### **Control Design Steps**

June 2008, 371003B-01

Use this help file to learn about Control Design steps.

© 2004–2008 National Instruments Corporation. All rights reserved.

### **Controller Design**

Use the Controller Design steps to create controller models.

### **Interactive Bode**

Creates a SISO controller based on the Bode analysis technique.

Parameter	Description
Add Real Pole	Adds a real pole at a specific frequency on the Interactive Open-Loop Bode Magnitude plot. Click the Add Real Pole button, then click the location on the plot at which you want to place the pole.
Add Complex Pole	Adds a complex pole to a specific frequency on the <b>Interactive Open-Loop Bode Magnitude</b> plot. Click the <b>Add Complex Pole</b> button, then click the location on the plot at which you want to place the pole.
Add Real Zero	Adds a real zero at a specific frequency on the Interactive Open-Loop Bode Magnitude plot. Click the Add Real Zero button, then click the location on the plot at which you want to place the zero.
Add Complex Zero	Adds a complex zero at a specific frequency on the <b>Interactive Open-Loop Bode Magnitude</b> plot. Click the <b>Add Complex Zero</b> button, then click the location on the plot at which you want to place the zero.
Remove Pole or Zero	Removes the controller pole or zero from the <b>Interactive Open-Loop Bode Magnitude</b> plot.
Move Pole or Zero	Enables you to click and drag a controller pole or zero around the <b>Interactive Open-Loop Bode Magnitude</b> plot.
Interactive Open-Loop Bode Magnitude	Displays the Bode magnitude of an open-loop transfer function defined by the loop transfer function, which calculates the controller in series with the plant and sensor (CPH). Closed-loop poles appear red. You can move the closed-loop poles to change the closed-loop gain by clicking and dragging the poles on the graph. You can tune the gain by changing position of the closed-loop poles. Controller poles and zeros appear blue. Click the buttons above the graph then click the controller poles and zeros on the plot to add, move, or delete the controller poles and zeros. Open-loop poles

	or zeros appear gray. You cannot move open-loop poles or zeros.
Interactive Bode Phase	Displays the phase (in degrees) of the model plotted against a set of frequencies. Closed-loop poles appear red. You can move the closed-loop poles to change the closed-loop gain by clicking and dragging the poles on the graph. You can tune the gain by changing position of the closed-loop poles. Controller poles and zeros appear blue. Click the buttons above the graph then click the controller poles and zeros on the plot to add, move, or delete the controller poles and zeros. Open-loop poles or zeros appear gray. You cannot move open-loop poles or zeros.
Stable/Unstable	Indicates whether the resulting closed-loop system with the controller that this step creates is stable.
Model Input	<ul> <li>Contains the following parameters:</li> <li>Plant (P)—Specifies the model to use as the plant (P) in the structure.</li> <li>Sensor (H)—Specifies the model in the feedback loop to use as the sensor (H) in the structure.</li> <li>Filter (F)—Specifies the model in the feedthrough with the feedback loop to use as the filter (F) in the structure.</li> <li>Controller (C)—Specifies the initial system controller (C) to use in the structure. If you place a checkmark in the Controller (C) checkbox, you can select the initial model that represents the controller. If you make changes to this model of the controller in a previous step, this step does not update the controller information automatically. You must click the Initialize controller button to update the initial model information that this step uses for the controller.</li> </ul>

