0.0005
🔽 L	0.05	1e-3	500e-3	0.5
📝 R	25	1	50	25

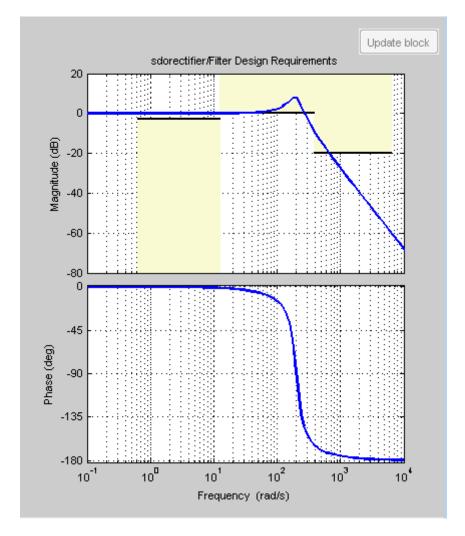
Click **OK**. A new variable **DesignVars** appears in **Design Optimization Workspace** of the Design Optimization tool.

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 3-42 and "Specify Design Variables" on page 3-45, respectively.

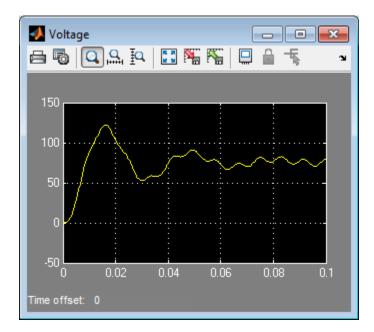
1 (Optional) View the current response of the model. Click

🗽 Plot Current Response

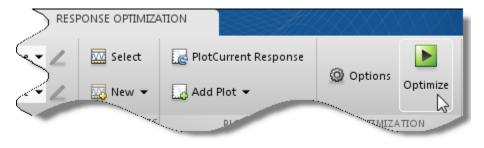


The plot shows that the model does not meet the specified gain and bandwidth requirements.

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.

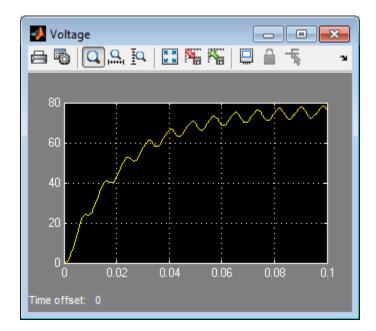


An optimization progress window opens.

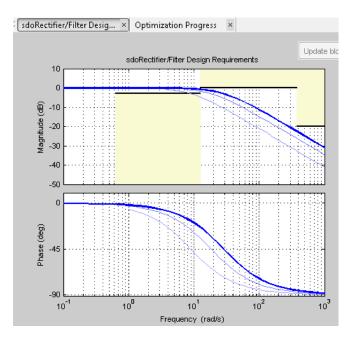
After the optimization completes, the optimization progress window resembles the next figure. After a few iterations, the optimization converges to satisfy the filter bandwidth requirements.

Iter	F-count	f(x)	max constraint
0	8	2	8.1361
1	16	2.2829	0.2409
2	21	1.1305	0
3	29	0.7484	1.2431e-13
4	34	0.7484	0
•)n started 13-Jun-2)n converged, 13-J		
•	variable values wri	tten to 'DesignVars' in SDOTOOL worksp	ace ,

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.

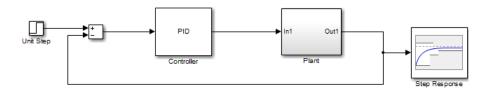
Time-Domain Model Verification

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.

Sink Block Parameters: Check Step Response Characteristics									
Check Step Response Characteristics									
Assert that the input signal satisfies bounds specified by step response characteristics.									
Bounds Assertion									
☑ Include step response	bound in assertion								
Step time (seconds):	0								
Initial value:	0	Final value:	1						
Rise time (seconds):	5	% Rise:	80						
Settling time (seconds):	7	% Settling:	1						
% Overshoot:	10	% Undershoot:	1						
☑ Enable zero-crossing	detection								
Show Plot Show plot	t on block open		Response Optimizatio	n					
0	ОК	Cancel	Help	у					

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 "Optimize Controller Parameters 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" 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:

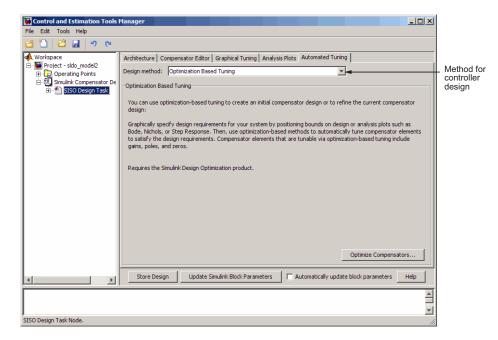
1 Open a SISO Design Tool session by typing the following command at the MATLAB prompt:

sisotool('projectname.mat')

