Simulink® Response Optimization

For Use with Simulink®

■ Modeling

■ Simulation

■ Implementation

Getting Started

The MathWorks

Version 3

How to Contact The MathWorks:

	www.mathworks.com	Web
	comp.soft-sys.matlab	Newsgroup
	www.mathworks.com/contact_TS.html	Technical Support
	suggest@mathworks.com	Product enhancement suggestions
	bugs@mathworks.com	Bug reports
	doc@mathworks.com	Documentation error reports
	service@mathworks.com	Order status, license renewals, passcodes
	info@mathworks.com	Sales, pricing, and general information

☎ 508-647-7000 (Phone)

📠 508-647-7001 (Fax)

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

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

Getting Started with Simulink Response Optimization

Trademarks

MATLAB, Simulink, Stateflow, Handle Graphics, Real-Time Workshop, and xPC TargetBox are registered trademarks of The MathWorks, Inc.

Other product or brand names are trademarks or registered trademarks of their respective holders.

Patents

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

Contents

What Is Simulink Response Optimization?

1

Response Optimization Tutorial

2

Response Optimization Using Functions

3

Troubleshooting

4

What Is Simulink Response Optimization?

Introduction

Simulink Response Optimization provides a graphical user interface (GUI) to assist in the tuning and optimization of control systems and physical systems. With this product, you can either directly tune response signals within Simulink models or tune responses of LTI systems within a SISO Design Task (requires the Control System Toolbox).

Tuning within Simulink Models

You can tune parameters within a nonlinear Simulink model to meet time-domain performance requirements by graphically constraining signals within a time-domain window or tracking and closely matching a reference signal.

You can tune any number of Simulink variables including scalars, vectors, and matrices. In addition, you can place uncertainty bounds on other variables in the model for robust design. Simulink Response Optimization makes attaining performance objectives and optimizing tuned parameters an intuitive and easy process.

To use Simulink Response Optimization to directly tune Simulink models, you need only to include a special block, the Signal Constraint block, in your Simulink diagram. Just connect the block to any signal in the model to signify that you want to place some kind of constraint on the signal. Simulink Response Optimization automatically converts time-domain constraints into a constrained optimization problem and then solves the problem using optimization routines taken from the Optimization Toolbox or the Genetic Algorithm and Direct Search Toolbox. The constrained optimization problem formulated by Simulink Response Optimization iteratively calls for simulations of the Simulink system, compares the results of the simulations with the constraint objectives, and uses gradient methods to adjust tuned parameters to better meet the objectives.

Two additional blocks, CRMS and DRMS, compute the continuous and discrete cumulative root mean square values of signals. Use them with the Signal Constraint block to optimize the cumulative root mean square of signals in your model.

Tuning within a SISO Design Task

When you have the Control System Toolbox installed, you can design compensators for control systems by tuning compensator elements or parameters within a SISO Design Task in the Control and Estimation Tools Manager. You can tune any elements or parameters, such as poles, zeros, and gains, within any compensators in the system to optimize the responses of both open and closed loops.

Optimize the responses of systems in the SISO Design Task to meet both time- and frequency-domain performance requirements by graphically constraining signals:

• Add frequency-domain design requirements to plots such as root-locus, Nichols, and Bode in the SISO Design Task graphical tuning editor called SISO Design Tool.

• Add time-domain design requirements to plots such as step or impulse response (when displayed within the LTI Viewer as part of a SISO Design Task).

You can use response optimization within a SISO Design Task in the Control and Estimation Tools Manager to tune both command-line LTI models as well as Simulink models:

• Create an LTI model using the Control System Toolbox command-line functions and use the sisotool function to create a SISO Design Task for the model.

• Use a Simulink Compensator Design task (from the product Simulink Control Design) to automatically analyze the model and then create a SISO Design Task for a linearized version of the model. You can then use the response optimization tools within the SISO Design Task to tune the response of the linearized Simulink model.

When using response optimization within a SISO Design Task you cannot add uncertainty to system parameters.

Applications

You can use Simulink Response Optimization for a variety of applications. Possible uses include

- Designing and optimizing control systems by tuning compensator elements such as poles, zeros, and gains.

- Designing physical systems by adjusting parameters in your system. For example, adjust the dimensions of physical devices such as robot arms or hydraulic pistons, tune properties of materials such as thermal emissivity, etc.

- Closely tracking a reference, or desired, signal (using the Signal Constraint block only).

- Optimizing responses for systems that include physical actuation limits and constraints on state/variable values.

- Minimizing the energy in a system by minimizing the root-mean-square signal.

- Including uncertainty in your parameter values to take into account imperfect knowledge of certain physical parameters in your model (using the Signal Constraint block only).

System Requirements

Simulink Response Optimization has the same system requirements as MATLAB®. Refer to the MATLAB documentation for details. For a list of required and related products, see the Simulink Response Optimization product page on the MathWorks Web site at `http://www.mathworks.com/products/simresponse/`.

Upgrading from the Nonlinear Control Design Blockset

Simulink Response Optimization replaces the Nonlinear Control Design (NCD) Blockset. For information on upgrading from the Nonlinear Control Design Blockset, refer to the Release Notes. In addition, the `ncdupdate` reference page has detailed information on updating NCD models.

Using the Documentation

Expected Background

Users of this guide should be familiar with dynamic systems design and analysis, and have experience creating Simulink models.

How to Use This Guide

To get started using Simulink Response Optimization, read Chapter 2, "Response Optimization Tutorial" which introduces the main product features and uses three simple examples to describe their use.

To use both time- and frequency-domain design requirements to optimize the response of LTI systems in a SISO Design Task (requires the Control System Toolbox), read "Frequency Domain Response Optimization Example" on page 2-23.

To perform response optimization using functions, read Chapter 3, "Response Optimization Using Functions".

For advice on solving typical problems, read Chapter 4, "Troubleshooting".

If you are upgrading from the Nonlinear Control Design (NCD) Blockset, read the Release Notes to learn about new features and how to convert your NCD models for use with Simulink Response Optimization.

Online Documentation

Further documentation is available online, including function and block references, and detailed discussions on setting up and running a response optimization.

Response Optimization Tutorial

You can use Simulink Response Optimization to solve a wide variety of control and physical design problems by either inserting Signal Constraint blocks in your model or creating a SISO Design Task for your system, and then using the graphical user interface (GUI) to specify constraints and design requirements for these signals and systems. You can specify tuned and optimization settings as well as uncertain parameters (with the Signal Constraint block only). To get started with Simulink Response Optimization, work through the three examples in this chapter or use the Quick Start guides.

Quick Start – Time Domain Response Optimization of Simulink Models

If you want to quickly get started using Simulink Response Optimization for time domain response optimization of Simulink models, this section summarizes the response optimization process:

1 Make a model of your (nonlinear) system and controller using Simulink. Add input signals (e.g., steps, ramps, observed data) for which you know what the desired output should look like.

2 Attach a Signal Constraint block to the signals you want to constrain. The Simulink library srolib contains the Signal Constraint block. To open the library, just type srolib at the MATLAB prompt or select **Simulink Response Optimization** from the Simulink Library Browser.

3 If the model's parameters do not already exist in the MATLAB workspace, initialize these parameters in the workspace with a best first guess.

4 Double-click each Signal Constraint block in your system to open the Signal Constraint window for each constrained output. Choose a method for constraining the response signal by selecting **Enforce signal bounds** and/or **Track reference signal** at the bottom of the window.

5 Within the Signal Constraint window, constrain each signal by positioning constraint bound segments and/or tracking a reference signal. You can position constraint bound segments by clicking and dragging the segment or right-clicking the segment and selecting **Edit** from the menu. Plot reference signals by selecting **Goals > Desired Response** from the Signal Constraint window and entering vectors of data for the reference signal.

6 Open the Tuned Parameters dialog box by selecting **Optimization > Tuned Parameters** within a Signal Constraint window. Click the **Add** button to add parameters to the list. Select a parameter in the list to specify initial guesses and maximum and minimum values.

7 Optional: Open the Uncertain Parameters dialog box by selecting **Optimization > Uncertain Parameters** within a Signal Constraint window. Click the **Add** button to add parameters to the list. Choose a

method and range for sampling the uncertain parameters and select which responses to optimize.

8 Optional: Save the project, including constraints, tuned parameters, uncertain parameters, and settings for optimization and simulation, to a file, to the MATLAB workspace, or to the model workspace by selecting **File > Save**. You can retrieve previously saved projects by selecting **File > Load**. To automatically save and reload the project with the Simulink model, select the check box at the bottom of the Save dialog box. Note that when saving the project, you are saving the *entire* optimization project, which might include several Signal Constraint blocks.

9 Click the **Start** button in the toolbar or select **Start** from the **Optimization** menu to optimize the response signal by adjusting the tuned parameters. The Optimization Progress window displays the new, optimized parameter values.

Quick Start – Mixed Time and Frequency Domain Response Optimization

This section outlines the response optimization process for linear systems within a SISO Design Task in the Control and Estimation Tools Manager using both time and frequency domain design requirements:

1 Create and import a linear model into a SISO Design Task using one of two methods:

- Create an LTI model of your system at the MATLAB command line and use the `sisotool` function to create a SISO Design Tool for the model.

- Create a Simulink model and use a Simulink Compensator Design Task with the product Simulink Control Design to automatically linearize the model and create a SISO Design Task for the linearized model.

2 Optional: Within the **SISO Design Task** node of the Control and Estimation Tools Manager, select the **Architecture** pane and then select a control architecture for your system.

3 Use the **Graphical Tuning** pane within the **SISO Design Task** node of the Control and Estimation Tools Manager to create any design plots you want to use to design the response of the system, such as root-locus diagrams, Bode plots, etc. Use the **Analysis Plots** pane to create any analysis plots you want to use to view the system, such as step or impulse response plots.

4 Within the **Automated Tuning** pane select `Optimization based tuning` as the **Design Method** and then click the **Optimize Compensators** button to create a **Response Optimization** task within the Control and Estimation Tools Manager.

5 Within the **Response Optimization** node, select the **Compensators** pane to select and configure the compensator elements you want to tune during the response optimization. Note that when optimizing responses in a SISO Design Task, you cannot add uncertainty to parameters or compensator elements.

6 Within the **Design requirements** pane of the **Response Optimization** node, select the design requirements you want the system to satisfy. To

add a new design requirement to a design or analysis plot, click the **Add new design requirement** button and then complete the New Design Requirement dialog box.

7 Optional: To change the default optimization settings and configure them for your problem, click the **Optimization Options** button within the **Optimization** pane.

8 Click the **Start Optimization** button within the **Response Optimization** node. The optimization progress results appear within the **Optimization** pane. The **Compensators** pane contains the new, optimized compensator element values.

9 Optional: There are several options to export and save the results of the response optimization:

- To export the newly designed compensators to the MATLAB Workspace or a MAT-file, select **File > Workspace** within the SISO Design Tool graphical tuning window.

- To save the SISO Design task, including the **Response Optimization** task, select **File > Save** within the Control and Estimation Tools Manager.

- If you are designing compensators for a Simulink model, click the **Update Simulink Block Parameters** button within the **SISO Design Task** node to write the newly designed compensators to the Simulink model.

Simple Control Design Example

Response optimization of Simulink models using Simulink Response Optimization uses time-domain constraint bounds to represent lower and upper bounds on response signals. You can stretch, move, split, or open constraint bounds in a variety of ways that are explained here and in the online documentation. This section guides you through an example of how you might perform control design directly in a Simulink model using Simulink Response Optimization. In this section, you will constrain a parameter to control a second order SISO system via integral action, shown in the diagram below.
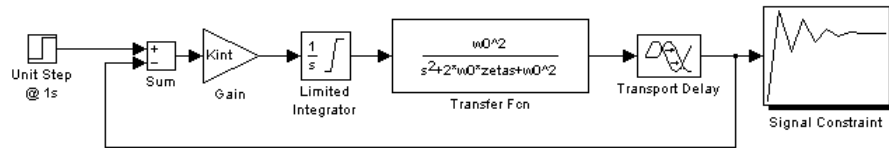
Specifically, the integral gain (`Kint`) should ensure that the closed loop system meets or exceeds the following performance specifications when you excite the system with a unit step input:

• A maximum 10% overshoot

• A maximum 10-second rise time

• A maximum 30-second settling time

Because of the actuator limits and system transport delay, standard linear control design techniques may not yield reliable results.

Simulink Response Optimization Startup

The Simulink model `srotut1` contains the system shown above. Open the system by typing `srotut1` at the MATLAB prompt.

You do not need to remodel any of your present Simulink systems to use Simulink Response Optimization. You need simply to

1 Attach a Signal Constraint block to all signals you want to constrain. In srotut1, a Signal Constraint block (square block with a step response icon) is attached to the plant output.

2 Add input signals to the system, for which you know what the output should look like. In srotut1, the input is a step since the desired step response characteristics of the system are known.

3 Change the model's **Stop time** to the desired value. In srotut1, the step response should settle within 30 seconds, so simulating for 50 seconds allows the step to go to completion. Since optimization with Simulink Response Optimization calls for many simulations of the system, you should make the simulation time as short as possible, but long enough to show dynamics of interest. You can change the **Start time** and **Stop time** through the Simulink Simulation Parameters dialog box by selecting **Simulation > Configuration Parameters**.

Adjusting Constraints

To open the Simulink Response Optimization Signal Constraint window, double-click the Signal Constraint block. The window, shown in the following figure, contains an amplitude versus time axis with default upper and lower constraint bounds. To optimize the tuned parameters so that the response signal lies within the constraint bound segments, select the **Enforce signal bounds** check box at the bottom of the window.
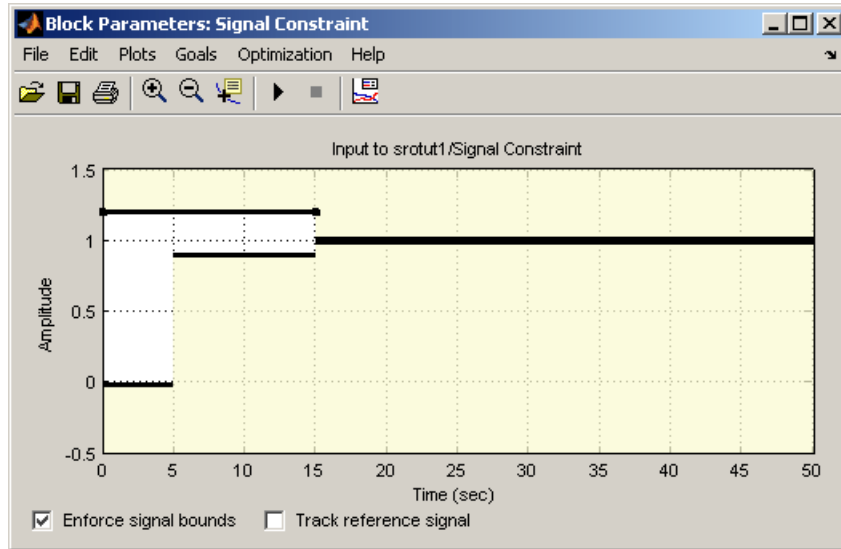
The lower and upper constraint bounds define a channel within which the signal response should lie. The default constraints effectively define a rise time of 5 seconds and a settling time of 15 seconds. These bounds must change to reflect the performance requirements proposed in the beginning of this section. To adjust the rise time constraint, position the mouse over the endpoint of the lower bound constraint that ends at 5 seconds. Press and hold down the (left) mouse button. The arrow should turn to a hand symbol as shown below.
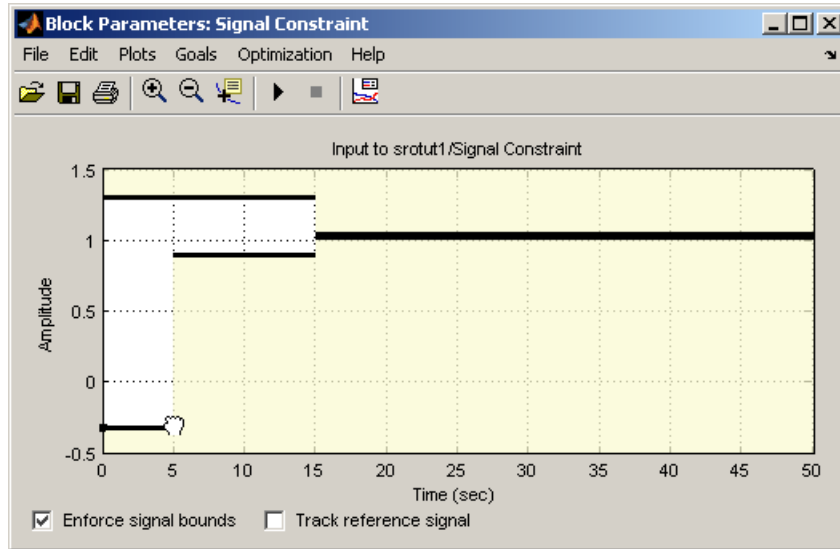
In this mode, you can change the time boundary between two constraints as well as change the slope of constraints. While still holding the mouse button down, drag the constraint boundary to the right. Release the mouse button after positioning the boundary as close as possible to 10 seconds. You may find it helpful to enable axis gridding while placing constraints. To turn axis gridding on or off, right-click anywhere within the white space of the Signal Constraint window and select **Grid** from the menu. If you need to precisely place constraint-bound segments, use the Edit Design Requirement dialog box, which appears when you right-click a constraint-bound segment and select **Edit** from the menu. See the online documentation for more information on the Edit Design Requirement dialog box. Alternatively, to ensure that the constraint remains perfectly horizontal (or vertical), you can press and hold down the **Shift** key while clicking and dragging the constraint segment. This causes the constraint segment to *snap* to the horizontal (or vertical).