	<ul> <li>controller. If you make changes to the model of the controller in a previous step, this step does not update the controller information until you click the Initialize controller button.</li> <li>Feedback—Specifies the type of feedback loop.</li> </ul>
Model Output	Contains the following options:
	<ul> <li>Export Models—Contains the following parameters:</li> </ul>
	<ul> <li>Output controller (c)—Specifies that the step returns the final controller system.</li> </ul>
	<ul> <li>Closed loop (r-y)—Specifies that the step returns the complete closed-loop system.</li> </ul>
	<ul> <li>Control output (r-u)—Specifies that the step returns the equivalent system to analyze the control effort, or the signal applied to the input of the system.</li> </ul>
	<ul> <li>Loop transfer (CPH)—Specifies that the step returns the equivalent system for loop transfer in the series controller-plant-sensor.</li> </ul>
	<ul> <li>Plant sensitivity (dy-y)—Specifies that the step returns the equivalent system model for output sensitivity or the model from the output y against a disturbance in the output y.</li> </ul>
	<ul> <li>Output sensitivity (du-y)—Specifies that the step returns the equivalent system model for plant sensitivity or a variation of y against a disturbance in the input u.</li> </ul>
	<ul> <li>Sensor sensitivity (dh-y)—Specifies that the step returns the equivalent</li> </ul>

	<ul> <li>system model for sensor sensitivity or the model from the output <i>y</i> against a disturbance after the sensor <i>H</i>.</li> <li>Sampling time (s)—Defines the smallest sampling time used in the discrete models. If the system model is continuous or has a higher sampling rate, this step discretizes the model using zero-order-hold and the smallest sampling time.</li> </ul>
Controller Synthesis	<ul> <li>Contains the following options:</li> <li>Autoscale magnitude—Automatically scales the y-axis of the magnitude on the Interactive Open-Loop Bode Magnitude graph.</li> <li>Autoscale phase—Automatically scales the y-axis of the phase on the Interactive Bode Phase graph.</li> <li>Autoscale frequency—Automatically scales the x-axis of the frequency on the Interactive Bode Phase graphs.</li> <li>Gain—Specifies the gain the step uses in the feedback loop.</li> <li>Controller zeros—Defines the array of zeros of the model. The zeros can be real or complex. If they are complex, they must be in complex conjugate pairs when you enter the real and imaginary parts followed by the symbol, <i>i</i>. For example, if you type -1 + 0.5i, the step generates the complex conjugate -1 ± 0.5i, which is equivalent to (-1 + 0.5i) * (-1 - 0.5i).</li> <li>Controller poles—Defines the array of poles of the model. The poles can be real or complex. If they are complex they must be in complex conjugate pairs when you enter the real and imaginary parts followed by the symbol, <i>i</i>. For example, if you type -1 + 0.5i, the step generates the complex conjugate -1 ± 0.5i, which is equivalent to (-1 + 0.5i) * (-1 - 0.5i).</li> <li>Controller poles—Defines the array of poles of the model. The poles can be real or complex. If they are complex, they must be in complex conjugate pairs. The step automatically calculates the complex followed by the symbol, <i>i</i>. For example, if you type -1 + 0.5i, the step generates the complex conjugate -1 ± 0.5i, which is equivalent to (-1 + 0.5i) * (-1 - 0.5i).</li> <li>Controller poles—Defines the array of poles of the model. The poles can be real or complex. If they are complex, they must be in complex conjugate pairs. The step automatically calculates the complex</li> </ul>

conjugate pairs when you enter the real and imaginary parts followed by the symbol, *i*. For example, if you type -1 + 0.5i, the step generates the complex conjugate  $-1 \pm 0.5i$ , which is equivalent to (-1 + 0.5i) \* (-1 - 0.5i).

## **PID Synthesis**

Creates a controller based on changing the PID (proportional-integralderivative) series algorithm as defined as below:

 $\frac{U(s)}{E(s)} = \mathcal{K}_{c}(1 + \frac{1}{T_{i}s})(\frac{T_{d}s + 1}{(T_{1}s + 1)(T_{2}s + 1)})$ 

where  $K_c$  is the proportional gain,

 $T_i$  is the integral time in seconds,

and  $T_d$  is the derivative time in seconds.

 $T_1$  is the **HF Rolloff 1** time (in seconds), which defines the time constant of a first order filter in series with the PID algorithm.

 $T_2$  is the **HF Rolloff 2** time (in seconds), which defines the time constant of a first order filter in series with the PID algorithm.