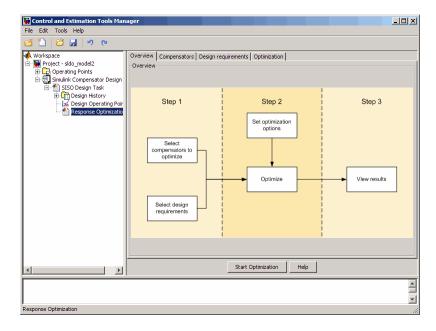
SISO Design Task for SISO Design Tool		Control and Estimation Tools	Manager	
SISO Design Task for SISO Design Tool Image: Siso Design Task image: Siso Design Task Ttle: Project - sido_model2 Subject: Image: Siso Design Task Subject: Image: Siso Design Task Tool Image: Siso Design Task Subject: Image: Siso Design Task Subject: Image: Siso Design Task Image: Siso Design Task Siso Design Task Image: Siso Design Task Image: Siso Design Task Siso Design Task Image: Siso Design Task Image: Siso Design Task Image: Siso Design Task Image: Siso Design Task Image: Siso Design Task Image: Siso Design Task Image: Siso Design Task Image: Siso Design Task				
	SISO Design Task for SISO Design — Tool	Image: Project - sldg_model2 Image: Project - sldg_model2	Title: Subject: Created by: Date modified: Simulink model:	 A N
Project node,		Project node.		

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.



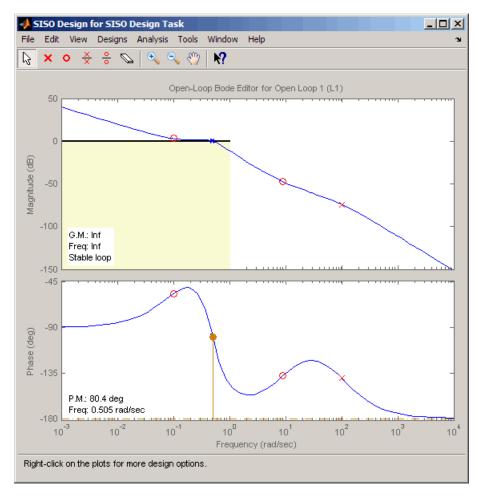
						parame	eter bounds	
Control and Estimation	n To	ools Manager						_ [
ile Edit Tools Help								
ž 🗋 🖬 🖬 🔊	(2						
Workspace		Select compens	npensators Design requirements	Optimization				
Simulink Compens		Coptimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical valu
🖃 🎦 SISO Design			sldo_model2/Controller					
🕀 🛄 Design Hi			Gain	0.1	0.1	-Inf	Inf	0.1
🖂 🚘 Design O			Real Zero	-8.999	-8.999	-Inf	Inf	-8.999
🚹 Response	Ш		Real Zero	-0.10102	-0.10102	-Inf	Inf	-0.10102
	ш		Real Pole	-100	-100	-Inf	Inf	-100
	F	light click on a cor	mpensator name to change its represent.	tion.			Use Value as	: Initial Guess

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

- 4 Specify the design requirements.
 - **a** 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.

Control and Estimation Tools Manage	,				X				
File Edit Tools Help		📣 New Desig	n Requirem	ent			_ 🗆 🗙		
Image: State State Image: State State Image: State State Image: State State Image: State <thimage: state<="" th=""> <thimage: state<="" th=""></thimage:></thimage:>	Overview Compensators Design requirements Optimization Select design requirements to satury Image: Compensators Design requirements Image: Compensators Response pixt	Design requirer Requirement fo	r response:	ters		tude lower lim m Unit Step to	t 🔹	-	Requirement type Response type
Cesign Operating Point		Segments:							
Response Optimization		St	art	Er	nd				
		Frequency	Magnitude	Frequency	Magnitude	Slope	Weight .	┝ -	_Requirement
		1	0	10	0	0	1		values
]	Insert	Delete		
					ОК	Cancel	Help		
	Cick the 'Add new design requirement' button or right clok on a plot to add a new de		Add new design	requirement 1	+	- Add requir	ement		
x	Start Optimization	Неір							
Response Optimization					1				
Kesponse Opcimization					lle				

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



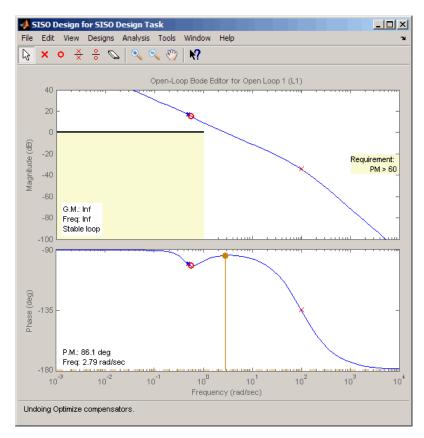
to	ect the requirement enforce during timization	Lists all specified design requirements
Gontrol and Estimation Tools	1anager	
File Edit Tools Help		
Workspace	Overview Compensators Design requirements Optimization	
🕀 🔁 Operating Points	Optimize Response plot	Design requirement
 Simulink Compensator De SISO Design Task 	Open Loop at outport 1 of Controller	
Design History	Open-Loop Bode Editor for Open Loop 1 (L1)	Lower gain limit from 0.001 to 1 rad/sec
L 🗠 Response Optimi	Click the 'Add new design requirement' button or right click on a plot to add a new desig	
I F	Start Optimization Help	
Undo		- -
Undo		1.