To adjust the overshoot constraint, press and hold the (left) mouse button somewhere in the middle of the upper bound constraint bound segment that extends from 0 to 15 seconds. Notice that the pointer becomes a four-way arrow. In this mode, you can drag the entire constraint edge anywhere within the axes. While still holding the mouse button down, drag the constraint until its lower boundary is at a height of 1.1 as shown below.
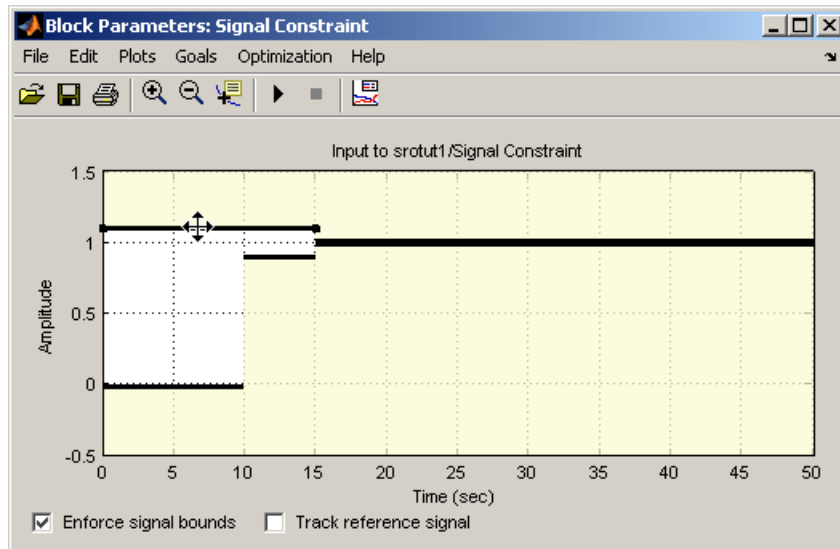
Finally, the settling time constraints require adjustment. Position the mouse button on the right edge of the lower constraint edge extending from 10 to 15 seconds. Press and hold down the (left) mouse button and notice that the constraint becomes selected and that the pointer changes to a hand symbol. In this mode, you can stretch the end of the constraint or change its angle. While still holding the mouse button down, drag the constraint so that the settling-time constraint begins at 30 seconds. Adjust the upper bound constraint so that it too defines a 30-second settling-time constraint. The constraint figure should now look the one shown below.
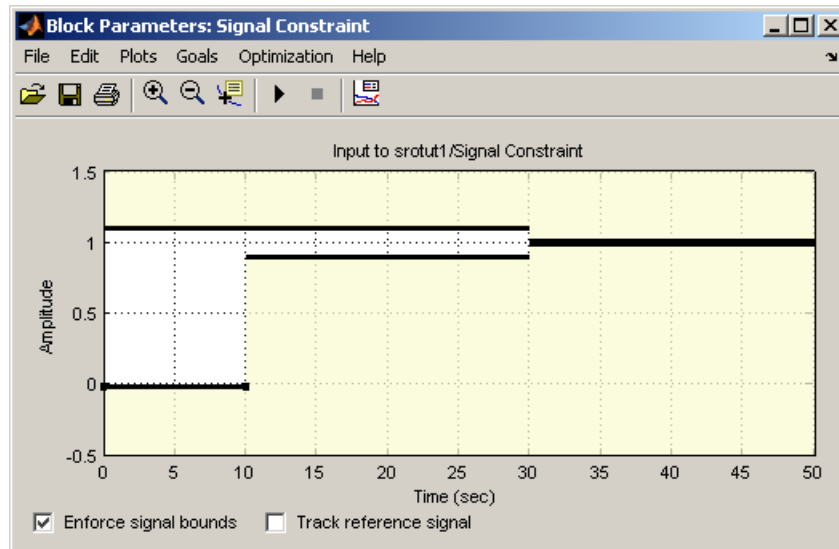
Alternatively, you could set the signal constraints using the Desired Response dialog box by selecting **Goals > Desired Response** from the Signal Constraint window. For more information on setting signal constraints, see the online documentation.

Specifying Tuned Parameters

Before beginning the optimization, you must tell Simulink Response Optimization which variables the optimization should tune. Open the Tuned Parameters dialog box by selecting **Optimization > Tuned Parameters** within the Signal Constraint window. Add the parameter Kint to the **Tuned parameters** list by clicking the **Add** button. This opens the Select Parameters dialog box with a list of all model parameters that are currently in the workspace.
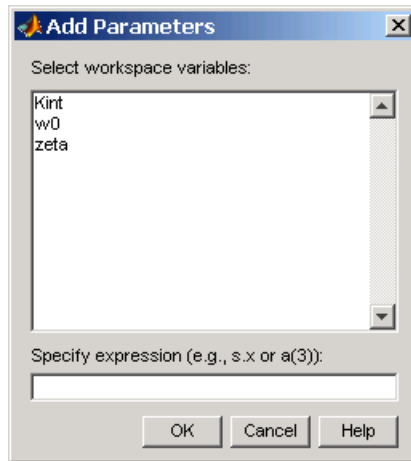
Select Kint in this list and click **OK**. This adds Kint to the **Tuned parameters** list as shown in the following figure.
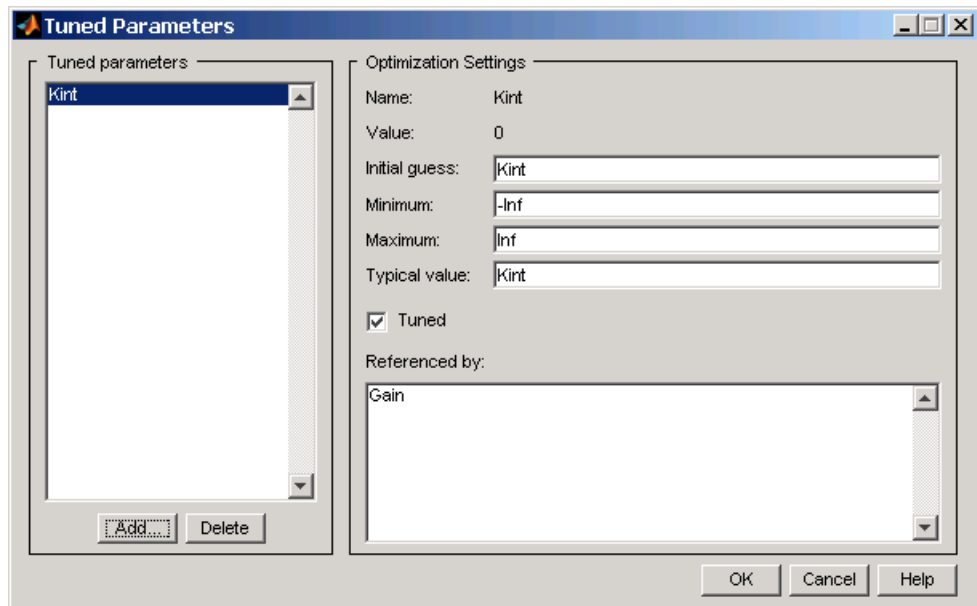
To display the optimization settings for this parameter, click Kint in the **Tuned parameters** list. The settings appear under **Optimization settings** on the right. To constrain Kint to be positive, enter 0 as the **Minimum** value. For more information on the various tuned parameter settings, see the online documentation.

Running the Optimization

After adjusting the constraint bounds in the Signal Constraint window and specifying the tuned parameters using the Tuned Parameters dialog box, you are ready to begin the optimization. You can start an optimization by clicking the **Start** button in the Signal Constraint window or by selecting **Optimization > Start**.

Simulink Response Optimization automatically converts the constraint bound data and tuned parameter information into a constrained optimization problem that it solves using functions from the Optimization Toolbox or the Genetic Algorithm and Direct Search Toolbox. It attempts to satisfy the constraints on the response signals by adjusting the tuned parameters. The results of each iteration appear in the Optimization Progress window shown in the following figure. The number of iterations necessary for the optimization to converge or terminate, will depend on the initial guess for the tuned parameters, the specific positioning of the constraints, and the optimization settings.

```
Optimization Progress                                                        _|□|x|

                       max             Directional  First-order
Iter S-count      f(x)  constraint   Step-size   derivative   optimality Procedure
   0    1          0      356.4
   1    6          0      44.33          1            0             1   infeasible
   2    9          0      0.2745         1            0             1   Hessian modified twice;   infeasible
   3   12          0      0.02024        1            0           993   Hessian modified twice
   4   15          0      0.0001516      1            0          14.6
Successful termination.
Found a feasible or optimal solution within the specified tolerances.

Kint =

   0.1852
```
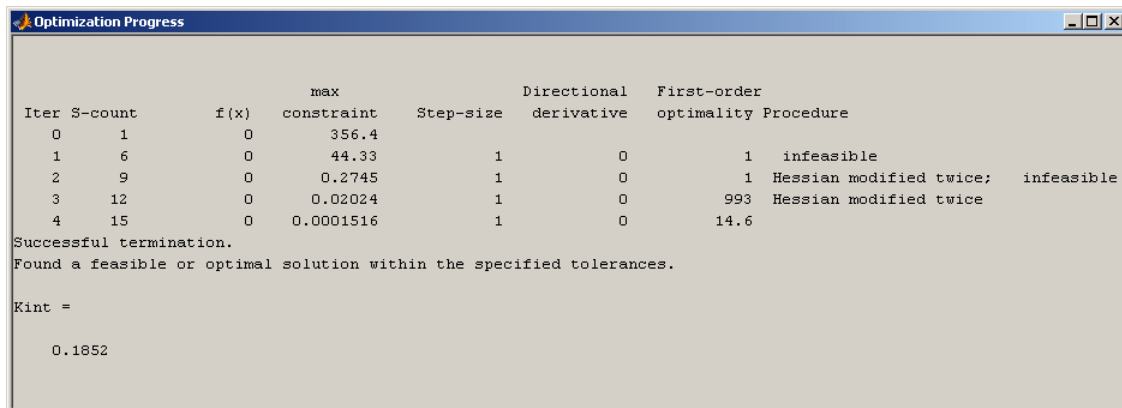
In this case the optimization converges after four iterations. For more information about the results shown in the Optimization Progress window, see the online documentation.

The Signal Constraint window, shown in the following figure, plots the responses at each iteration.

Start button
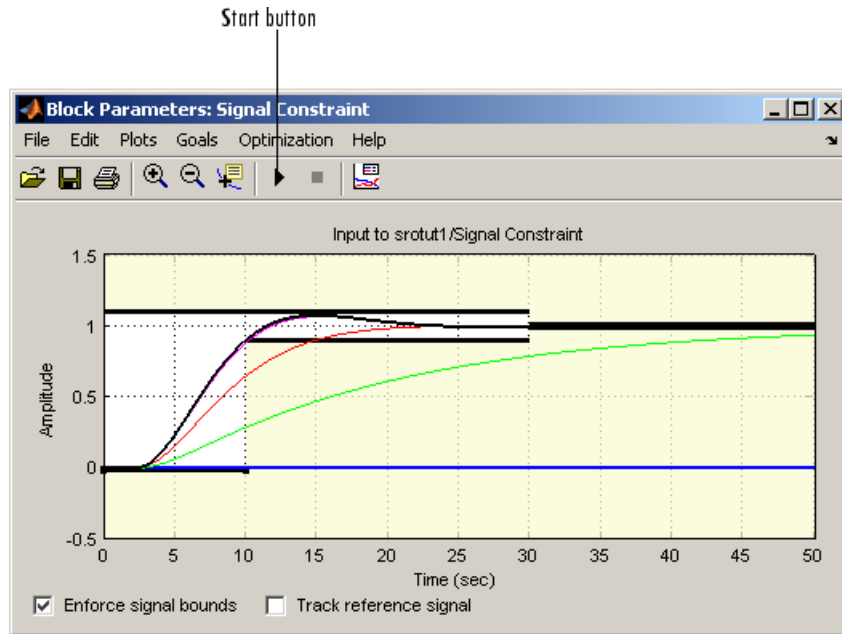
The blue line shows the initial response and the black line shows the current, or final, response. As you can see, the final response signal lies within the constraint bounds.

The new value of the tuned parameter Kint appears in the Optimization Progress window and is also changed in the MATLAB workspace. In this case the new value of Kint is 0.1852

Because of different numerical precision, the results of the optimization may differ slightly across different platforms.

Tracking a Signal

In addition to constraining the signal by constraint bounds, you can constrain the signal by requiring it to closely match a reference signal. To do this, select the **Track reference signal** option at the bottom of the Signal Constraint window. To enter the reference signal, select **Goals > Desired Response**. This displays the Desired Response dialog box, as shown in the following figure.
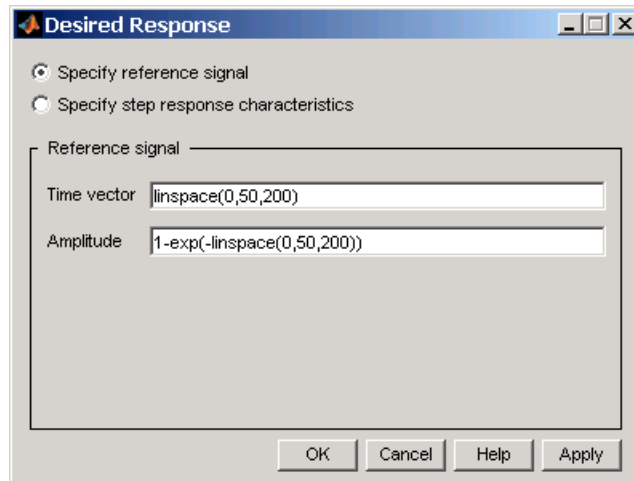
For this example, use the reference signal given by

$$y = 1 - e^{-t}$$

Enter time and amplitude vectors for the reference signal in the Desired Response dialog box as shown above, and then click **OK**. This displays the following plot in the Signal Constraint window. Note, to remove the plots of the optimized response signals, as was done in the figure, select **Plots > Clear Plots**.
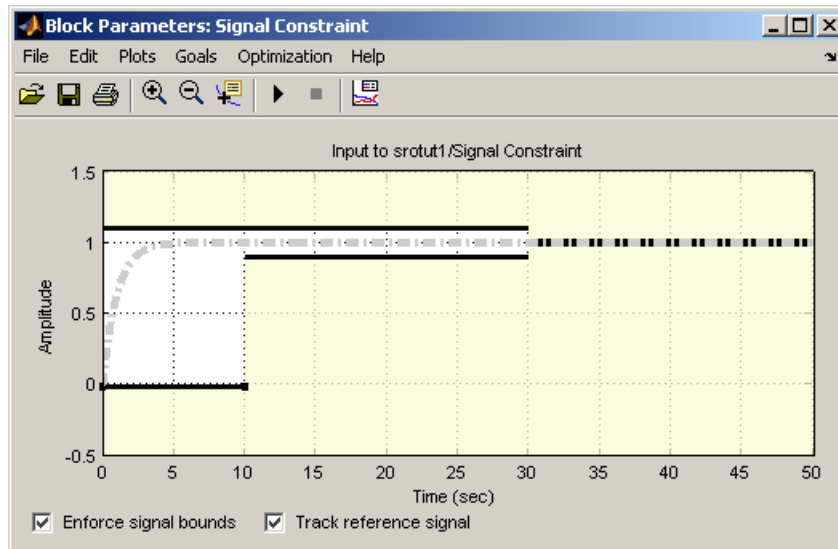
Before running the optimization, change the function tolerance to 0.01 within the Options dialog box. This ensures that the new objective function meets the imposed tolerances. To do this, open the Options dialog box by selecting **Optimization > Optimization Options** from the menu in the Signal Constraint window and then change the **Function tolerance** value to 0.01 within the **Optimization Options** pane of the Options dialog box, as shown in the following figure.
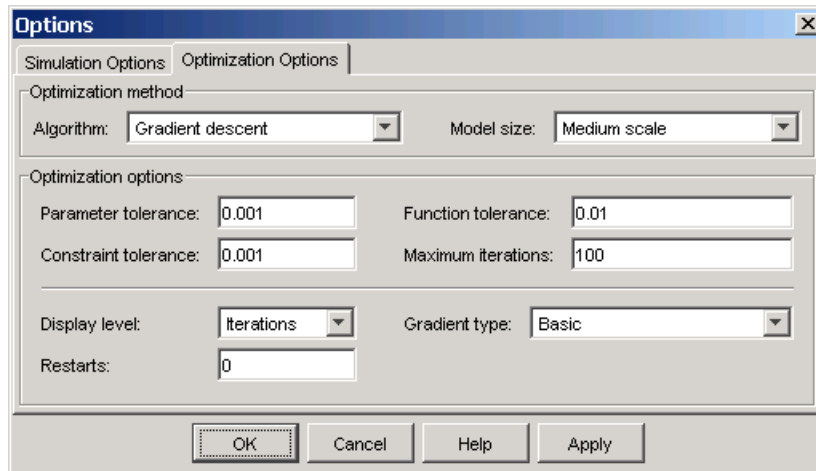
To run the optimization, select **Optimization > Start** or click the **Start** button on the toolbar in the Signal Constraint window. The results, shown below, are similar to the previous ones when only constraint bounds were used.
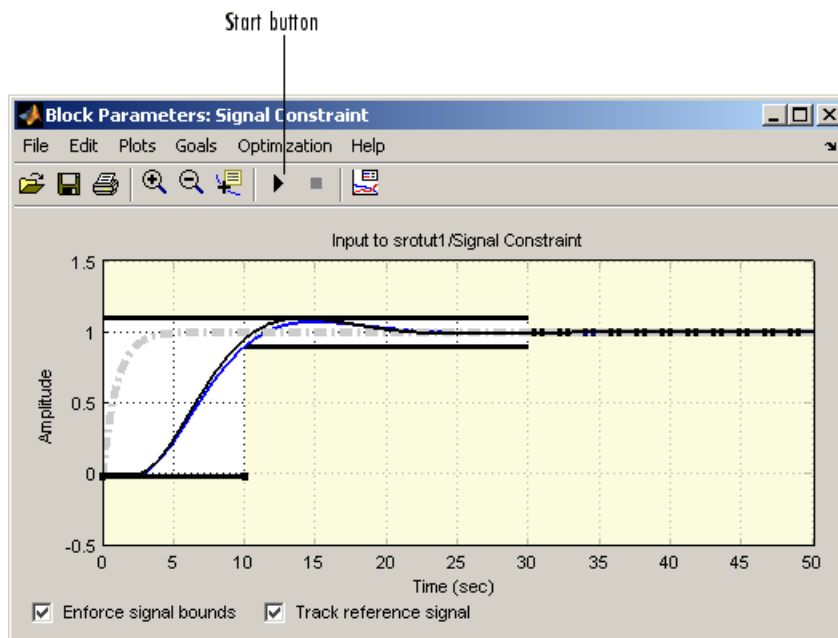
The Optimization Progress window shows the final value of Kint.

```
Optimization Progress                                                    _ □ X

                          max                    Directional    First-order
 Iter S-count       f(x)  constraint  Step-size   derivative    optimality Procedure
    0      1     8.26386   0.0001516
    1      8     7.95159   0.001293          1       -0.283         6.05
    2     11     7.96691          0          1        0.0156        0.0192
Successful termination.
Found a feasible or optimal solution within the specified tolerances.

Kint =

    0.1995
```
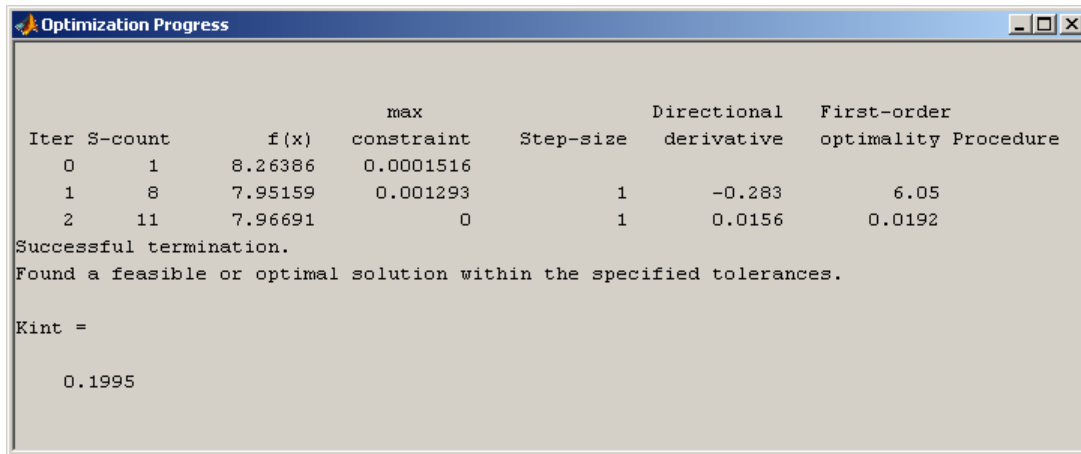
Before proceeding to the next section, make sure to clear the **Track reference signal** check box at the bottom of the Signal Constraint window.