#### **Details**

Parameter	Description
Step Response	Displays the output of the system when it is excited by a step input.
Autoscale time	Automatically scales the x-axis on the <b>Step Response</b> graph.
Stable/Unstable	Indicates whether the resulting closed-loop system with the controller that this step creates is stable.
Model Input	<ul> <li>Contains the following parameters:</li> <li>Plant (P)—Specifies the model to use as the plant (P) in the structure.</li> <li>Sensor (H)—Specifies the model in the feedback loop to use as the sensor (H) in the structure.</li> <li>Filter (F)—Specifies the model in the feedthrough with the feedback loop to use as the filter (F) in the structure.</li> <li>Feedback—Specifies the type of feedback loop.</li> </ul>
Model Output	Contains the following options:

- **Export Models**—Contains the following parameters:
  - Output controller (c)—Specifies that the step returns the final controller system.
  - Closed loop (r-y)—Specifies that the step returns the complete closed-loop system.
  - **Control output (r-u)**—Specifies that the step returns the equivalent system to analyze the control effort, or the signal applied to the input of the system.
  - **Loop transfer (CPH)**—Specifies that the step returns the equivalent system for loop transfer in the series controller-plant-sensor.
  - **Plant sensitivity (dy-y)**—Specifies that the step returns the equivalent system model for output sensitivity or the model from the output *y* against a disturbance in the output *y*.
  - Output sensitivity (du-y)—Specifies that the step returns the equivalent system model for plant sensitivity or a variation of y against a disturbance in the input u.
  - **Sensor sensitivity (dh-y)**—Specifies that the step returns the equivalent system model for sensor sensitivity or the model from the output *y* against a disturbance after the sensor *H*.
- **Sampling time (s)**—Defines the smallest sampling time used in the discrete models. If the system model is continuous or has a higher sampling rate, this step discretizes the model using zero-order-hold and the smallest

	sampling time.
Controller Synthesis	<ul> <li>Contains the following parameters:</li> <li>Gain—Represents the proportional gain (Kc) of the controller. The default is 1.</li> </ul>
	<ul> <li>Integral (s)—Specifies the controller parameter (Ti) that adjusts the effect of the error integral term on the controller output.</li> </ul>
	<ul> <li>Derivative (s)—Specifies the controller parameter (Td) that adjusts the effect of the error derivative term on the controller output.</li> </ul>
	• <b>HF Rolloff 1 (s)</b> —Defines the cutoff frequency of a low frequency first order filter. If you place a checkmark in the <b>Derivative (s)</b> checkbox, the step automatically places a checkmark in the <b>HF Rolloff 1 (s)</b> checkbox.
	• <b>HF Rolloff 2 (s)</b> —Defines the cutoff frequency of a second low frequency first order filter.

### **PID Synthesis Details**

This step accepts four model inputs: plant, sensor, filter, and initial controller. Each input can be continuous or discrete and can have a unique sampling time. The PID Synthesis step determines the appropriate model representation for synthesis. For example, if all model inputs are continuous, the resulting synthesized PID controller also is continuous. If one or more input models is discrete, the resulting synthesized PID controller is a discrete controller sampled with the smallest sampling time of the discrete inputs.

Internally, all input models are resampled to use the smallest sampling time of the discrete input models before synthesis proceeds. All discretization is performed using the Zero-Order-Hold algorithm. If you want to use a different discretization method, use the <u>Discretize Model</u> step on all models before running the PID Synthesis step.

### **Root Locus**

Creates a controller based on the root locus technique.