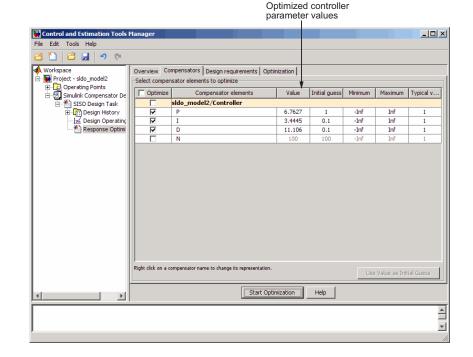
The **Design Requirements** tab also lists the design requirement.

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

- 🙀 Control and Estimation Tools Manager <u>- 🗆 ×</u> File Edit Tools Help 🗃 🛅 🖆 🛃 🔊 🤊 📣 Workspace Overview Compensators Design requirements Optimization Wonspace
 Project - sldo_model2
 Project - sldo_model2
 Project - gloperating Points
 Project - gloperating Points Optimization progress Iteration Eval-Count Cost function Constraint... Step size Procedure Optimization options... 57.12 1.777 0 7 0 8.36 Display options. Design Histor 1 14 0 21 0 1.697 12.1 2 Hessian m... 3 28 0 0 21.7 Optimization progress Constructing optimization problem... Optimization started 10-Nov-2008 14:01:11 ٠ Optimization finished 10-Nov-2008 14:01:21 Successful termination. Found a feasible or optimal solution within the specified Optimization tolerances. status Start Optimization Help ► • 1 ▲ ▼ Redo
- **5** Start the optimization to design the controller by clicking **Start Optimization** in the **Optimization** tab.

- 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" on page 4-15.

Design Optimization-Based PID Controller for Linearized Simulink Model

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

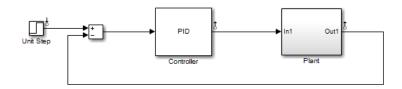
In this tutorial, you learn to use optimization methods to design a PID controller to meet frequency-domain design requirements on a system's response.

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 sldo_model2, 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.

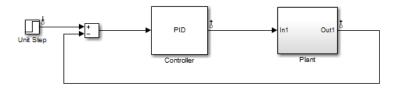
1 Open a SISO Design Tool session for the linearized Simulink model by typing the following command at the MATLAB prompt:

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

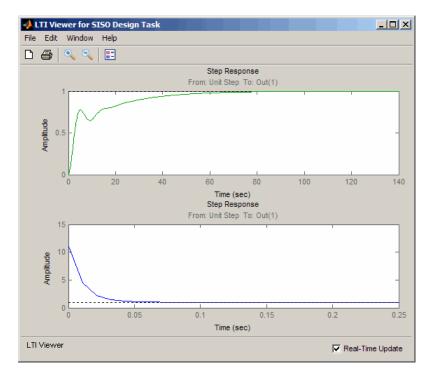


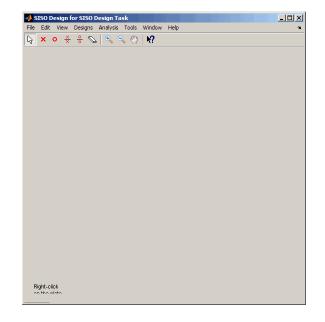
To learn more about the model, see "About the Model" on page 4-15.

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

Gontrol and Estimation Tools	Manager	
File Help		
🗃 🛅 🖨 🔒		
Workspace Project - sido_model2 Operating Points Simulink Compensator De SISO Design Task	Project Settings Title: Subject: Created by: Date modified: Simulink model: Description:	Project - sldo_model2
×>		× ×
Project node.		

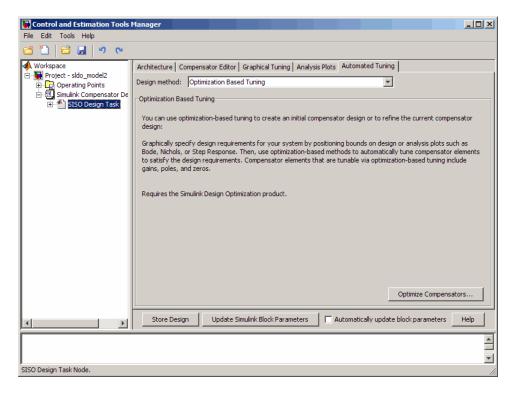
- LTI Viewer for SISO Design Task window, which contains the following plots:
 - Closed-loop step response of the system in the top plot
 - Output of the Controller block in the bottom plot





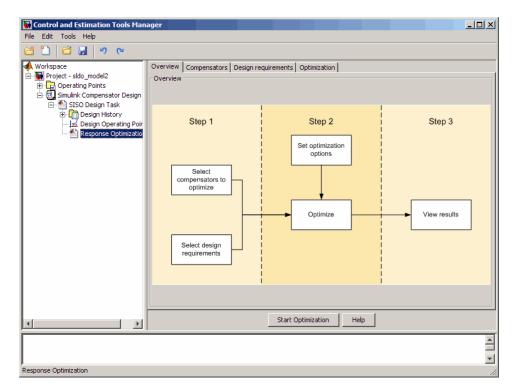
• A blank SISO Design for SISO Design Task window

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.