Adding Uncertainty

In your particular problem, you might not have a precise plant model. Instead you might know what the nominal plant should be and have some idea of the uncertainty inherent in various components of the plant. For example, assume that the plant parameter zeta varies 5% about its nominal value and w0 varies between 0.7 and 1.45.

When using a Signal Constraint block to optimize response, Simulink Response Optimization allows you to optimize response signals in the face of this uncertainty by specifying uncertainty in parameter values. Note that you cannot add uncertainty to parameters or compensator elements when optimizing responses in a SISO Design Task.

Open the Uncertain Parameters dialog box by selecting **Optimization > Uncertain Parameters** in the Signal Constraint window. To add zeta and w0 as uncertain parameters, click the **Add** button, which opens the Select Parameters dialog box. This dialog box contains all parameters from the model that are currently in the workspace and are *not*

selected as tuned parameters. Select zeta and w0 in this list, and then click **OK**.
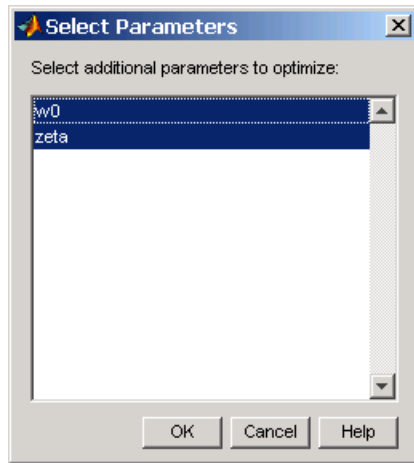
The parameters appear within the Uncertain Parameters dialog box with their default uncertainty settings. Their nominal values are the current parameter values, and the uncertainty range is 10% on either side of the nominal value.

Change the uncertainty ranges to those given previously: 0.95 to 1.05 for zeta and 0.7 to 1.45 for w0. The Uncertain Parameters dialog box should now look like that in the following figure.
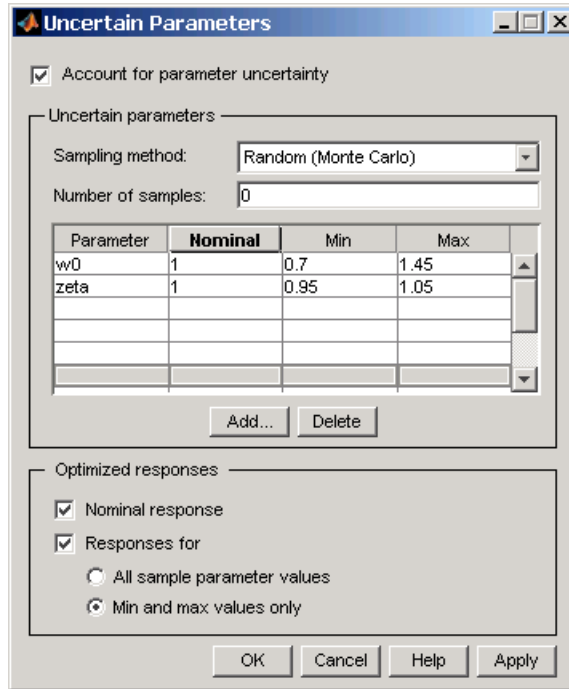
In this example, the sampling method for uncertainty is Random (Monte Carlo). The other choice is Grid in which case you would enter a vector of sample values for each parameter instead of minimum and maximum values. The number of samples indicates how many parameter value combinations to use other than the nominal, minimum, and maximum values.

By default, Simulink Response Optimization optimizes the nominal response along with all responses corresponding to combinations of minimum and maximum values of w0 and zeta (five responses total). To speed up to optimization you can optimize the nominal response only. To include more uncertainty, you can optimize all sample parameter values. Select which responses to optimize in the **Optimized responses** section of the Uncertain Parameters dialog box. If you choose not to optimize all uncertain responses, you can still use them for analysis by plotting them in the Signal Constraint window after the optimization. For more information on specifying uncertainty in your models, see the online documentation.

Click **OK** to save your uncertain parameter information, and then run the optimization again.

Simulink Response Optimization plots the uncertain responses as dashed lines. This time, the optimization does not find a feasible solution due to the uncertain response signals violating the constraints as shown in the following figure.
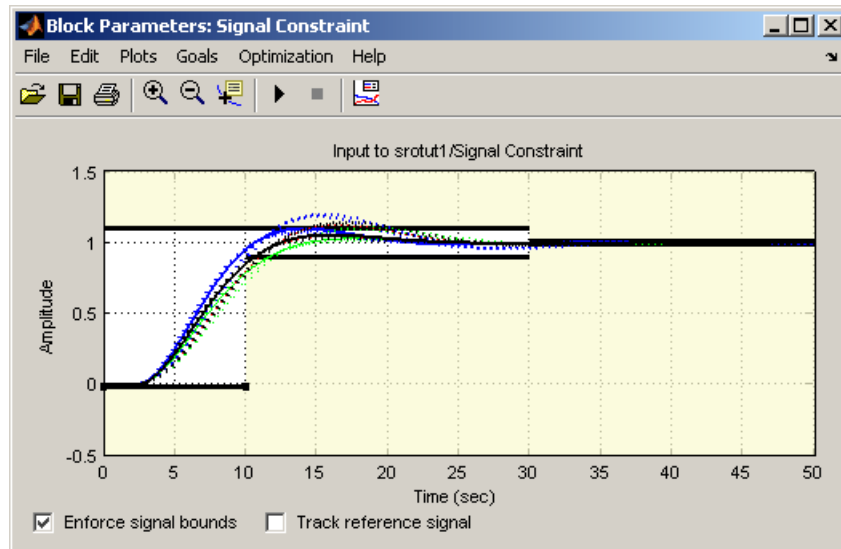
To get the optimization to converge, you might try a different initial guess, relax the constraints slightly, or change the uncertain parameter values. If the uncertain response signals do not violate the constraints by a large amount, the result might be acceptable for your design needs.

To turn off the uncertain response plots, right-click within the figure axes and select **Show > Uncertainty**. Turn them back on again by repeating this step. To clear all plots, right-click within the figure axes and select **Clear plots**.

Saving the Project

A response optimization project includes the constraints (from all Signal Constraint windows in the model), tuned parameter settings, uncertain

parameter settings, and settings for the optimization and simulation. After creating a project, you can save it for later use.

To save this project, select **File > Save** in the Signal Constraint window. This opens the Save Project dialog box as shown below.
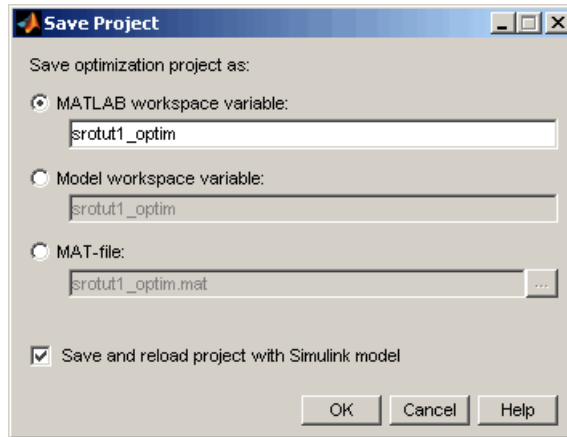
You can save the project to either a MATLAB workspace variable, model workspace variable, or a MAT-file. By selecting the **Save and reload project with Simulink model** check box, the project will automatically reload when you reopen the model. If the model cannot find the saved project when you reopen it, it gives a warning.

For more information on saving and reloading projects, see the online documentation.

Frequency Domain Response Optimization Example

When you have the Control System Toolbox, you can place Simulink Response Optimization design requirements or constraints on plots in the SISO Design Tool graphical tuning editor and analysis plots that are part of a SISO Design Task. This allows you to include design requirements for response optimization in the frequency-domain in addition to the time-domain.

You can specify frequency-domain design requirements to optimize response signals for any model that you can design within a SISO Design Task:

• Command-line LTI models created with the Control System Toolbox

• Simulink models that have been linearized using Simulink Control Design

This section guides you through an example using frequency-domain design requirements to optimize the response of a system in the SISO Design Task. The example uses a linearized version of the srotut1 model from "Simple Control Design Example" on page 2-6 and uses response optimization to design a compensator so that the closed loop system meets the following design specifications when you excite the system with a unit step input:

• A maximum 30-second settling time

• A maximum 10% overshoot

• A maximum 10-second rise time

• A limit of ±0.7 on the actuator signal

Creating an LTI Plant Model

In the srotut1 model, the plant model is composed of a gain, a limited integrator, a transfer function, and a transport delay.

We want to design the compensator for the open loop transfer function of the linearized srotut1 model. The linearized srotut1 plant model is composed of the gain, an unlimited integrator, the transfer function, and a Padé approximation to the transport delay.

To create an open loop transfer function based on the linearized srotut1 model, enter the following commands:

```
w0     = 1;
zeta   = 1;
Kint   = 0.5;
Tdelay = 1;
[delayNum,delayDen] = pade(Tdelay,1);
integrator  = tf(Kint,[1 0]);
transfer_fcn = tf(w0^2,[1 2*w0*zeta w0^2]);
delay_block  = tf(delayNum,delayDen);
open_loopTF  = integrator*transfer_fcn*delay_block;
```

Note It is also possible to directly linearize the Simulink model srotut1 using the product Simulink Control Design.

Creating Design and Analysis Plots

This example uses a root locus diagram to design the response of the open loop transfer function, open_loopTF. To create a SISO Design Task, that contains a root-locus plot, for the open loop transfer function, use the following command:

```
sisotool('rlocus',open_loopTF)
```

A **SISO Design Task** is created within the Control and Estimation Tools Manager, as shown below.
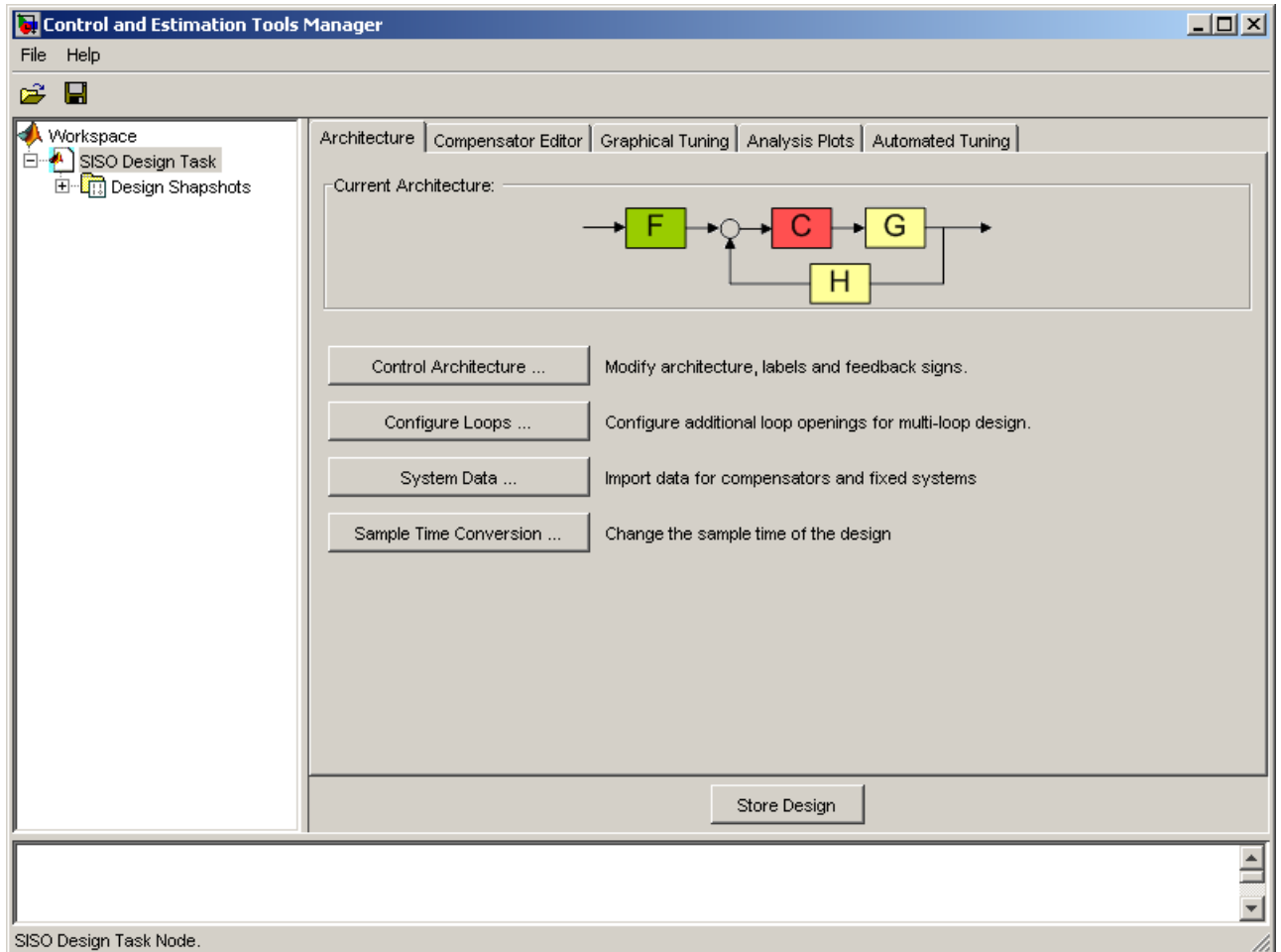
The SISO Design Task also contains a root locus diagram in the SISO Design Tool graphical tuning editor.
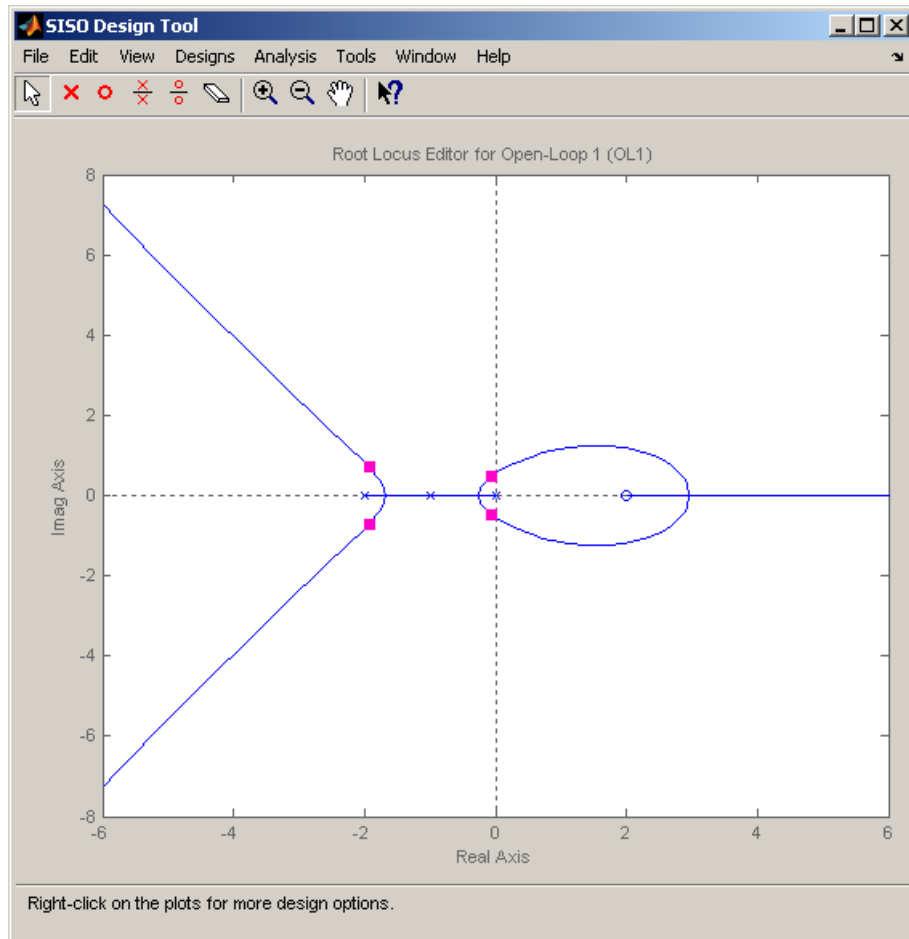
The Control and Estimation Tools Manager is a graphical environment for managing and performing tasks such as the design of SISO systems. The SISO Design Task node contains five panels that perform actions related to designing SISO control systems.