Parameter	Description
Add Real Pole	Adds a single pole to the controller on the real axis. The imaginary part of the pole is zero. To add the pole to the plot, click the <b>Add Real Pole</b> button then click the location on the plot at which you want to place the pole. You can move the pole after you place it on the plot by clicking the <b>Move Pole or Zero</b> button and then clicking and dragging the pole around the plot.
Add Complex Pole	Adds a complex conjugate pole to the controller. To add the pole to the plot, click the <b>Add Complex Pole</b> button then click the location on the plot at which you want to place the pole. The step automatically adds the complex conjugate to the plot. You can move the pole after you place it on the plot by clicking the <b>Move</b> <b>Pole or Zero</b> button and then clicking and dragging the pole around the plot.
Add Real Zero	Adds a single zero to the controller on the real axis. The imaginary part of the zero is zero. To add the zero to the plot, click the <b>Add Real Zero</b> button then click the location on the plot at which you want to place the zero. You can move the zero after you place it on the plot by clicking the <b>Move Pole or Zero</b> button and then clicking and dragging the zero around the plot.
Add Complex Zero	Adds a complex conjugate zero to the controller. To add the zero to the plot, click the <b>Add Complex Zero</b> button then click the location on the plot at which you want to place the zero. The step automatically adds the complex conjugate to the plot. You can move the zero after you place it on the plot by clicking the <b>Move</b> <b>Pole or Zero</b> button and then clicking and dragging the zero around the plot.
Remove Pole or Zero	Removes the controller pole or zero from the <b>Interactive Open-Loop Bode Magnitude</b> plot.

Move Pole or Zero	Enables you to click and drag a controller pole or zero around the <b>Interactive Open-Loop Bode Magnitude</b> plot.
Interactive Root Locus	Plots the root locus controller. Closed-loop poles appear red. You can move the closed-loop poles to change the closed-loop gain by clicking and dragging the poles on the graph. You can tune the gain by changing position of the closed-loop poles. Controller poles and zeros appear blue. Click the buttons above the graph then click the controller poles and zeros on the plot to add, move, or delete the controller poles and zeros. Open-loop poles or zeros appear gray. You cannot move open-loop poles or zeros.
Stable/Unstable	Indicates whether the resulting closed-loop system with the controller that this step creates is stable.
Model Input	<ul> <li>Contains the following parameters:</li> <li>Plant (P)—Specifies the model to use as the plant (P) in the structure.</li> <li>Sensor (H)—Specifies the model in the feedback loop to use as the sensor (H) in the structure.</li> <li>Filter (F)—Specifies the model in the feedthrough with the feedback loop to use as the filter (F) in the structure.</li> <li>Controller (C)—Specifies the initial system controller (C) to use in the structure. If you place a checkmark in the Controller (C) checkbox, you can select the initial model that represents the controller. If you make changes to this model of the controller in a previous step, this step does not update the controller information automatically. You must click the Initialize controller button to update the initial model information that this step uses for the controller.</li> </ul>

	<ul> <li>controller. If you make changes to the model of the controller in a previous step, this step does not update the controller information until you click the Initialize controller button.</li> <li>Feedback—Specifies the type of feedback loop.</li> </ul>
Model Output	Contains the following options:
	<ul> <li>Export Models—Contains the following parameters:</li> </ul>
	<ul> <li>Output controller (c)—Specifies that the step returns the final controller system.</li> </ul>
	<ul> <li>Closed loop (r-y)—Specifies that the step returns the complete closed-loop system.</li> </ul>
	<ul> <li>Control output (r-u)—Specifies that the step returns the equivalent system to analyze the control effort, or the signal applied to the input of the system.</li> </ul>
	<ul> <li>Loop transfer (CPH)—Specifies that the step returns the equivalent system for loop transfer in the series controller-plant-sensor.</li> </ul>
	<ul> <li>Plant sensitivity (dy-y)—Specifies that the step returns the equivalent system model for output sensitivity or the model from the output y against a disturbance in the output y.</li> </ul>
	<ul> <li>Output sensitivity (du-y)—Specifies that the step returns the equivalent system model for plant sensitivity or a variation of y against a disturbance in the input u.</li> </ul>
	<ul> <li>Sensor sensitivity (dh-y)—Specifies that the step returns the equivalent system model for sensor sensitivity or</li> </ul>