- 🖬 Project - sldo model2	Overview Co	mpensators Design requirements	Optimization				
Project - sldo_model2	Select comper	sator elements to optimize					
Simulink Compens	C Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical value
🖻 📥 SISO Design 📗		sldo_model2/Controller					
🕀 🛅 Design Hi		Gain	0.1	0.1	-Inf	Inf	0.1
Design O		Real Zero	-8.999	-8.999	-Inf	Inf	-8.999
Response		Real Zero	-0.10102	-0.10102	-Inf	Inf	-0.10102
		Real Pole	-100	-100	-Inf	Inf	-100
	Pipht dick on a C	mmension name to channe its remesent	sino.				
	Right click on a c	ompensator name to change its represent	ation.	n (Helo	1	Use Value at	s Initial Guess

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.

🙀 Control and Estimation Tools Man	ager						_ 🗆 ×
File Edit Tools Help							
🖆 🗋 😂 🛃 🔊 🛯							
Workspace Project - sldo_model2 Operating Points		mpensators Design requirements (nsator elements to optimize	Optimization				
Simulink Compensator Design	Opti	Compensator elements	Value	Initial gu	Minimum	Maximum	Typical
E SISO Design Task		sldo_model2/Controller		·			
Design History		Gain	 Pole/Zero for 		-Inf	Inf	0.1
Design Operating Poir		Real Zero	Parameterize		-Inf	Inf	-8.999
Response Optimizatio		Real Zero	-0.10102	-0.10102	-Inf	Inf	-0.10102
		Real Pole	-100	-100	-Inf	Inf	-100
	Right click on a c	ompensator name to change its representat		Help	Use	Value as Initi	al Guess
		Start Op	otimization	неір			
							* *
Response Optimization							

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.

Control and Estimation Tools Manage File Edit Tools Help	r						<u>_ ×</u>
e 🗋 🖆 🖬 🤊 🖻							
Workspace		pensators Design requirements Optimiz ator elements to optimize	ation				
Operating Points Simulink Compensator Design Task	Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical value
SISO Design Task		sldo_model2/Controller					
🕀 🛄 Design History		P	1	1	-Inf	Inf	1
Design Operating Point		I	0.1	0.1	-Inf	Inf	1
esponse Optimization		D	0.1	0.1	-Inf	Inf	1
		N	100	100	-Inf	Inf	1
	Right click on a cor	npensator name to change its representation.				Jse Value as Ii	itial Guess
		Start O	ptimization	Help			
							▲ ▼
Response Optimization							

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.

ile Edit Tools Help							
≝ " <u>)</u> 🗳 🔛 ୬ ભ							
Workspace		pensators Design requirements Optimiz	ation				
Project - sldo_model2	Select compens	ator elements to optimize					
E Simulink Compensator Design Task	C Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical value
🖃 🎦 SISO Design Task		sldo_model2/Controller					
🕀 📳 Design History		P	1	1	-Inf	Inf	1
Design Operating Point Response Optimization		I	0.1	0.1	-Inf	Inf	1
Response Optimization		D N	0.1 100	0.1	-Inf -Inf	Inf	1
		N	100	100	-11.11	110	1
	Right click on a cor	npensator name to change its representation.				Jse Value as I	nitial Guess
	Right click on a co					Jse Value as In	nitial Guess
< <u> </u>	Right click on a cor		ptimization	Help		Jse Value as Is	nitial Guess
٩۶	Right click on a cor		ptimization	Help		Jse Value as I	nitial Guess

 ${\bf 3}$ In the ${\bf Optimize}$ 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:

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

W Control and Estimation Tools Manage		
File Edit Tools Help		
😅 🖺 😂 🔙 🔊 P		
Workspace	Overview Compensators Design requirements Optimization Select design requirements to satisfy	
Operating Points Jourian Compensator Design Task SISO Design Task Design History Losign Operating Point Response Optimization	Click the 'Add new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design requirement' button or right click on a plot to add a new design	Design requirement
	Show	
	Start Optimization Help	
		4
Response Optimization		

2 Click Add new design requirement.

This action opens the New Design Requirement dialog box.

📣 New Design Requi	irement				_ 🗆 🗙
Design requirement typ	e: Step	response	bounds		•
Requirement for respor	nse: Close	d Loop fr	om Unit S	tep to F	Plant 💌
Design requirement pa	arameters –				
Initial value:	0	Final va	lue:		1
Step time:	0				
Rise time:	5	% Rise:			80
Settling time:	10	% Settli	ng:		1.0000
% Overshoot:	10.0000	% Unde	rshoot:		1
		ок	Cance	el	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.

📣 New Design Requirement							
Design requirement type:			Bode magnitude lower limit				
Requirement for response:			Loop from Unit Step to Controller				
Design requirement parameters Segments:							
Γ	Start Er		d				
	Frequency	Magnitude	Frequency	Magnitude	Slope	Weight	
	1	0	10	0	0	1	
Insert Delete							
				ОК	Cancel	Help	

- **b** Select Open Loop at outport 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.