The **Architecture** pane, within the **SISO Design Task** node, lets you choose the architecture for the control system you are designing. This example uses the default architecture. In this system, the plant model, G, is the open loop transfer function open_loopTF, the prefilter, F, and the sensor, H, are set to

1, and the compensator, *C*, is the compensator that will be designed using response optimization methods.

In addition to the root-locus diagram, it is helpful to visualize the response of the system with a step response plot. To add a step response:

1 Select the **Analysis Plots** pane with the **SISO Design Task** node of the Control and Estimation Tool Manager.

2 Select Step for the **Plot Type** of **Plot 1**.

3 Under **Contents of Plots**, select the check box in column **1** for the response **Closed Loop r to y**.

A step response plot appears in an LTI Viewer. The plot shows the response of the closed loop system from r (input to the prefilter, *F*) to y (output of the plant model, *G*):
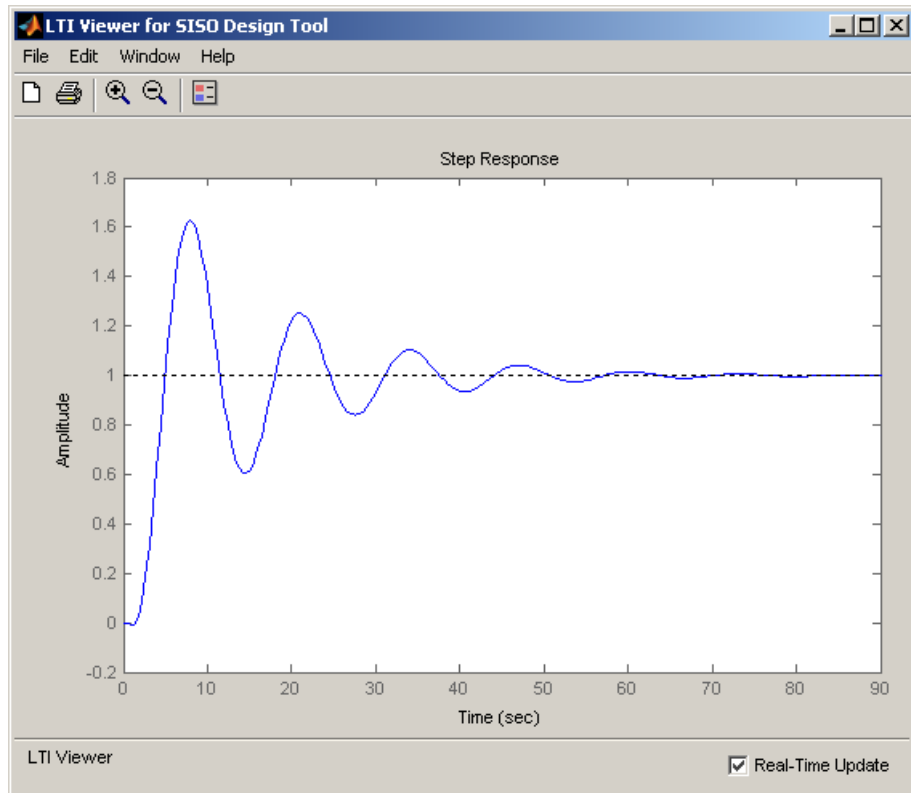
Creating a Response Optimization Task

There are several possible methods for designing a SISO system; this example uses an automated approach involving response optimization methods. After creating the design and analysis plots in "Creating Design and Analysis Plots" on page 2-24, you are ready to start a response optimization task to design the compensator.

To create a Simulink Response Optimization task:

1 Select the **Automated Tuning** pane within the **SISO Design Task** node in the Control and Estimation Tools Manager.

2 In the **Analysis Plots** pane, select `Optimization based tuning` as the **Design Method**.

3 Click the **Optimize Compensators** button to create the **Response Optimization** node under the **SISO Design Task** node in the tree browser in the left pane of the Control and Estimation Tools Manager.

The **Response Optimization** node contains four panes. With the exception of the first pane, each corresponds to a step in the response optimization process:

- **Overview**: A schematic diagram of the response optimization process.

- **Compensators**: Select and configure the compensator elements that you want to tune. See "Selecting Tunable Compensator Elements" on page 2-29.

- **Design requirements**: Select the design requirements that you want the system to meet after tuning the compensator elements. See "Adding Design Requirements" on page 2-30.

- **Optimization**: Configure optimization options and view the progress of the response optimization. See "Optimizing the System's Response" on page 2-39.

Note When optimizing responses in a SISO Design Task, you cannot add uncertainty to parameters or compensator elements.

Selecting Tunable Compensator Elements

Simulink Response Optimization tunes elements or parameters within compensators in your system so that the response of the system meets the design requirements you specify. To specify the compensator elements to tune:

1 Select the **Compensators** pane within the **Response Optimization** node.

2 Within the **Compensators** pane, select the check boxes in the **Optimize** column that correspond to the compensator elements you want to tune.

In this example, to tune the **Gain** in the compensator **C**, select the check box next to this element, as shown below.
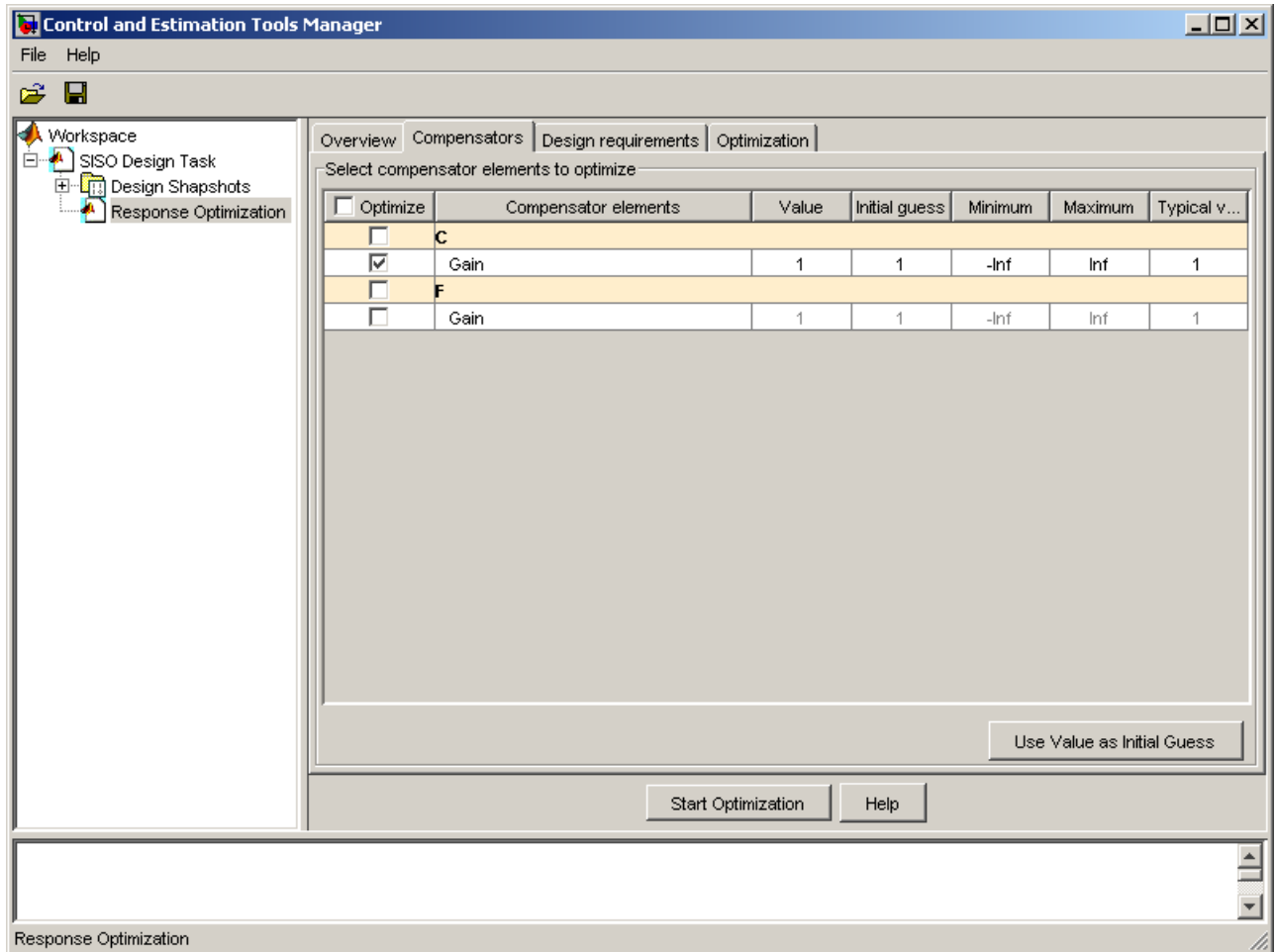
Note Compensator elements or parameters cannot have uncertainty when used with frequency-domain based response optimization.

Adding Design Requirements

Simulink Response Optimization can use both frequency-domain and time-domain design requirements to tune parameters in a control system.

The **Design requirements** pane within the **Response Optimization** node of the Control and Estimation Tools Manager provides an interface to create new design requirements and select those you want to use for a response optimization.

This example uses the design specifications from "Frequency Domain Response Optimization Example" on page 2-23. The following sections each create a new design requirement to meet these specifications.

Settling Time Design Requirement

The first design specification for this example is to have a settling time of 30 seconds or less. This specification can be represented on a root-locus diagram as a constraint on the real parts of the poles of the open loop system.

To add this design requirement:

1 Select the **Design requirements** pane within the **Response Optimization** node of the Control and Estimation Tools Manager.

2 Click the **Add new design requirement** button. This opens the New Design Requirement dialog box.

Within this dialog box you can specify new design requirements and add them to a new or existing design or analysis plot.

3 Add a design requirement to the existing root-locus diagram:

a Select Pole/zero settling time from the **Design requirement type** menu.

b Select Open-Loop L from the **Requirement for response** menu.

c Enter 30 seconds for the **Settling time**.

d Click **OK**.

A vertical line should appear on the root-locus diagram, as shown below.
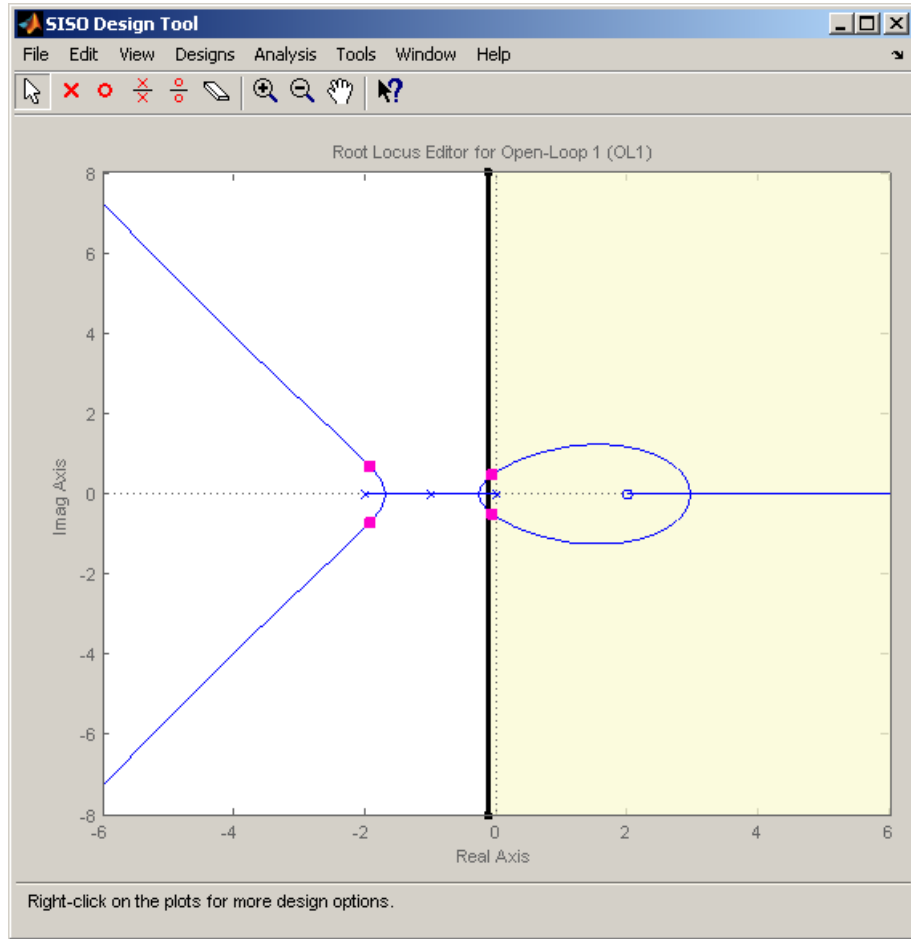
Overshoot Design Requirement

The second design specification for this example is to have a percentage overshoot of 10% or less. This specification is related to the damping ratio on a root-locus diagram. In addition to adding a design requirement with the **Add new design requirement** button, you can also right-click directly on the design or analysis plots to add the requirement, as shown below.

To add this design requirement:

1 Right-click anywhere within the white space of the root-locus diagram in the SISO Design Tool window. Select **Design Requirements > New** to open the New Design Requirement dialog box.

2 Select Percent overshoot as the **Design requirement type** and enter 10 as the **Percent overshoot**.

3 Click **OK** to add the design requirement to the root-locus diagram. The design requirement appears as two lines radiating at an angle from the origin, as shown below.
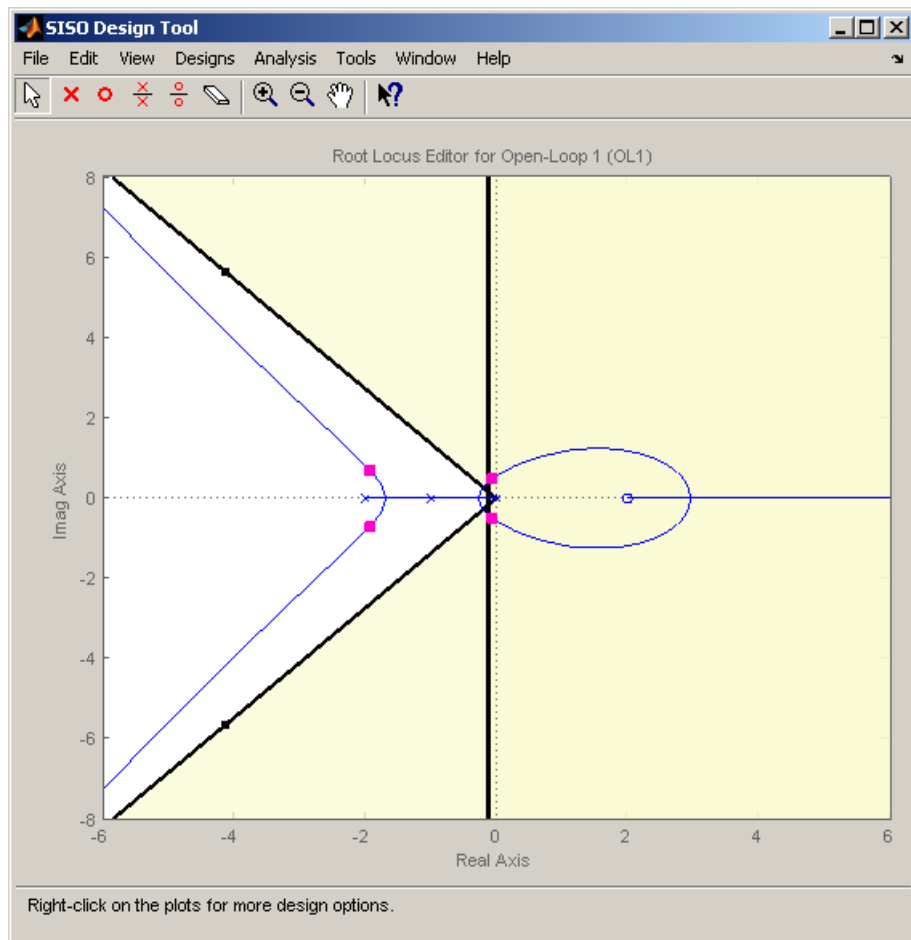
Rise Time Design Requirement

The third design specification for this example is to have a rise time of 10 seconds or less. This specification is related to a lower limit on a Bode Magnitude diagram.