	<ul> <li>the model from the output <i>y</i> against a disturbance after the sensor <i>H</i>.</li> <li>Sampling time (s)—Defines the smallest sampling time used in the discrete models. If the system model is continuous or has a higher sampling rate, this step discretizes the model using zero-order-hold and the smallest sampling time.</li> </ul>
Controller Synthesis	<ul> <li>Contains the following options:</li> <li>Gain—Specifies the gain the step uses in the feedback loop.</li> <li>Center to poles—Automatically adjusts the graph scales to center the plot in the dynamics of the system.</li> <li>Controller zeros—Defines the array of zeros of the model. The zeros can be real or complex. If they are complex, they must be in complex conjugate pairs. The step automatically calculates the complex conjugate pairs when you enter the real and imaginary parts followed by the symbol, <i>i</i>. For example, if you type -1 + 0.5i, the step generates the complex conjugate -1 ± 0.5i, which is equivalent to (-1 + 0.5i) * (-1 - 0.5i).</li> <li>Controller poles—Defines the array of poles of the model. The poles can be real or complex. If they are complex, they must be in complex conjugate to (-1 + 0.5i) * (-1 - 0.5i).</li> <li>Controller poles—Defines the array of poles of the model. The poles can be real or complex. If they are complex, they must be in complex conjugate pairs. The step automatically calculates the complex conjugate pairs. The step automatically calculates the complex the real and imaginary parts followed by the symbol, <i>i</i>. For example, if you type -1 + 0.5i, the step generates the complex conjugate -1 ± 0.5i, which is equivalent to (-1 + 0.5i) * (-1 - 0.5i).</li> </ul>

### **Create Models**

Use the Create Models steps to create new system models.

## **Special Transfer Function**

Generates a transfer function model using the PID series algorithm as defined below:

 $\frac{U(s)}{E(s)} = \mathcal{K}_{c}(1 + \frac{1}{T_{i}s})(\frac{T_{d}s + 1}{(T_{1}s + 1)(T_{2}s + 1)})$ 

where  $K_c$  is the proportional gain,

 $T_i$  is the integral time in seconds,

and  $T_d$  is the derivative time in seconds.

 $T_1$  is the **High frequency rolloff 1** time (in seconds), which defines the time constant of a first order filter in series with the PID algorithm.

 $T_2$  is the **High frequency rolloff 2** time (in seconds), which defines the time constant of a first order filter in series with the PID algorithm.

Parameter	Description
PID Transfer Function Model	Displays the mathematical expression that defines the PID transfer function model you create.
Proportional gain	Represents the proportional gain ( $K_c$ ) of the controller. The default is 1.
Integral time (s)	Specifies the controller parameter $(T_i)$ that adjusts the effect of the error integral term on the controller output.
Derivative time (s)	Specifies the controller parameter ( $T_d$ ) that adjusts the effect of the error derivative term on the controller output.
High frequency rolloff 1 (s)	Defines the cutoff frequency of a low frequency first order filter. If you place a checkmark in the <b>Derivative time (s)</b> checkbox, the step automatically places a checkmark in the <b>High frequency rolloff 1 (s)</b> checkbox.
High frequency rolloff 2 (s)	Defines the cutoff frequency of a second low frequency first order filter.

### **State-Space**

Creates a single-input single-output (SISO) state-space model of a system using **Matrix A**, **Matrix B**, **Matrix C**, **Matrix D**, and the **Sampling Time (s)**. This step also can produce a state-space model in which you can specify the data in symbolic form.

#### **Details**

Parameter	Description
Name	Specifies the name of the model you create.
Inputs	Displays the number of inputs in the system model.
States	Displays the number of states in the system model.
Outputs	Displays the number of outputs in the system model.
Model Type	<ul> <li>Specifies the type of model you create. Contains the following options:</li> <li>Continuous—Specifies that the model you create is in the continuous time domain.</li> <li>Discrete—Specifies that the model you create is in the discrete time domain. When you place a</li> </ul>
	<ul> <li>checkmark in the Discrete checkbox, the Sampling Time (s) parameter appears.</li> <li>Sampling Time (s)—Defines the sampling time used in the discrete model. The default is 1. By using the coefficients of the continuous model and setting the Sampling Time (s) of the system to a value greater than zero, you do not create the discrete-time equivalent of the system. If you create a continuous model and want to convert the model to a discrete model, you must use the Discretize Model step.</li> <li>If you already know the coefficients of a discrete the model using this step. Enter the coefficients for the discrete model and set the Sampling Time (s) of the system to a value greater than zero.</li> </ul>