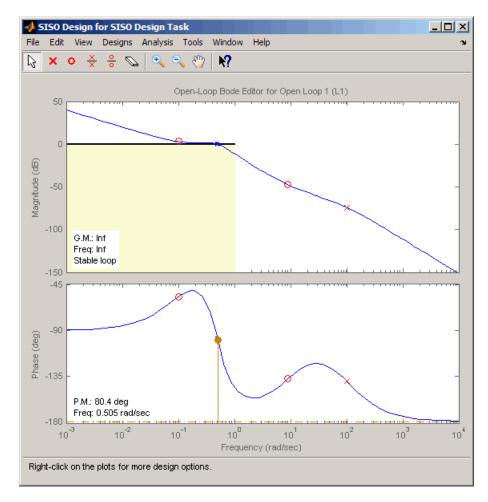
A New Design Requirement							
0	Design requirement type:			Bode magnitude lower limit			
F	Requirement for response:			en Loop at outport 1 of Controller			
Г	Design requirement parameters						
	Segments:						
	Start End		t l				
	Frequency	Magnitude	Frequency	Magnitude	Slope	Weight	
	1.0000e-03	0	1	0	0	1	
Insert Delete							
				ОК	Cancel	Help	

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

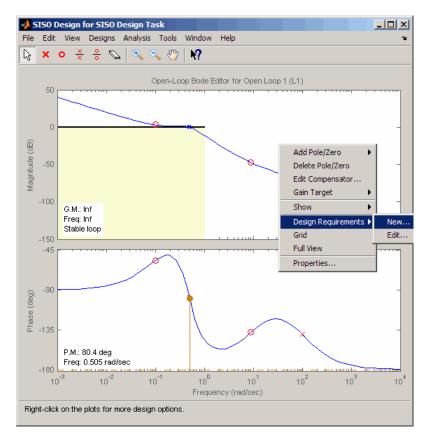
The Bode lower magnitude limit is added to the **Design requirements** tab, as shown in the next figure.

Control and Estimation Tools	fanager and a second	
Workspace Project - sido_model2 Operating Points Simulink Compensator De Sisson History Kasponse Optimi	Overview Compensators Design requirements Optimization -Select design requirements to satisfy Optimize Response plot Image: Optimize of Open Loop at outport 1 of Controller Image: Open-Loop Bode Editor for Open Loop 1 (L1)	Design requirement
	Click the 'Add new design requirement' button or right click on a plot to add a new desig	
I D	Start Optimization Help	
Undo		h

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:
 - **a** 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.

📣 New Desig	jn Requirem	ent			_ 🗆 🗙				
Design requirement type: Upper gain limit									
-Design require	Design requirement parameters								
Segments:									
St	art	Er	nd						
Frequency	Magnitude	Frequency	Magnitude	Slope	Weight				
1	0	10	0	0	1				
				Insert	Delete				
	OK Cancel Help								

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.

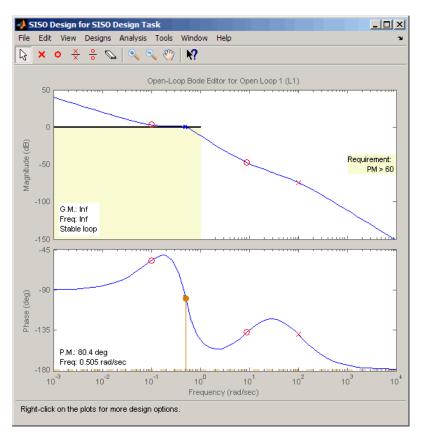
📣 New Design Requirement 📃 🔲 🗙							
Design requirement ty	/pe:in & Ph	nase margins	-				
Design requirement parameters							
Gain margin >			20				
Phase margin >			30				
OK	Cano	el He	lp				

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

📣 New Design Requirement 📃 🗌 🗙							
Design requirement ty	ype:in & Phase margins 💌						
Design requirement parameters							
Gain margin >	20						
I Phase margin >	60						
ок	Cancel Help						

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.

Sontrol and Estimation Tools	Manager				_ 🗆 ×
File Edit Tools Help					
🗂 🗋 🖬 🔊 🍋					
Workspace		mpensators Design requirements	Optimization		
Project - sldo_model2 Operating Points	Select design i	requirements to satisfy			
Simulink Compensator De	Coptimize	Response p	lot	Design require	ement
🖃 🏝 SISO Design Task		Open Loop at outport 1 of Cor			
🕕 🛅 Design History		Open-Loop Bode Editor for Open		Lower gain limit from 0.00	01 to 1 rad/sec
Design Operating		Open-Loop Bode Editor for Open	Loop 1 (L1)	Requirement: PM > 60	
	Click the 'Add ne	w design requirement' button or right clic	Show P		equirement



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

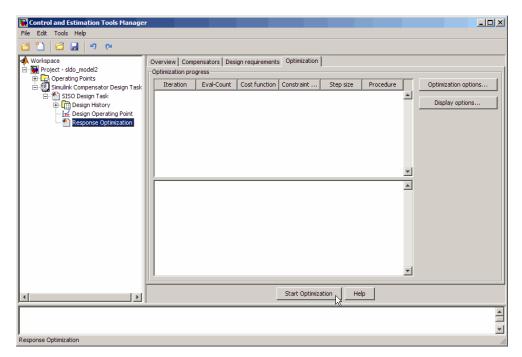
Designing the Controller

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.