To add this design requirement:

1 Select the **Graphical Tuning** pane within the **SISO Design Task** node of the Control and Estimation Tools Manager.

2 For Plot 2, set **Plot Type** to Open-Loot Bode.

3 Right-click anywhere within the white space of the root-locus diagram in the SISO Design Tool window. Select **Design Requirements > New** to open the New Design Requirement dialog box.

4 Create a design requirement representing the rise time and add it to a new Bode plot:

   a Select Lower gain limit from the **Design requirement type** menu.

   b Enter 1e-2 to 0.17 for the **Frequency** range.

   c Enter 0 to 0 for the **Magnitude** range.

   d Click **OK**.

A Bode diagram appears within the SISO Design Tool window. The magnitude plot of the Bode diagram includes a horizontal line representing the design requirement, as shown below.
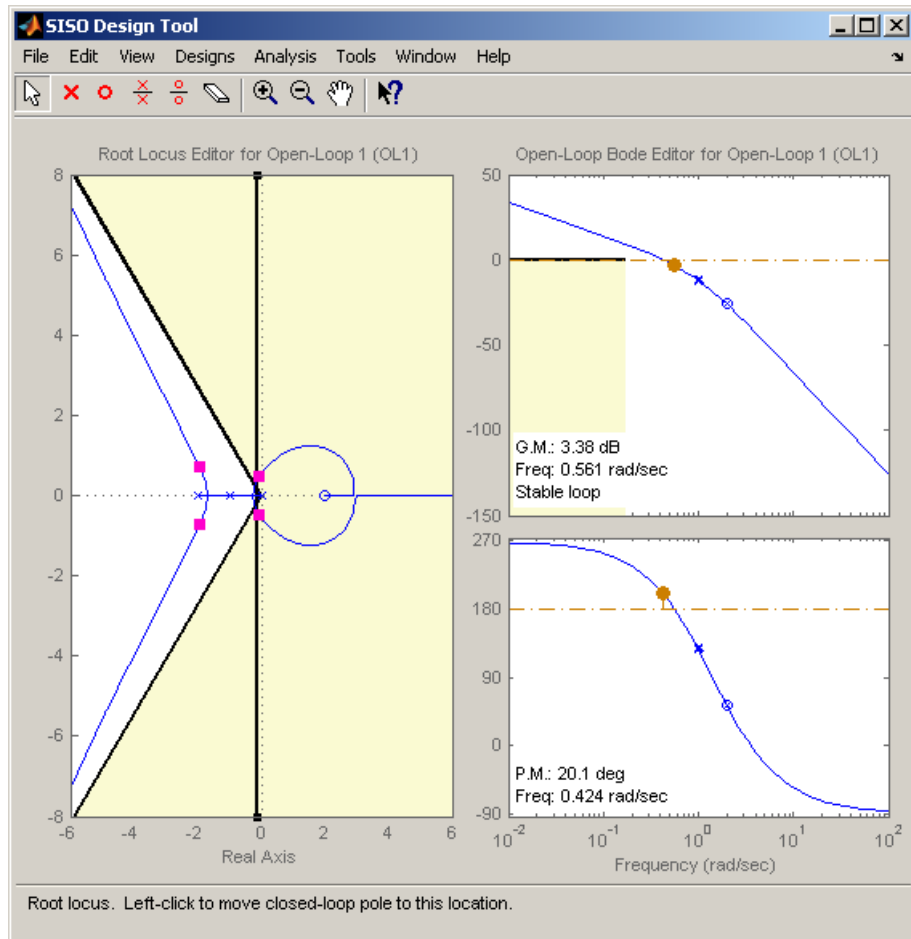
Actuator Limit Design Requirement

The fourth design specification for this example is to limit the actuator signal to within ±0.7. To add this design requirement:

1 Select the **Design requirements** pane within the **Response Optimization** node of the Control and Estimation Tools Manager.

2 Click the **Add new design requirement** button to open the New Design Requirement dialog box.

3 Create a time-domain design requirement representing the upper limit on the actuator signal, and add it to a new step response plot in the LTI Viewer:

a Select `Step response upper amplitude limit` from the **Design requirement type** menu.

b Select `Closed Loop r to u` from the **Requirement for response** menu.

c Enter `0` to `10` for the **Time** range.

d Enter `0.7` to `0.7` for the **Amplitude** range.

e Click **OK**. A second step response plot for the closed loop response from `r` to `u` appears in the LTI Viewer. The plot contains a horizontal line representing the upper limit on the actuator signal.

f To extend this limit for all times (to $t=\infty$), right click on the black edge of the design requirement, somewhere toward the right edge, and select **Extend to inf**. The diagram should now appear as shown below.
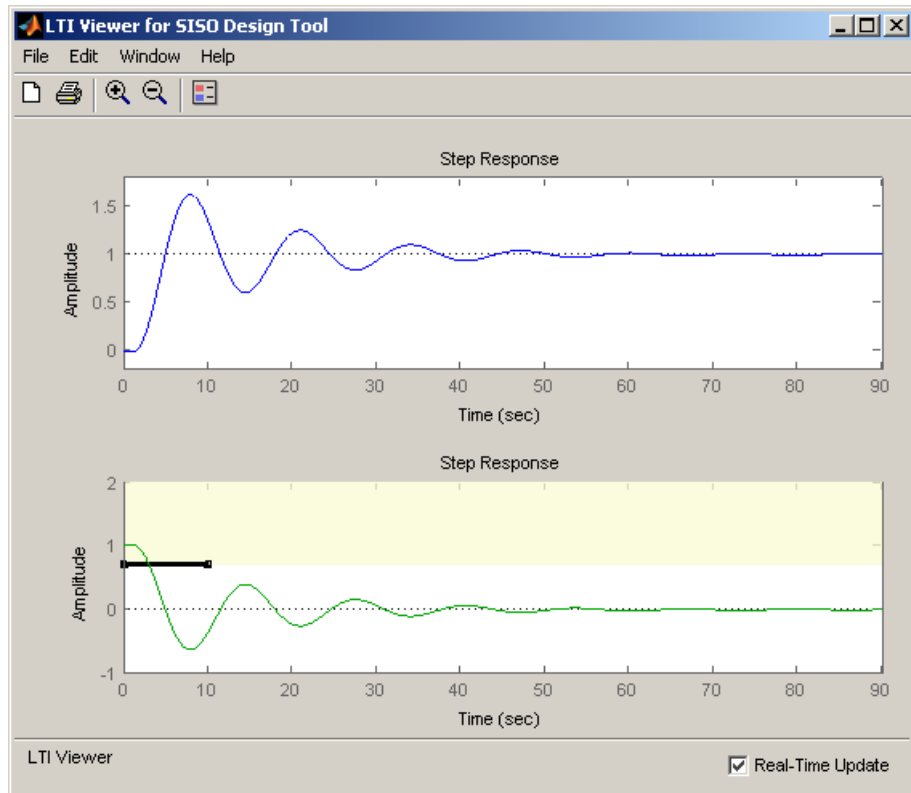
To add the corresponding design requirement for the lower limit on the actuator signal:

1 Within the white space of the lower step response plot, right-click and select **Design Requirements > New** to open the New Design Requirement dialog box.

2 Create a time-domain design requirement representing the lower limit on the actuator signal, and add it to the step response plot in the LTI Viewer:

 a Select Step response lower amplitude limit from the **Design requirement type** menu.

 b Select Closed Loop r to u from the **Requirement for response** menu.

    c Enter 0 to 10 for the **Time** range.

    d Enter -0.7 to -0.7 for the **Amplitude** range.

    e Click **OK**. The step response plot now contains a second horizontal line representing the lower limit on the actuator signal.

    f To extend this limit for all times (to $t=\infty$), right-click in the yellow shaded area and select **Extend to inf**. The diagram should now appear as shown below.
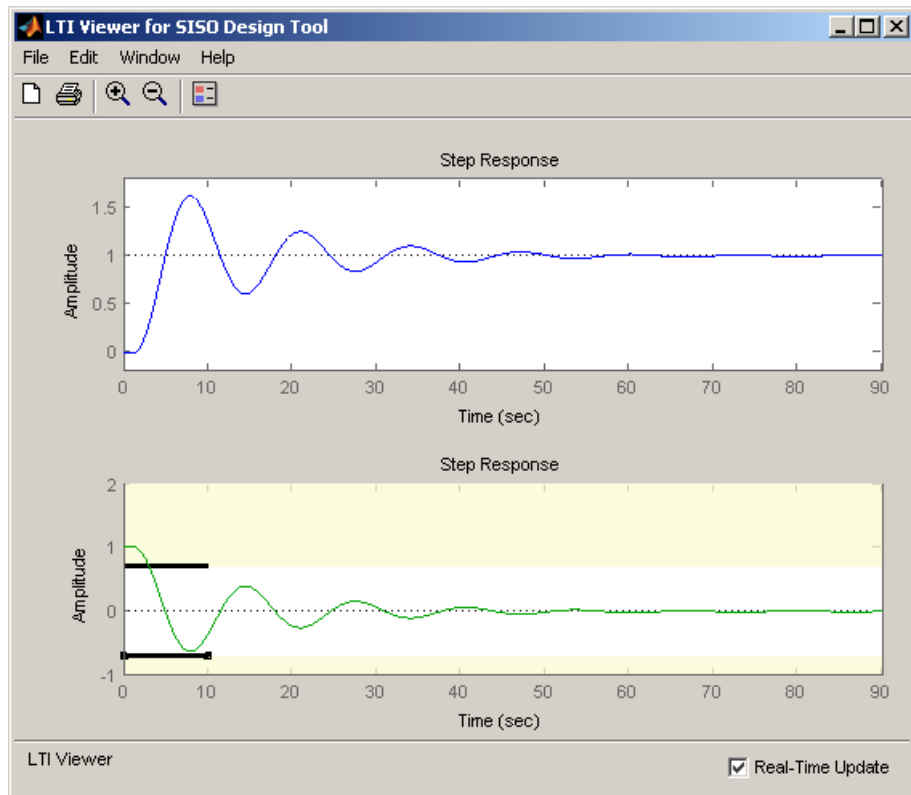
Selecting the Design Requirements to Use During Response Optimization

The design requirements give constraints on the dynamics of the system and the values of response signals. The table lists all design requirements

in the design and analysis plots. Select the check boxes next to the design requirements you want to use in the response optimization. This example uses all the current design requirements.

Optimizing the System's Response

After selecting the compensator elements to tune and adding design requirements for the response signals to satisfy, you are ready to being the response optimization.

The **Optimization** pane within the **Response Optimization** node of the Control and Estimation Tools Manager displays the progress of the response optimization. The pane also contains options to configure the types of progress information displayed during the optimization and options to configure the optimization methods and algorithms.

To optimize the response of the system in this example, click the **Start Optimization** button.

The **Optimization** pane displays the progress of the optimization, iteration by iteration, as shown below. Termination messages from the optimization algorithm and suggestions for improving convergence also appear here.
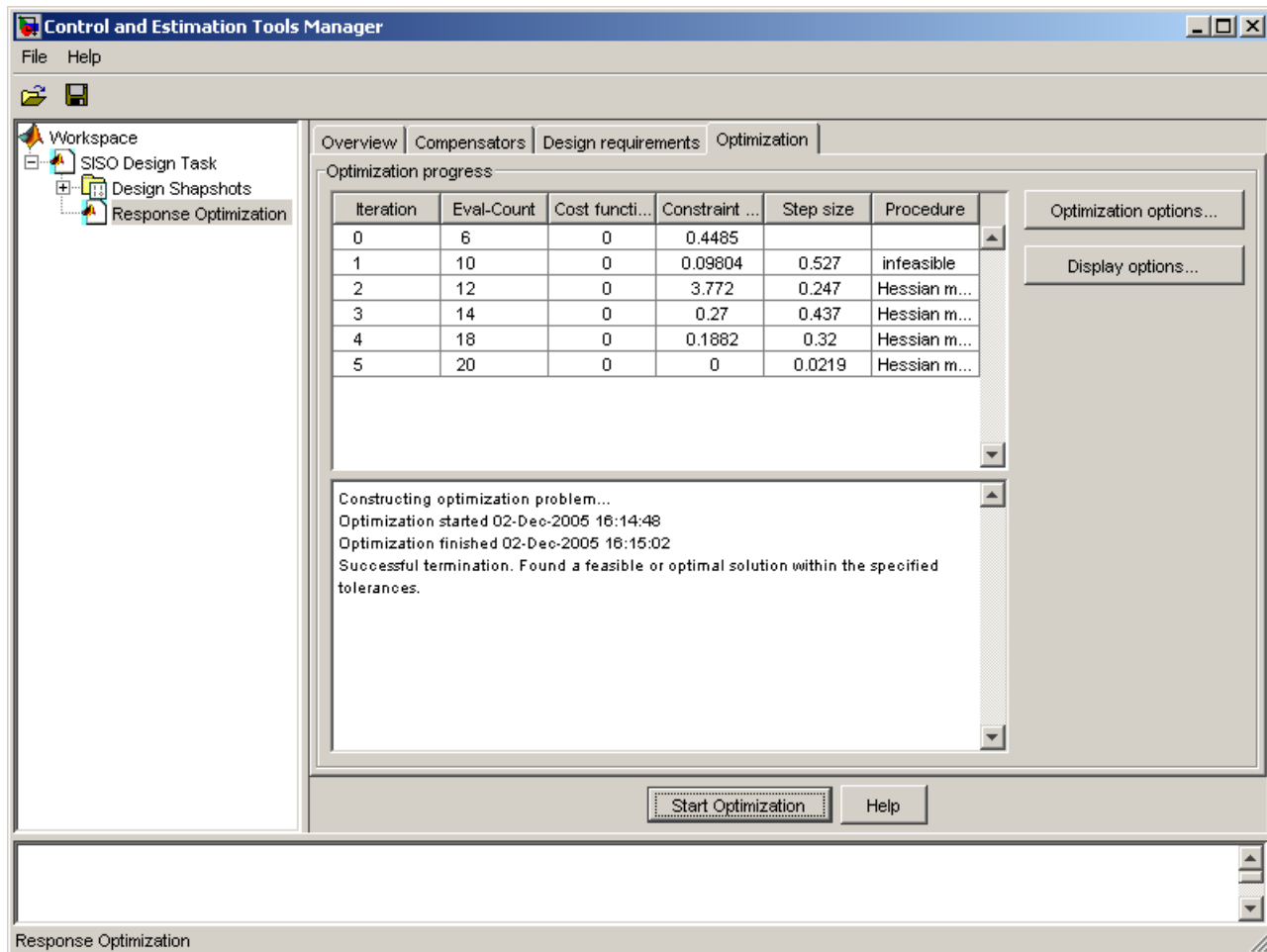
The optimized signals in the design and analysis plots appear as follows:
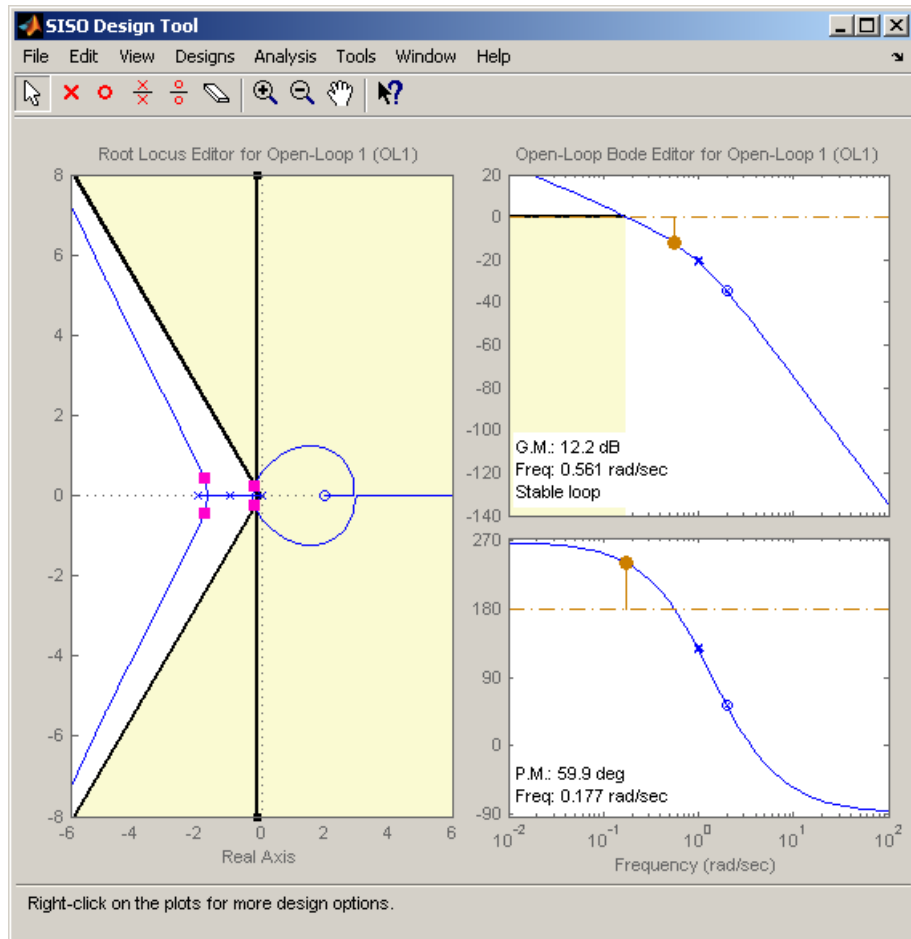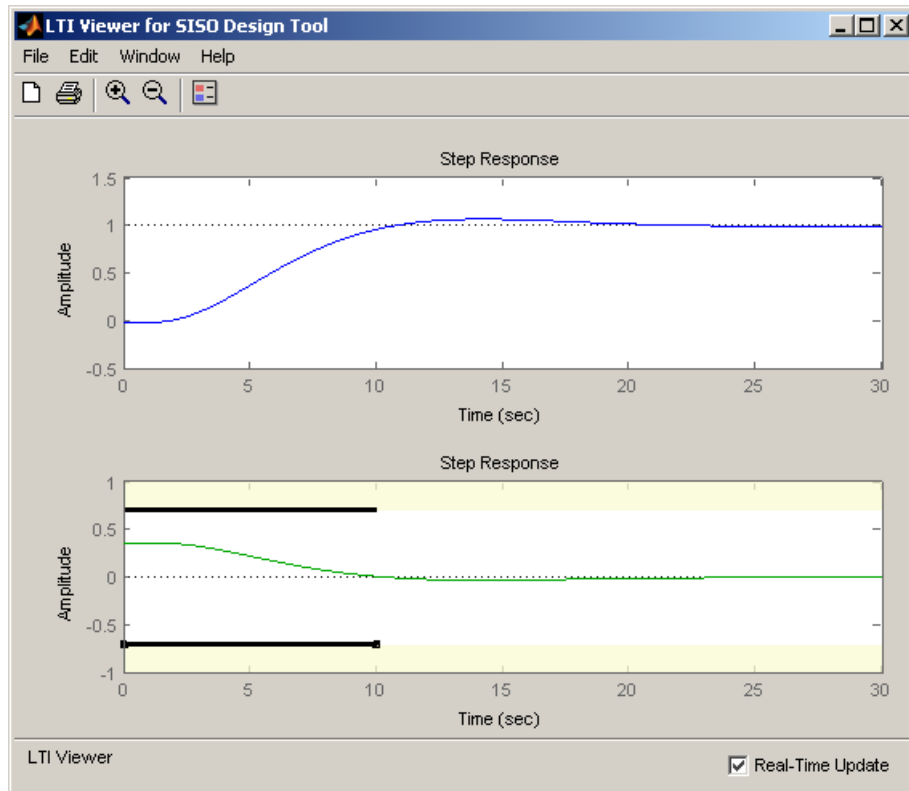
Creating and Displaying the Closed-Loop System