Symbolic Coefficients	<ul> <li>Specifies that you can use variables to define the model. Define the variable name and the value that the variable represents in the left and right columns, respectively, that appear in the Symbolic coefficients section.</li> <li>Number of Variables—Specifies the maximum number of variables used. The maximum number of variables you can specify is nine.</li> <li>Variables—Lists the coefficient variables and values. The number of variables you specify in the Number of Variables control corresponds to the number of rows in the Variables array.</li> </ul>
Matrix A	Defines the system matrix that describes the dynamics of the system.
Matrix B	Defines the input matrix that relates the inputs to the states of the system.
Matrix C	Defines the output matrix that relates the outputs to the state of the system.
Matrix D	Defines the transmission matrix that relates the inputs to the outputs of the systems.
Generate Random Model	Generates a random model.

### **State-Space Details**

The state-space model is defined by the following equations:

Continuous  $x = \mathbf{A}x + \mathbf{B}u$   $y = \mathbf{C}x + \mathbf{D}u$ Discrete  $x(k+1) = \mathbf{A}x(k) + \mathbf{B}u(k)$  $y(k) = \mathbf{C}x(k) + \mathbf{D}u(k)$ 

where k is the sampling time,

*n* is the number of states,

*m* is the number of inputs,

*r* is the number of outputs,

x is the state vector,

*u* is the input vector,

y is the output vector,

**A** is an  $n \times n$  state matrix of the given system,

**B** is an  $n \times m$  input matrix of the given system,

**C** is an  $r \times n$  input matrix of the given system,

and **D** is an  $r \times m$  input matrix of the given system.

## **Transfer Function**

Creates a single-input single-output (SISO) transfer function model of a system using the **Sampling Time (s)**, **Numerator**, and **Denominator**. This step also can produce a transfer function model in which the data is specified in symbolic form.

### **Details**

Parameter	Description
Name	Specifies the name of the model you create.
Inputs	Displays the number of inputs in the system model.
Outputs	Displays the number of outputs in the system model.
Model Type	<ul> <li>Specifies the type of model you create. Contains the following options:</li> <li>Continuous—Specifies that the model you create is in the continuous time domain.</li> <li>Discrete—Specifies that the model you create is in the discrete time domain. When you place a checkmark in the Discrete checkbox, the Sampling Time (s) parameter appears.</li> <li>Sampling Time (s)—Defines the sampling time used in the discrete model. The default is 1. By using the coefficients of the continuous model and setting the Sampling Time (s) of the system to a value greater than zero, you do not create the discrete-time equivalent of the system. If you create a continuous model and want to convert the model to a discrete model, you must use the Discretize Model step.</li> <li>If you already know the coefficients of a discrete transfer function model, you can create the model using this step. Enter the Sampling Time (s) of the system to a value greater than zero.</li> </ul>
Symbolic Coefficients	Specifies that you can use variables to define the model. Define the variable name and the value that the variable

	represents in the left and right columns, respectively, that appear in the <b>Symbolic coefficients</b> section.
	<ul> <li>Number of Variables—Specifies the maximum number of variables used. The maximum number of variables you can specify is nine.</li> </ul>
	<ul> <li>Variables—Lists the coefficient variables and values. The number of variables you specify in the Number of Variables control corresponds to the number of rows in the Variables array.</li> </ul>
Resulting Model	Displays the mathematical expression that defines the model you create.
Numerator	Defines the constant coefficients of a polynomial that represent the numerator of a SISO transfer function model. The <i>i</i> <sup>th</sup> element of <b>Numerator</b> is the coefficient of the <i>i</i> <sup>th</sup> order term of the polynomial. The index is zero- based.
Denominator	Defines the constant coefficients of a polynomial that represent the denominator of a SISO transfer function model. The <i>i</i> <sup>th</sup> element of <b>Denominator</b> is the coefficient of the <i>i</i> <sup>th</sup> order term of the polynomial. The index is zero- based.
Generate Random Model	Generates a random model.