Control and Estimation Tools	Manager	×
 Workspace Project - sldo_model2 Operating Points Simulink Compensator De SISO Design Task Design History Design Operating Response Optimi 	Overview Compensators Design requirements Optimization Optimization progress Iteration Eval-Count Construction Optimization process 0 7 0 57.12 Iteration Display options Display options Optimization problem Optimization started 12-Nov-2008 11:44:04 Iteration	
	Stop Optimization Help	
		*
Response Optimization		

- 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.

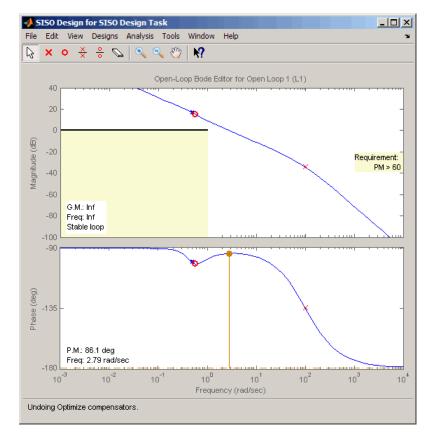
Control and Estimation Too Edit Tools Help	ols Manager							
1 🗳 🖬 🤊 e	i .							
Workspace Dect - sldo_model2	Overview Con Optimization pr		esign requirem	ents Optimiza	ition			
Operating Points Simulink Compensator	Iteration	Eval-Count	Cost function	Constraint	Step size	Procedure		Optimization options
SISO Design Task	0	7	0	57.12				
Design History	1	14	0	1.777	8.36			Display options
🚽 Design Opera	2	21	0	1.697	12.1	Hessian m]	
Response Opl	3	28	0	0	21.7			
			v-2008 14:01:2	1 · optimal solutio	on within the s	pecified	¥	
			[Start Optimiz	zation	Help		

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.

Control and Estimation Tools	Manager						<u>_ X</u>	
File Edit Tools Help								
🖆 🗋 🖨 📕 🤊 (°								
Workspace	Overview Cor	mpensators Design requirements Optir	mization					
Project - sldo_model2	Select compen	Select compensator elements to optimize						
Simulink Compensator De	C Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical v	
SISO Design Task		sldo_model2/Controller	1					
🕂 📆 Design History		P	6.7627	1	-Inf	Inf	1	
		I	3.4445	0.1	-Inf	Inf	1	
Response Optimi		D	11.106	0.1	-Inf	Inf	1	
		N	100	100	-Inf	Inf	1	
	Right click on a co	ompensator name to change its representation.			Use	e Value as Init	ial Guess	
		Start Opti	mization	Help				
								
I							-	
							//	

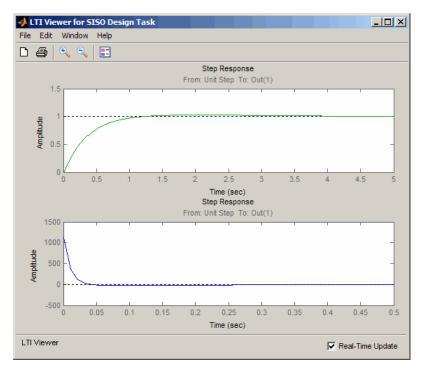
b Examine the system's response on the following plots:



• 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.





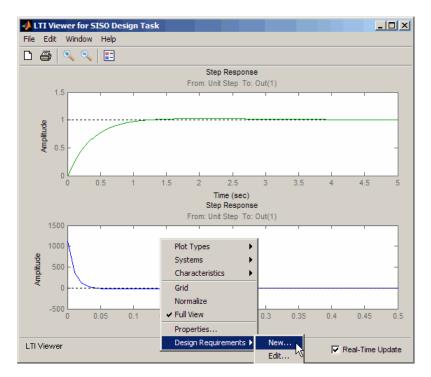
- The top plot shows that the closed-loop response of the system is stable. The system with the designed controller thus meets both the magnitude and phase margin requirements.
- The bottom plot in the LTI Viewer shows that the peak value of the controller's output is 1000, which is very large and can 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.

📣 New Design Requirement								
Design requirement type: Step response bounds								
Design requirement parameters								
Initial value:	Final value: 1							
Step time:	2							
Rise time:	5 % Rise: 80							
Settling time: 10	% Settling: 1.0000							
% Overshoot: 10.0000	% Undershoot: 1							
Oł	Cancel Help							

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

•	🙏 New Desig	ın Requirem	ent			_ 🗆 🗙					
	Design requirement type: Upper time response bound										
Г	Design requirement parameters										
	Segments:										
	Sta	art	Er	nd							
l	Time	Amplitude	Time	Time Amplitude		Weight					
I	0	1	10	1	0	1					
	1										
	Insert Delete										
				ОК	Cancel	Help					

- 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.