After designing a compensator by optimizing the response of the system, you can create export the compensator to the MATLAB Workspace and create a model of the full closed-loop system.

1 Within the SISO Design Tool window, select **File > Export** to open the SISO Tool Export dialog box.

2 Select the compensator you designed, **Compensator C**, and then click the **Export to Workspace** button.

At the command-line, enter the following command to create the closed-loop system, CL, from the open-loop transfer function, open_loopTF, and the compensator, C:

```
CL=feedback(open_loopTF,C);
```

This returns the following model:

```
Zero/pole/gain:
                    -0.5 (s-2)
-----------------------------------------------
(s^2  + 0.4244s + 0.1079) (s^2  + 3.576s + 3.375)
```

To create a step response plot of the closed loop system, enter the following command:

```
step(CL);
```

This produces the following figure:
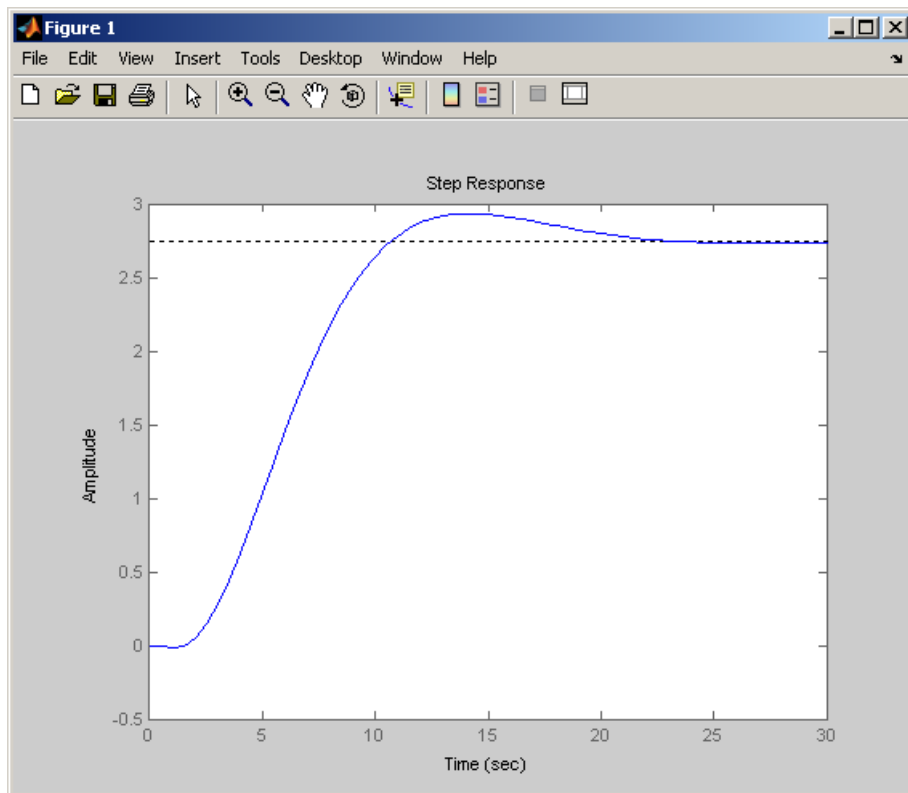
Physical Modeling Example

The following example demonstrates further techniques for using Simulink Response Optimization to directly tune Simulink models. It uses the model srotut2 to show how to constrain more than one signal and tune multiple parameters.

Specifically, you will design a hydraulic system that extends a piston by 0.07 m while keeping the pressure from the pump below 12×10^5 Pa. The physical system consists of a hydraulic pump supplying pressure to a hydraulic cylinder. The cylinder contains a piston, which extends and compresses a spring. This system is modeled in the Simulink model srotut2. The system parameters that you can adjust, or tune, to achieve the required design characteristics are the cross-sectional area of the piston, Ac, and the maximum pump flow-rate, Qmax.

The specifications for the response of the piston position signal are

- Maximum overshoot of 15%

- Rise time of 0.25 second

- Settle to within 10% of final value after 0.5 second

- Final value of 0.07 m

The specifications for the response of the pump pressure signal are

- Lower bound of zero (Pressures below this value are not physically possible.)

- Upper bound of 12×10^5 Pa

Opening the Hydraulic Cylinder Model

The Simulink system srotut2 contains a block diagram representation of the hydraulic system described above. You can open the system by typing srotut2 at the MATLAB prompt.
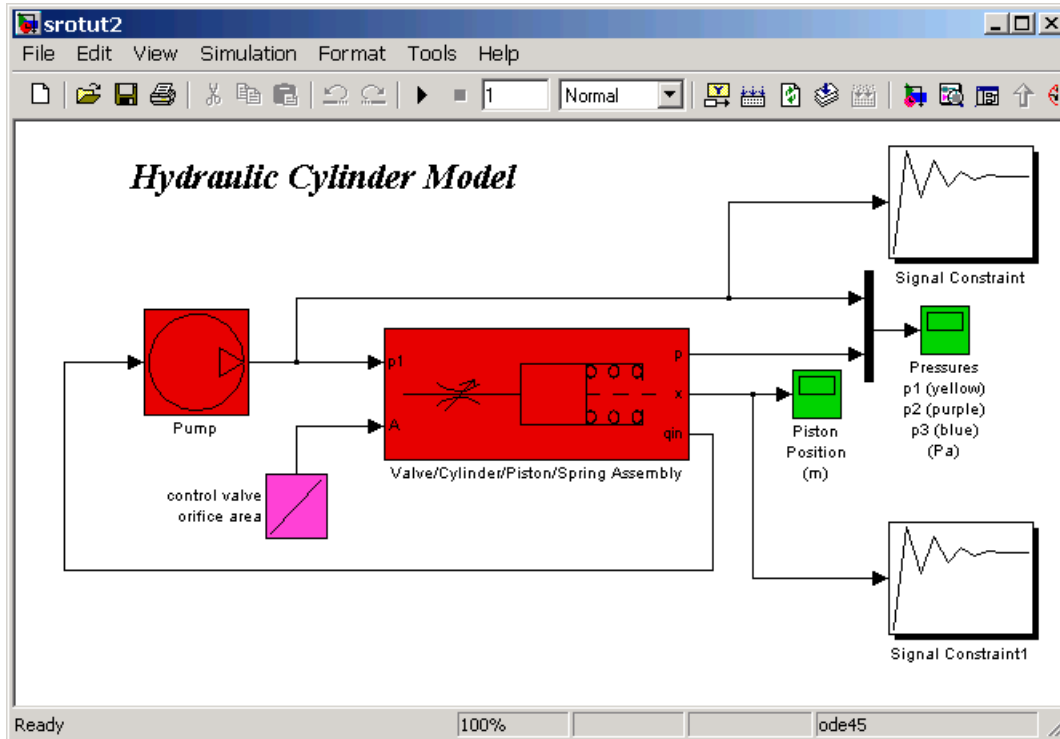
Notice that the system contains two Signal Constraint blocks. These blocks define constraints on the piston position and pump pressure signals. Simulink Response Optimization allows you to constrain multiple responses including responses within subsystems of your model. For each signal that you want to constrain, you need to attach a Signal Constraint block. Within each Signal Constraint block, set constraint bounds and/or reference signals for each signal. The tuned and uncertain parameters, along with optimization and simulation options are set for the *whole system* within any one of the Signal Constraint blocks.

Adjusting Constraints

In this section you will adjust the signal constraints in the system.

Adjusting Pump Pressure Signal Constraints

Double-click the block labeled Signal Constraint, in the upper-right corner of the model diagram. This block defines the constraints on the pump pressure signal.

First, delete some constraint bound segments so that there is just a single upper bound and a single lower bound. To do this, right-click the left-most lower segment and select **Delete** from the menu.
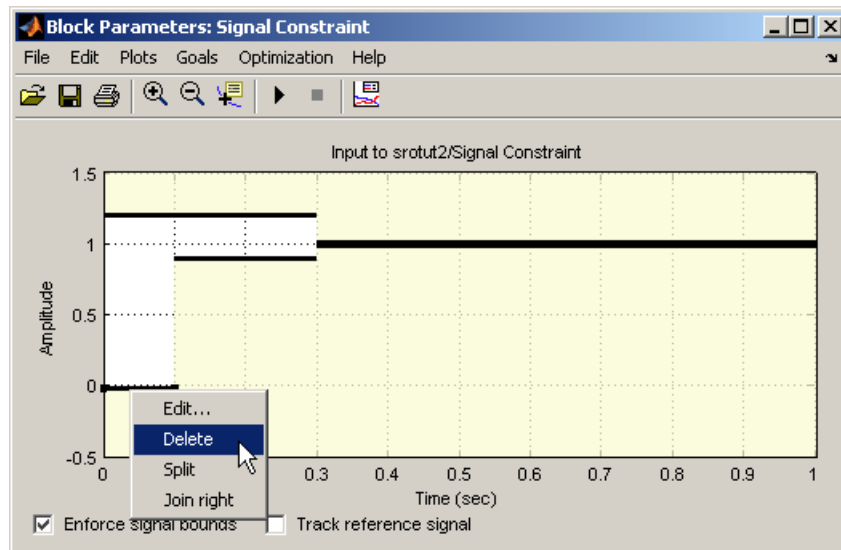
Repeat for one additional lower segment and one upper segment. The figure window should now look something like that in the following figure. Note that the thick black constraint line is actually two constraint edges positioned very close to each other.
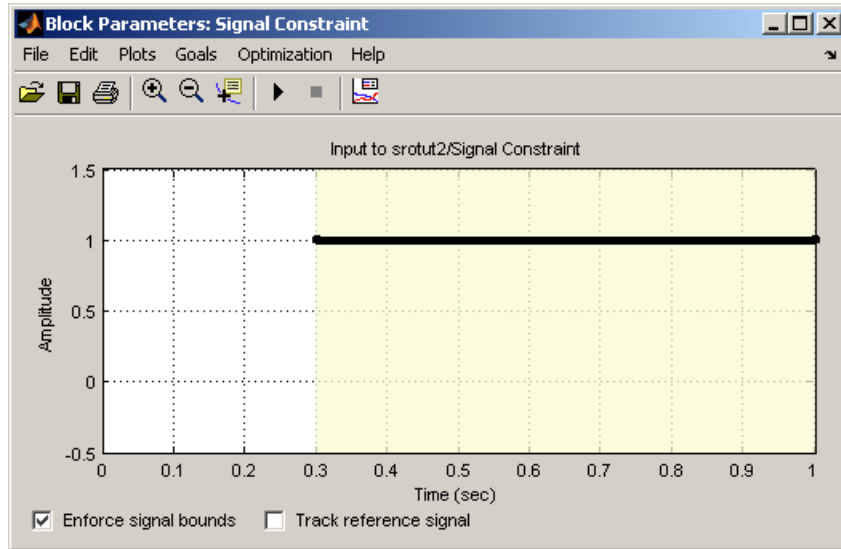
Reposition the constraint bound segments using the Edit Design Requirement dialog box. To move the lower constraint, right-click anywhere on the lower constraint segment and select **Edit** from the menu. Within the Edit Design Requirement dialog box, enter the start and end time of the constraint edge (0 and 1 respectively) as well as the amplitude of the constraint edge (since it should lie flat, both ends have the same magnitude of 0).

The following figure shows the Edit Design Requirement dialog box with the position settings for the lower constraint segment.
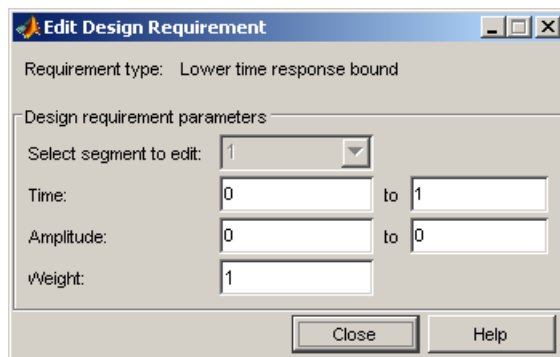
Repeat for the upper constraint, moving the constraint edge so that it begins at 0, ends at 1, and has a magnitude of 12e5. See the online documentation for more information on the Edit Design Requirement dialog box.

The new constraints are positioned at the edge of the axes and cannot be seen. To adjust the axes so that the constraints lie within the axes limits, select **Edit > Axes Properties** in the Signal Constraint window. Change the **Y-limits** to -5e5 and 15e5. The Signal Constraint window should now look like that in the following figure.
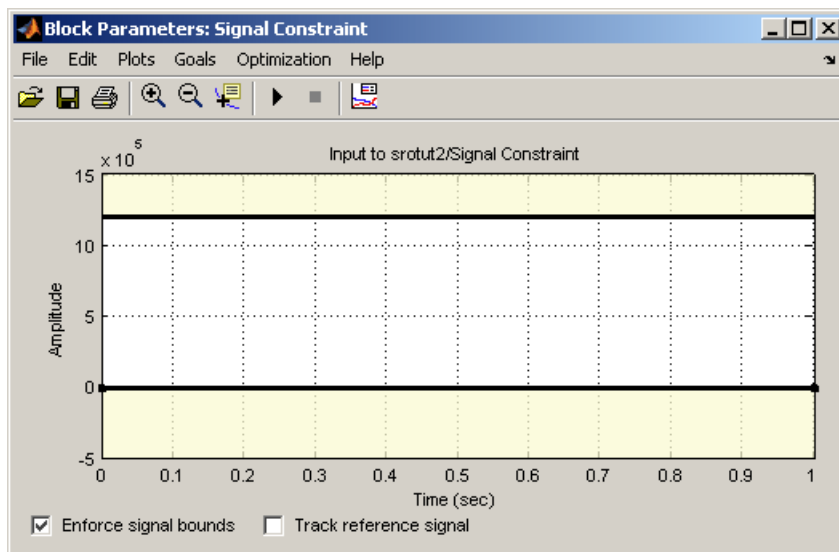

Adjusting Piston Position Signal Constraints

Double-click the block labeled Signal Constraint1, in the lower-right corner of the model diagram. This block defines the constraints on the piston position signal.