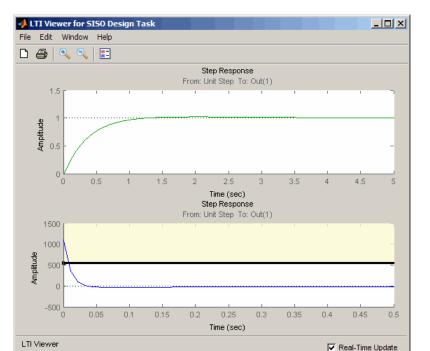
The New Design Requirements dialog box resembles the following figure.

New Design Requirement									
Design requirement type: Upper time response bound									
	Design requirement parameters								
Segments:									
St	art	Er	nd						
Time	Amplitude	Time	Amplitude	Slope	Weight				
0	550	Inf	550	0	1				
	Insert Delete								
			ОК	Cancel	Help				

 $f \quad {\rm Click} \ OK \ {\rm to} \ {\rm close} \ {\rm the} \ {\rm dialog} \ {\rm box}.$

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.

Control and Estimation File Edit Tools Help	Tools Manager		_ 🗆 X					
🖆 🗋 😂 🖬 🔊	С ^и							
Workspace	Overview Com	pensators Design requirements Optimization						
🖻 🙀 Project - sldo_model2	-Select design re	quirements to satisfy						
Operating Points Simulink Compens	Optimize	Response plot	Design requirement					
E SISO Design		Closed Loop from Unit Step to Controller						
⊡ 🛄 Design Hi	<u> </u>	Step Response	Upper time response bound from 0 to 10 sec					
🚽 🛃 Design O		Open Loop at outport 1 of Controller						
	V	Open-Loop Bode Editor for Open Loop 1 (L1)	Lower gain limit from 0.001 to 1 rad/sec					
		Open-Loop Bode Editor for Open Loop 1 (L1)	Requirement: PM > 60					
	Click the 'Add new	design requirement' button or right click on a plot to add a new design rec	quirement.					
		Show Pl	ots Add new design requirement					
		Start Optimization Help						
			4					



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:
 - a 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.

•	🙏 New Desig	ın Requirem	ent			_ 🗆 X					
	Design requirement type: Lower time response bound										
Г	Design requirement parameters										
	Segments:										
I	Sta	art	Er	nd							
	Time	Amplitude	Time	Amplitude	Slope	Weight					
I	0	1	10	1	0	1					
	Insert Delete										
-				ОК	Cancel	Help					

- c In the Time field of the End column, enter Inf.
- $\boldsymbol{\mathsf{d}}\$ In the $\boldsymbol{\mathsf{Amplitude}}\ field of the <math display="inline">\boldsymbol{\mathsf{Start}}\ column,$ enter -250.

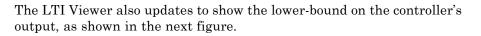
e In the Amplitude field of the End column, enter -250.

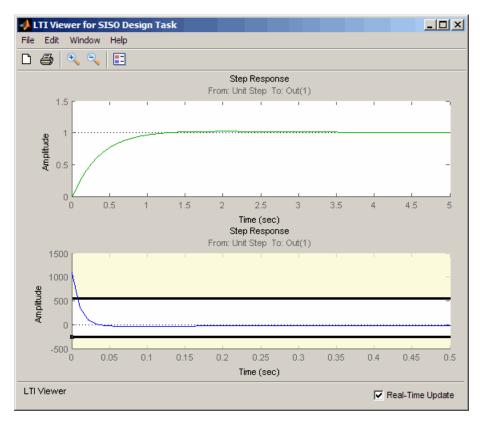
The New Design Requirement dialog box resembles the next figure. Click **OK** to close the dialog box.

📣 New Desig	n Requirem	ent			_ 🗆 X		
Design require	ement type: Lo	wer time resp	onse bound		v		
-Design require	ement parame	ters					
Segments:	Segments:						
Sta	art	En	nd				
Time	Amplitude	Time	Amplitude	Slope	Weight		
0	-250	Inf	-250	0	1		
				Insert	Delete		
			ОК	Cancel	Help		

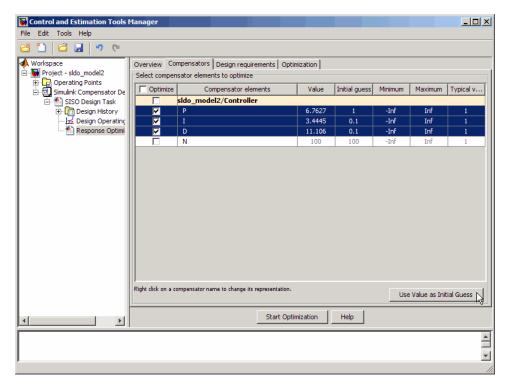
The **Design requirements** tab in the Control and Estimation Tools Manager GUI updates. It now displays the lower-limit requirement, as shown in the next figure.