Reposition the constraint bound segments using the Desired Response dialog box. Open the dialog box by selecting **Goals > Desired Response** from the Signal Constraint window. Select **Specify step response characteristics**. The list at the beginning of "Physical Modeling Example" on page 2-45 specifies the step response characteristics. Enter the information shown in the following figure, and then click **OK**.
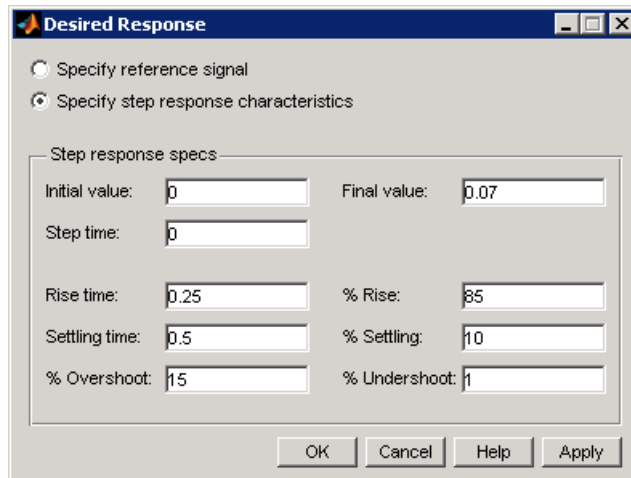
These step response characteristics require that

- After 0.5 second, the signal must settle to within 10% of its final value (between 0.063 m and 0.077 m).

- After 0.25 second, the signal must rise to 85% of the final value (or approximately 0.06 m).

- The signal has a maximum overshoot of 15% (or up to approximately 0.08 m).

- The signal must have very little undershoot.

The constraint-bound segments should now look like those in the following figure.
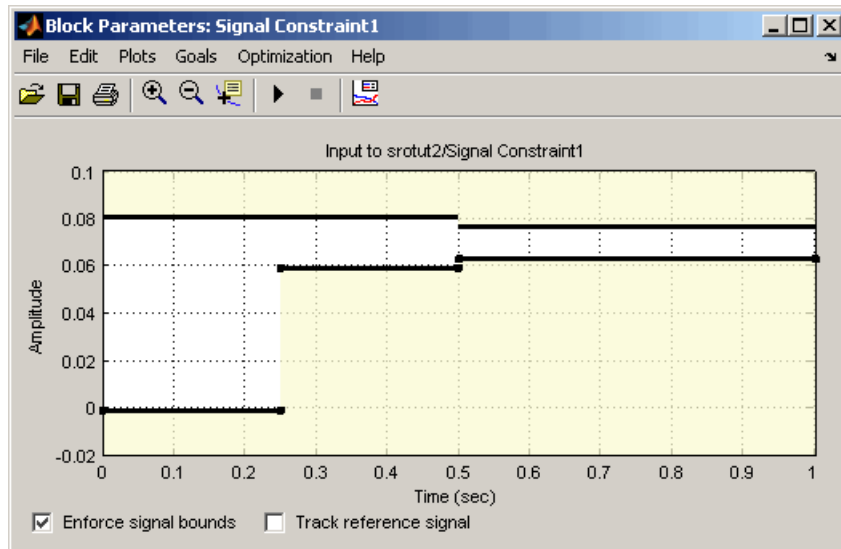
See the online documentation for more information on step response settings.

Specifying Tuned Parameters

To optimize the signal responses in this example, you will tune two parameters: the cross-sectional area of the piston, Ac, and the maximum pump flow rate, Qmax. Although there are two constrained signals and two tuned parameters, you could have any number of tuned parameters in your response optimization.

You can specify tuned parameters within any Signal Constraint block in the model; the tuned parameter settings are for the *whole system*, not just for a particular optimized signal.

To specify the tuned parameters for this example, open any Signal Constraint block in the model and select **Optimization > Tuned Parameters**. Within the Tuned Parameters dialog box, click **Add**, and then select Ac and Qmax from the list in the Add Parameters dialog box. Hold down the **Ctrl** key to select multiple parameters from the list. Click **OK** to continue.

In the Tuned Parameters dialog box, enter specifications for each tuned parameter by selecting the parameter in the list on the left, and then entering information such as initial guesses, minimum values, and maximum values on the right. For Ac, enter 0 as a **Minimum** value since it does not make sense for the cross-sectional area to be negative. Similarly, enter 0 as a **Minimum** value for Qmax to avoid negative flow rates, which imply that the flow reverses direction through the system.
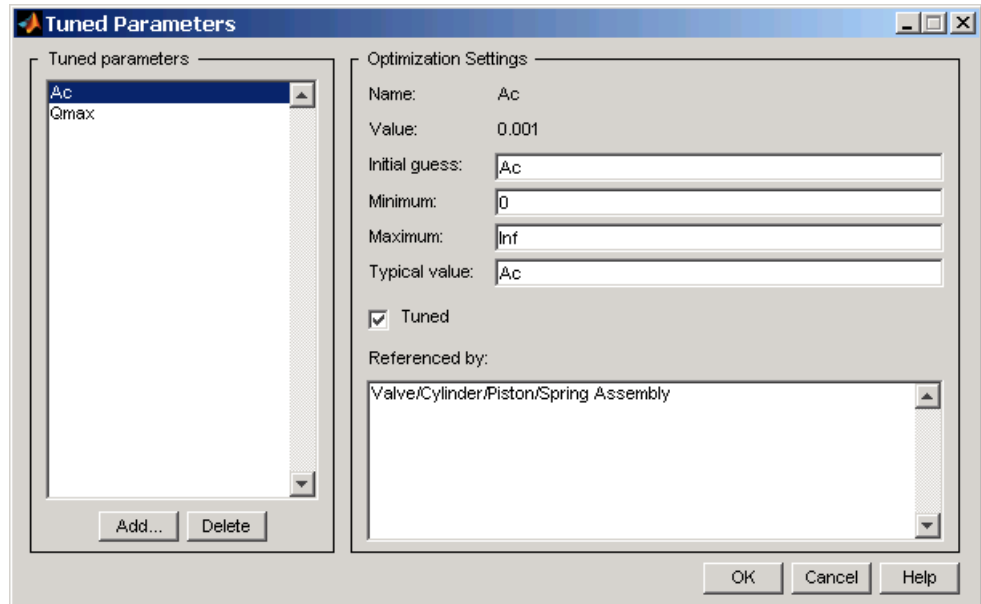
See the online documentation for more information on tuned parameter settings.

Running the Optimization

After adjusting the constraints and defining tuned parameters, start the optimization by selecting **Optimization > Start**, or by clicking the **Start** button below the menu bar in a Signal Constraint window.

In this case the optimization algorithm is not able to find a solution that satisfies all the constraints, as seen in the Optimization Progress window below.

```
Optimization Progress                                                                    _ □ ×

                            max                  Directional  First-order
 Iter   S-count       f(x)  constraint  Step-size  derivative   optimality  Procedure
   0        1          0     176.8
   1       10          0      53.4           1          0             1    infeasible
   2       15          0      51.72          1          0           863
   3       23          0      46.44       0.0521        0             1    infeasible
   4       31          0      34.63       0.125         0             1   Hessian modified twice;   infeasible
   5       39          0      11.45       0.125         0             1   Hessian modified twice;   infeasible
   6       46          0       2.286      0.25          0             1   Hessian modified;   infeasible
   7       51          0       0.2925     0.5           0             1    infeasible
   8       56          0       0.8408     0.5           0             1   Hessian modified twice;   infeasible
Could not find a solution that satisfies all constraints.
Relax the constraints or increase the constraint tolerance to find a feasible solution.

Ac =

    0.0052


Qmax =

    0.0023
```

Changing Optimization Settings

When the optimization algorithm does not find a successful solution, you can often find a better solution by changing some of the optimization settings and running the optimization again. Useful things to try are

- Changing the **Gradient type** to Refined. This is more accurate for some models.

- Increasing the tolerances.

- Using a different optimization algorithm.

For this example, change the gradient type to Refined by selecting **Optimization > Optimization Options** in the Signal Constraint window, and then selecting Refined as the **Gradient type**. Click **OK** to save the changes and close the Options dialog box. See the online documentation for more information on optimization settings.

Run the optimization again with these new settings. The optimization algorithm should now find a feasible solution as shown in the following figure.


```
Optimization Progress                                                    _ □ ×

                               max              Directional  First-order
 Iter   S-count      f(x)    constraint  Step-size  derivative  optimality  Procedure
    0        1         0        0.8408
    1        6         0        0.03851        1          0       0.00136
    2        9         0        0.005946       1          0      3.06e-005   Hessian modified
    3       12         0        0.0003484      1          0      3.19e-006   Hessian modified
Successful termination.
Found a feasible or optimal solution within the specified tolerances.

Ac =

    0.0034


Qmax =

    0.0030
```
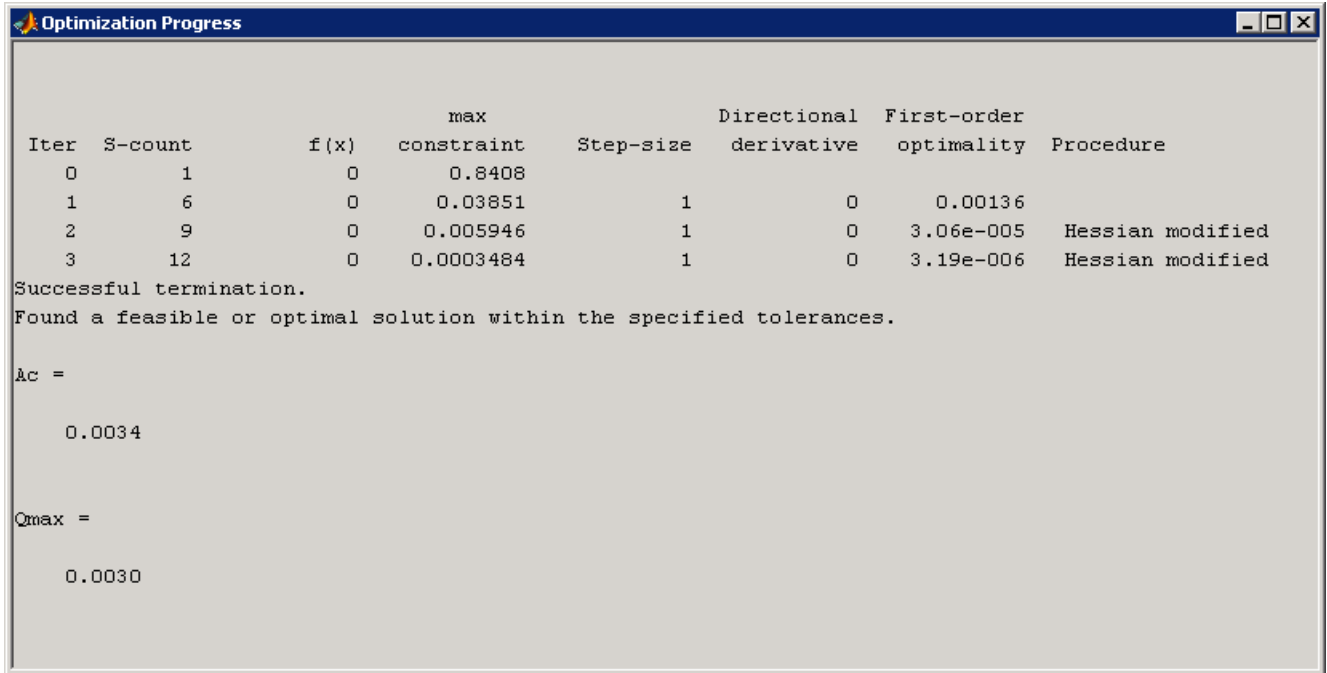
The Optimization Progress window shows the new optimized parameter values of 0.0034 m^2 for the cross-sectional area of the piston, and 0.0030 m^3/s for the pump flow rate. These values are also changed within the MATLAB workspace.

The Signal Constraint windows also show that the optimized response signals satisfy both sets of constraints. To see this more clearly, first remove all response signals from the axes by selecting **Plots > Clear Plots**. Next, plot the current response, based on the optimized parameter values, by selecting **Plots > Plot Current Response**. The Signal Constraint windows should now look like those in the following two figures for piston position and pump pressure, respectively.
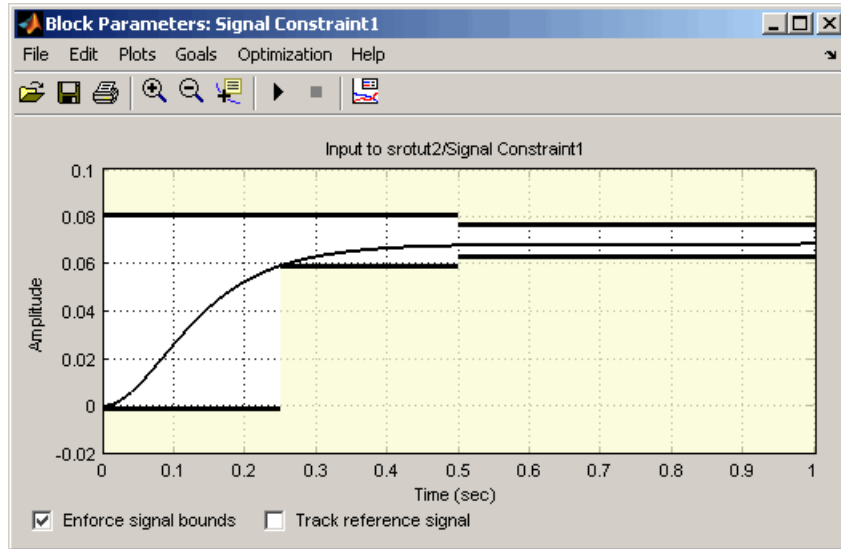
Piston Position
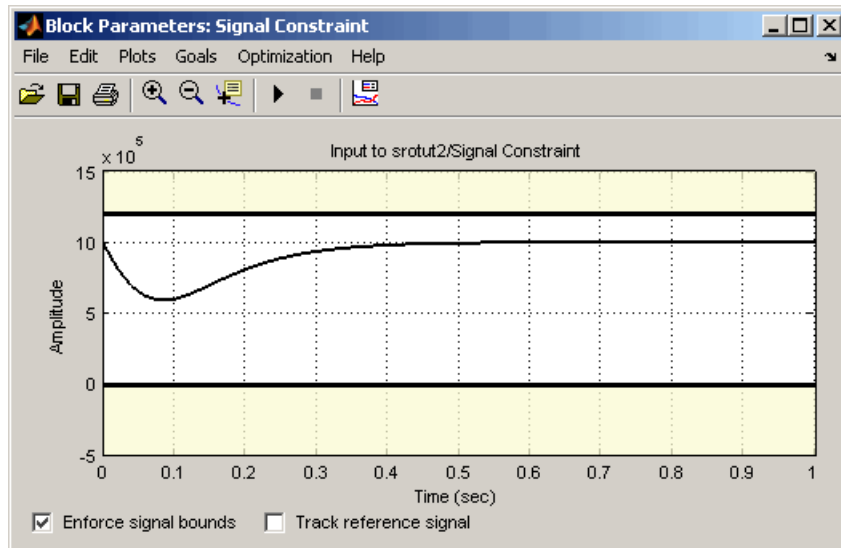
Pump Pressure

See Chapter 4, "Troubleshooting" for more advice on adjusting the response optimization to achieve the desired results.

Finding a Maximally Feasible Solution

Although the results in the previous section satisfy the constraints, the response signal for the piston position lies near the lower edge of the constraint region. This is because the optimization stops as soon as a feasible solution is found. To search for a solution that satisfies the constraints even better, you can

1 Select **Optimization > Optimization Options** from a Signal Constraint window to open the Options dialog box.

2 Select **Look for maximally feasible solution**. By selecting this option, Simulink Response Optimization will continue to search for an optimal solution, after the initial solution is found.

3 Click **OK** to save the settings and close the Options dialog box.

4 Click the **Start** button to run the optimization again.

The optimization finds a new solution that lies further inside the constraint region. The new optimized parameter values are 0.0032 m^2 for the cross-sectional area of the piston, and 0.0033 m^3/s for the pump flow rate.

Response Optimization Using Functions

In addition to the graphical user interface, Simulink Response Optimization has a command-line interface that lets you use functions to optimize the responses of signals within a Simulink model. The command-line interface only includes time-domain constraints, not frequency-domain requirements although it is possible to include parameter uncertainty with the command-line interface. This chapter includes an example outlining the use of these functions. A function reference is provided in the online documentation.

Control Design Example Using Functions

This example uses functions to optimize the response of a signal in the Simulink model `srotut1`. For a more detailed description of these functions, refer to the Function Reference in the online documentation.

Choosing Signals to Constrain

Open the Simulink model `srotut1` by typing

    srotut1

at the MATLAB prompt. The model opens and should look like that in the following figure.
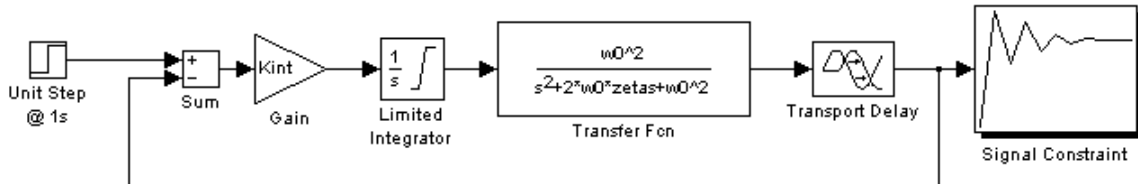
The first step in the response optimization process is to choose which signals in your Simulink model you want to constrain and to attach Signal Constraint blocks to these signals. In `srotut1` there is already a Signal Constraint block attached to a signal. Refer to the online documentation for more information on the Signal Constraint block.