Control and Estimation	Tools Manager		X			
	-					
	6					
Workspace		pensators Design requirements Optimization				
Project - sldo_model2 Operating Points	Select design re	equirements to satisfy				
Simulink Compens	Coptimize	Response plot Design requirement				
🖻 🌯 SISO Design		losed Loop from Unit Step to Controller				
🕀 🛄 Design Hi		Step Response	Upper time response bound from 0 to 10 sec			
Design O		Step Response	Lower time response bound from 0 to 10 sec			
🛃 Response		Open Loop at outport 1 of Controller Open-Loop Bode Editor for Open Loop 1 (L1)	Lower gain limit from 0.001 to 1 rad/sec			
		Open-Loop Bode Editor for Open Loop 1 (L1) Open-Loop Bode Editor for Open Loop 1 (L1)	Requirement: PM > 60			
			Requirement. THE Sol			
	Click the 'Add new	design requirement' button or right click on a plot to add a new design rec	uirement.			
			·			
		Show Pl	ots Add new design requirement			
		Start Optimization Help	1			
		Start Optimization Help				
			<u> </u>			
1			•			
			///			





- **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.

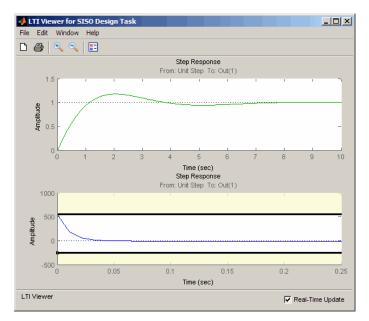
Clicking **Use Value as Initial Guess** updates the values in the **Initial Guess** column, as shown in the next figure.

🙀 Control and Estimation Tools Manager							
File Edit Tools Help							
📣 Workspace		npensators Design requirements Optim	nization				
Project - sldo_model2	Select compension	sator elements to optimize					
E- D Simulink Compensator De	Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical v
SISO Design Task		sldo_model2/Controller					
🕀 📺 Design History	V	P	6.7627	6.7627	-Inf	Inf	1
		I	3.4445	3.4445	-Inf	Inf	1
Response Optimi	V	D	11.106	11.106	-Inf	Inf	1
		N	100	100	-Inf	Inf	1
	Right click on a co	mpensator name to change its representation.	ination	Help	Use	: Value as Init	ial Guess
Start Optimization Help							
							A
							-
Response Optimization							

- **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.

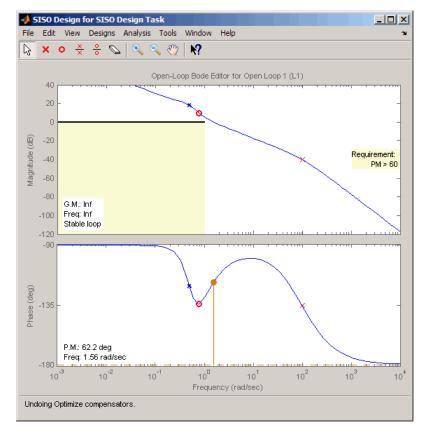
Control and Estimation Tools	1anager	<u>_</u> _×
Workspace Project - sldo_model2 Derating Points Simulink Compensator De SISO Design Task Design History Design Operating Response Optimi	Overview Compensators Design requirements Optimization Optimization progress Iteration Eval-Count Cost funct Constrain Step size Procedure 0 14 0 1.031 Iteration Iteration Iteration 1 28 0 2.914e-014 5.67 Iteration Iteraticacaa Iteraticaa <t< td=""><td>Optimization options Display options</td></t<>	Optimization options Display options
	Start Optimization Help	

- 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.

♦ Workspace ∃ ₩ Project - sldo_model2	· · · · · · · · · · · · · · · · · · ·	Overview Compensators Design requirements Optimization Select compensator elements to optimize						
Operating Points Image: Comparison of Comp	Optimize		Value	Initial guess	Minimum	Maximum	Typical v.	
SISO Design Task		sldo model2/Controller						
Design History	V	P	6.706	6.7627	-Inf	Inf	1	
🚽 🛃 Design Operating	V	I	3.4445	3.4445	-Inf	Inf	1	
📥 Response Optimi		D	5.4329	11.106	-Inf	Inf	1	
		N	100	100	-Inf	Inf	1	
	Right click on a c	compensator name to change its representation.				a Value as Trij	tial Granes	
4	Right click on a c		timization	Help	Use	e Value as Inii	tial Guess	

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.

📣 Save Projects	
Projects:	
Project - sldo_model2	
	_
OK Cancel	Help

- 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.



Examples

Use this list to find examples in the documentation.

Getting Started

"Prepare Data for Parameter Estimation Using the GUI" on page 2-11 "Estimate Parameters from Measured Data Using the GUI" on page 2-47 "Optimize Controller Parameters to Meet Step Response Requirements (GUI)" on page 3-4

"Optimize Controller Parameters to Meet Step Response Requirements (Code)" on page 3-18

"Optimize Controller Parameters to Track Reference Signal (GUI)" on page 3-26

"Design Optimization-Based PID Controller for Linearized Simulink Model" on page $4{\text -}15$

Index

С

configuring control design 4-17 Control and Estimation Tools Manager 4-17

D

design requirement line segments 3-5 design requirements time-domain 3-5 3-27 designing optimization-based linear controller 4-37

L

linear control design 4-17

0

optimization-based control design 4-15

Ρ

plots 4-27

R

response optimization reference signal using GUI 3-26 step response using GUI 3-4 tutorial using functions 3-18

S

Simulink Design Optimization product related products 1-7 required products 1-7 SISO Design Task 4-17 specifying controller parameters 4-22 specifying design requirements 4-27

T

time-domain design requirements 3-19