Creating an Optimization Project

After attaching Signal Constraint blocks to the appropriate signals within your Simulink model, you need to create a Simulink Response Optimization project. This project is an object that contains information about the response optimization. There are two options for creating the project object:

- To create a default project with all project properties set to the default settings, use the `newsro` function.

- To create a project based on the current response optimization settings in the model, use the `getsro` function.

Creating a Default Project

Create a new project with default settings using the following command:

```
proj=newsro('srotut1',{'Kint'})
```

The first input to the `newsro` function is the model name. The second input is a cell array of the tuned parameters. In this case `Kint` is the only tuned parameter.

This returns the following object:

```
          Name: 'srotut1'
    Parameters: [1x1 ResponseOptimizer.Parameter]
  OptimOptions: [1x1 ResponseOptimizer.OptimOptions]
         Tests: [1x1 ResponseOptimizer.SimTest]
         Model: 'srotut1'

Simulink Response Optimization Project.
```

Creating a Project Based on Current Response Optimization Settings

Create a new project based on the current response optimization settings in the model using the following command. This is useful when you previously saved a project for the model and want to optimize this project at the command line or when you have set the project up with the graphical interface and would like to continue the analysis at the command line.

```
proj2=getsro('srotut1')
```

This command returns a project object with the same properties as the object returned with `newsro`, although these properties will possibly have different current values. The model must already be open to use the `getsro` function.

Properties of a Response Optimization Project

Within the project object you can set characteristics of the tuned parameters, specify uncertain parameters, position constraint bound segments, and set options for the optimization and simulations.

Specifying Tuned Parameter Attributes

To specify bounds and initial guesses for the tuned parameters, first extract the tuned parameters from the project object with the `findpar` function:

```
param=findpar(proj,'Kint')
```

This returns a tuned parameter object:

```
        Name: 'Kint'
       Value: 0
 InitialGuess: 0
     Minimum: -Inf
     Maximum: Inf
 TypicalValue: 0
 ReferencedBy: {0x1 cell}
  Description: ''
       Tuned: 1

Tuned parameter.
```

Next, use dot notation to edit these properties to specify maximum values, minimum values, initial guesses, etc. For example, to set the initial guess to `0.4` and the minimum value to `0`, use the following commands:

```
param.InitialGuess=0.4;
param.Minimum=0
```

The new parameter settings are shown below:

```
        Name: 'Kint'
       Value: 0
 InitialGuess: 0.4000
     Minimum: 0
     Maximum: Inf
 TypicalValue: 0
```

```
      ReferencedBy: {0x1 cell}
       Description: ''
             Tuned: 1
```

```
Tuned parameter.
```

For more information on specifying tuned parameters, see the online documentation.

Adjusting Signal Constraints

To define the constrain bound segments within which the optimized signal response must lie, first extract a signal constraint object from the project, using the findconstr function:

```
constr=findconstr(proj,'srotut1/Signal Constraint')
```

This function takes the project object and the location of a Signal Constraint block in the model as inputs and creates the following signal constraint object:

```
        ConstrEnable: 'on'
          isFeasible: 1
          CostEnable: 'off'
              Enable: 'on'
                Name: 'Signal Constraint'
          SignalSize: [1 1]
         LowerBoundX: [3x2 double]
         LowerBoundY: [3x2 double]
    LowerBoundWeight: [3x1 double]
         UpperBoundX: [2x2 double]
         UpperBoundY: [2x2 double]
    UpperBoundWeight: [2x1 double]
          ReferenceX: []
          ReferenceY: []
     ReferenceWeight: []
```

```
Signal Constraint.
```

To move the constraints, edit the `LowerBoundX`, `LowerBoundY`, `UpperBoundX`, and `UpperBoundY` matrices. These matrices specify the x and y positions of the endpoints of each segment. For example:

LowerBoundX gives
x-positions of all lower bound
constraint segments.

x_{b1} gives x-position of beginning of first constraint segment.

x_{e1} gives x position of endpoint of first constraint segment.

$$\text{LowerBoundX} = \begin{bmatrix} x_{b1} & x_{e1} \\ x_{b2} & x_{e2} \\ x_{b3} & x_{e3} \end{bmatrix}$$

x_{e2} gives x position of endpoint of second constraint segment.

x_{b2} gives x-position of beginning of second constraint segment.

To move the constraint bounds to the positions shown in the following figure, use the following commands:

```
constr.LowerBoundX=[0 12;12 30;30 50];
constr.LowerBoundY=[0 0;0.9 0.9;0.99 0.99];
constr.UpperBoundX=[0 30;30 50];
constr.UpperBoundY=[1.1 1.1;1.01 1.01];
```
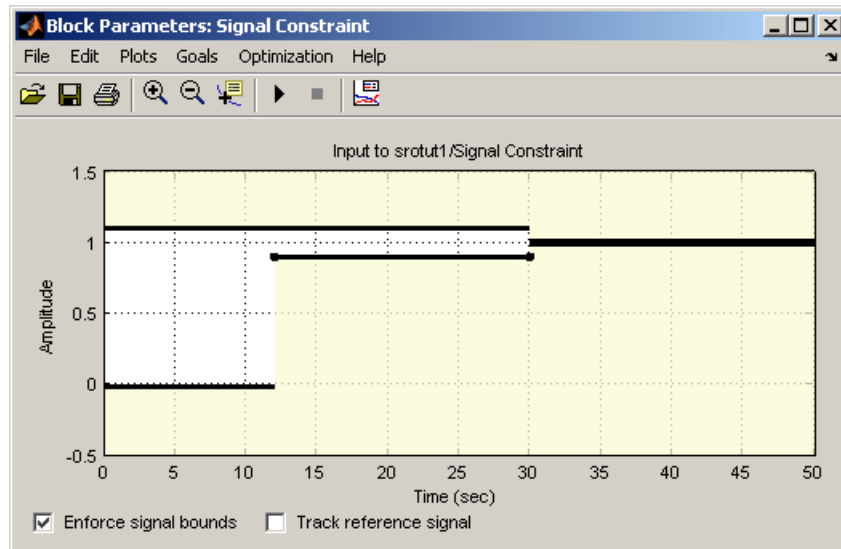
For more information on constraint bound segments, see the online documentation.

Setting Optimization Options

Before running the optimization, it is sometimes useful to change or set some optimization options. These options define the algorithms and methods Simulink Response Optimization uses to optimize the signals. For a list of all options and their uses, see in the online documentation or the documentation for the MATLAB function `optimset`.

Get the current optimization options settings with the `optimget` function:

```
optimget(proj)
```

This returns a list of the options and their current values:

```
              Algorithm: 'fmincon'
                Display: 'iter'
            GradientType: 'basic'
      MaximallyFeasible: 0
                MaxIter: 100
```

```
            TolCon: 1.0000e-003
            TolFun: 1.0000e-003
              TolX: 1.0000e-003
          Restarts: 0
      SearchMethod: []
```

Use the `optimset` function to change any values within the optimization options object. For example, to change the tolerance on the parameter values, `TolX`, to `1e-4` use the following command:

```
optimset(proj,'TolX',1e-4)
```

Setting Simulation Options

In addition to specifying optimization options, it is sometimes useful to specify simulation options. These options define the solvers, simulation times, and other settings that Simulink Response Optimization uses when simulating the models during the response optimization. For a list of all options and their uses, see in the online documentation or the documentation for the Simulink function `simset`.

Get the current simulation options settings with the `simget` function:

```
simget(proj)
```

This returns a list of the options and their current values:

```
       AbsTol: 1.0000e-006
     FixedStep: 'auto'
    InitialStep: 'auto'
        MaxStep: 'auto'
        MinStep: 'auto'
         RelTol: 1.0000e-003
         Solver: 'ode45'
       ZeroCross: 'on'
       StartTime: '0.0'
        StopTime: '50'
```

To change values within the simulation options object, use the `simset` function. For example, change the solver to `ode23` with the following command:

```
simset(proj,'Solver','ode23')
```

Specifying Uncertain Parameters

When some parameters in your model are not known exactly, but you do know the nominal plant and the level of uncertainty surrounding this model, you can include this uncertainty in your response optimization by specifying uncertain parameters in your model.

In this example, assume that w0 varies between 0.8 and 1.2 while zeta varies between 0.95 and 1.05. First, create a set of sample parameter values within these ranges using the randunc function:

```
unc_rand=randunc(2,'w0',{0.8 1.2},'zeta',{0.95 1.05});
```

The randunc function creates combinations of parameter values based on the endpoints of their uncertainty ranges. In addition, it creates several random parameter value combinations. The first argument to the function specifies the number of random parameter value combinations that the function creates. The remaining input arguments specify the uncertain parameters and the ranges over which they vary. This particular example creates two random combinations of w0 and zeta values in addition to the combinations of parameter values at the endpoints.

When you would rather specify uncertain parameter values on a grid, use the gridunc function:

```
unc_grid=gridunc('w0',{0.8 0.9 1.0 1.1 1.2},'zeta',{0.95 1 1.05});
```

The gridunc function creates combinations of the given parameters at values specified in the cell-arrays of parameter values.

For more information on creating sets of uncertain parameter value combinations, see the reference pages for randunc and gridunc.

By default, when adjusting the tuned parameters, the response optimization algorithm does not take responses based on these uncertain parameter values into account. To include an uncertain parameter combination in the optimization, you must set its Optimized property to true.

For example, to include all the parameter combinations within unc_rand, enter the following command:

```
unc_rand.Optimized(1:end)=true
```

After creating the set of uncertain parameter values, and choosing to include some or all of them in the optimization, add these values to the project object with the setunc function:

```
setunc(proj,unc_rand)
```

For more information on specifying uncertain parameter values, see the online documentation.

Running the Optimization

To run the optimization for this project, enter

```
optimize(proj)
```

The results appear in the MATLAB Command Window after each iteration. See the online documentation for more information on the optimization results.

During the optimization, Simulink Response Optimization changes the tuned parameter values in the MATLAB workspace. The optimization also displays the new, optimized parameter values after terminating. In this example, the optimized value of Kint is 0.1539.

Troubleshooting

Where possible, Simulink Response Optimization provides visual cues to help you formulate problems and inform you about the progress of an optimization. However, sometimes problems can occur when optimizing responses. This chapter lists common problems along with recommendations for dealing with them.

Common Questions About Response Optimization (p. 4-2)	Solutions for commonly encountered response optimization problems

Common Questions About Response Optimization

The following list of questions represent commonly encountered problems with response optimization. Solutions, advice, and tips are given for each question.

Questions

How do I quit an optimization and revert to my initial parameter values?

- When using a Signal Constraint block within a Simulink model, click the **Stop** button or select **Optimization > Stop** in the Signal Constraint window to stop the optimization, and then select **Edit > Undo Optimize Parameters** to revert to your initial parameter values.

- When using a SISO Design Task, the **Start Optimization** button becomes a **Stop Optimization** button after the optimization has begun. To quit the optimization, click the **Stop Optimization** button. To revert to the initial parameter values, select **Edit > Undo Optimize compensators** from the menu in the SISO Design Tool window.

Why don't the responses and parameter values change at all?

- The optimization problem you formulated might be nonsmooth. This means that small parameter changes have no effect on the amount by which response signals satisfy or violate the constraints and only large changes will make a difference. Try switching to a search-based algorithm such as simplex search or pattern search. Alternatively, look for initial guesses outside of the dead zone where parameter changes have no effect. If you are directly optimizing the response of a Simulink model using a Signal Constraint block, you could also try removing nonlinear blocks such as the Quantizer or Dead Zone block.

- If you are using the Refined option for **Gradient type** with the gradient descent algorithm, try the Basic option for **Gradient type** instead. The gradient model that the Refined option uses might be invalid for your problem.

Why doesn't the optimization get close to an acceptable solution?

- If you're using gradient descent, the default algorithm, try the Refined option for **Gradient type**. This option yields more accurate gradient estimates when using variable-step solvers and can facilitate convergence.

- If you are using pattern search, check that you have specified appropriate maximum and minimum values for all your tuned parameters or compensator elements. The pattern search algorithm looks inside these bounds for a solution. When they are set to their default values of Inf and -Inf, the algorithm searches within ±100% of the initial values of the parameters. In some cases this region is not large enough and changing the maximum and minimum values can expand the search region.

- Your optimization problem might have local minima. Consider running one of the search-based algorithms first to get closer to an acceptable solution.

- Reduce the number of tuned parameters and compensator elements by removing from the **Tuned parameters** list (when using a Signal Constraint block) or from the **Compensators** pane (when using a SISO Design Task) those parameters that you know only mildly influence the optimized responses. After you identify reasonable values for the key parameters, add the fixed parameters back to the tunable list and restart the optimization using these reasonable values as initial guesses.

Why does the optimization terminate before exceeding the maximum number of iterations, with a solution that does not satisfy all the constraints or design requirements?

- It might not be possible to achieve your specifications. Try relaxing the constraints or design requirements that the response signals violate the most. After you find an acceptable solution to the relaxed problem, tighten some constraints again and restart the optimization.

- The optimization might have converged to a local minimum that is not a feasible solution. Restart the optimization from a different initial guess and/or use one of the search-based methods to identify another local minimum that satisfies the constraints.

Why does the optimization drive the tuned compensator elements and parameters to undesirable values?

- When you know that a tuned compensator element or parameter should remain positive, or when its value is physically constrained to a given range, enter this information in the Tuned Parameters dialog box (from a Signal Constraint block) or the **Compensators** pane (in a SISO Design Task) as lower and upper bounds (**Minimum** and **Maximum**). This information helps guide the optimization algorithm toward a reasonable solution.

- In the Tuned Parameters dialog box (from a Signal Constraint block) or the **Compensators** pane (in a SISO Design Task), specify initial guesses that are within the range of desirable values.

Why is the optimization taking a long time to converge even though it is close to a solution?

- In a Signal Constraint window, use the Stop button, or select **Optimization > Stop**, to interrupt the optimization when you think the current optimized response signals are acceptable. When using a SISO Design Task, click **Stop Optimization** button in the **Optimization** panel of the **Response Optimization** node of the Control and Estimation Tools Manager, the when you think the current optimized response signals are acceptable.

- If you use the gradient descent algorithm, try restarting the optimization. This resets the Hessian estimate and might speed up convergence.

- Increase the convergence tolerances in the Optimization Options dialog to force earlier termination.

- Relax some of the constraints or design requirements to increase the size of the feasibility region.

What to do if the response becomes unstable and does not recover?

While the optimization formulation has explicit safeguards against unstable or divergent response signals, the optimization can sometimes venture into an unstable region where simulation results become erratic and gradient

methods fail to find a way back to the stable region. In these cases, you can try one of the following solutions:

- Add or tighten the lower and upper bounds on compensator element and parameter values. Instability often occurs when you allow some parameter values to become too large.

- Use a search-based algorithm to find parameter values that stabilize the response signals and then start the gradient-based algorithm using these initial values.

- When optimizing responses in a SISO Design Task, you can try adding additional design requirements that achieve the same or similar goal. For example, in addition to a settling time design requirement on a step response plot, you could add a settling time design requirement on a root-locus plot that restricts the location of the real parts of the poles. By adding overlapping design requirements in this way, you can force the optimization to meet the requirements.

Why does the optimization stall?

When using a Signal Constraint block to directly optimize a Simulink model, certain parameter combinations can make the simulation stall for models with strong nonlinearities or frequent mode switching. In these cases the ODE solvers take smaller and smaller step sizes. Stalling can also occur when the model's ODEs become too stiff for some parameter combinations. A symptom of this behavior is when the Simulink model status is `Running` and clicking the **Stop** button fails to interrupt the optimization. When this happens, you can try one of the following solutions:

- Switch to a different ODE solver, especially one of the stiff solvers.

- Specify a minimum step size.

- Disable zero crossing detection if chattering is occurring.

- Tighten the lower and upper bounds on parameters that cause simulation difficulties. In particular, eliminate regions of the parameter space where some model assumptions are invalid and the model behavior can become erratic.

How do I accelerate the optimization?

- Since the time it takes to simulate the model dominates the optimization time, when you use a Signal Constraint block to directly optimize a Simulink model you can also use the Simulink Accelerator to dramatically reduce the optimization time.

- The choice of ODE solver can also significantly affect the overall optimization time. Use a stiff solver when the simulation takes many small steps, and use a fixed-step solver when such solvers yield accurate enough simulations for your model. (These solvers must be accurate in the entire parameter search space.)

- Reduce the number of tuned compensator elements or parameters and constrain their range to narrow the search space.

- When specifying parameter uncertainty (not available when optimizing responses in a SISO Design Task), keep the number of sample values small since the number of simulations grows proportionally with the number of samples. For example, a grid of 3 parameters with 10 sample values for each parameter requires 10^3=1000 simulations per iteration.

How should I pick the start and stop time?

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

- When using a SISO Design Task: Simulink Response Optimization automatically sets the model's simulation start and stop time and you cannot directly change them. By default, the simulation starts at 0 and continues until the SISO Design Task determines that the dynamics of the model have settled out. In addition, when the design requirements extend beyond this point, the simulation continues to the extent of the design requirements. Although you cannot directly adjust the start or stop time of

the simulation, you can adjust the design requirements to extend further in time and thus force the simulation to continue to a certain point.

Should I worry about the scale of my responses and how constraints and design requirements are discretized?

No. Simulink Response Optimization automatically normalizes constraints, design requirement and response data and, unlike its predecessor, the Nonlinear Control Design Blockset, it does not discretize the constraints or design requirements.

Index