### **Transfer Function Details**

The transfer function model is defined by the following equations:

**Continuous**  $H(s) = \frac{b_0 + b_1 s + \dots + b_{m-1} s^{m-1} + b_m s^m}{a_0 + a_1 s + \dots + a_{n-1} s^{n-1} + a_n s^n}$ **Discrete**  $H(z) = \frac{b_0 + b_1 z + \dots + b_{m-1} z^{m-1} + b_m z^m}{a_0 + a_1 z + \dots + a_{n-1} z^{n-1} + a_n z^n}$ 

### Zero-Pole-Gain

Creates a single-input single-output (SISO) zero-pole-gain model of a system using the **Zeros**, **Poles**, **Gain**, and **Sampling Time (s)**. This step also can produce a zero-pole-gain model in which the data is specified in symbolic form.

### **Details**

Parameter	Description
Name	Specifies the name of the model you create.
Inputs	Displays the number of inputs in the system model.
Outputs	Displays the number of outputs in the system model.
Model Type	Specifies the type of model you create. Contains the following options:
	<ul> <li>Continuous—Specifies that the model you create is in the continuous time domain.</li> </ul>
	<ul> <li>Discrete—Specifies that the model you create is in the discrete time domain. When you place a checkmark in the Discrete checkbox, the Sampling Time (s) parameter appears.</li> </ul>
	• Sampling Time (s)—Defines the sampling time used in the discrete model. The default is 1. By using the coefficients of the continuous model and setting the Sampling Time (s) of the system to a value greater than zero, you do not create the discrete-time equivalent of the system. If you create a continuous model and want to convert the model to a discrete model, you must use the Discretize Model step.
	If you already know the coefficients of a discrete transfer function model, you can create the model using this step. Enter the coefficients for the discrete model and set the <b>Sampling Time (s)</b> of the system to a value greater than zero.
Symbolic Coefficients	Specifies that you can use variables to define the model. Define the variable name and the value that the variable

	represents in the left and right columns, respectively, that appear in the <b>Symbolic coefficients</b> section.
	<ul> <li>Number of Variables—Specifies the maximum number of variables used. The maximum number of variables you can specify is nine.</li> <li>Variables—Lists the coefficient variables and values. The number of variables you specify in the Number of Variables control corresponds to the number of rows in the Variables array.</li> </ul>
Resulting Model	Displays the mathematical expression that defines the model you create.
Gain	Defines the scalar gain of the model. The default is 1.
Zeros	Defines the array of zeros of the model. The zeros can be real or complex. If they are complex, they must be in complex conjugate pairs.
Poles	Defines the array of poles of the model. The poles can be real or complex. If they are complex, they must be in complex conjugate pairs.
Generate Random Model	Generates a random model.

### **Zero-Pole-Gain Details**

The zero-pole-gain model is defined by the following equations:

**Continuous**  $H(s) = \frac{k(s-Z_1)(s-Z_2)...(s-Z_m)}{(s-P_1)(s-P_2)...(s-P_n)}$ **Discrete**  $H(z) = \frac{k(z-Z_1)(z-Z_2)...(z-Z_m)}{(z-P_1)(z-P_2)...(z-P_n)}$ 

### Import/Export Model

Use the Import/Export Model steps to save or load a model.

# Load Control Design Model

Opens a model (.lti) file and reads all the records in the file. A .lti file stores control design and linear time-invariant models. Each record contains a separate model. To retrieve all records in the file, <Ctrl>-click or <Shift>-click the records in the **Information in file** table.

Parameter	Description
File name	Specifies the name and location of the file you want to import. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the <b>My Documents</b> folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project.
Information in file	Displays a list of models available in the .lti file. <b>Record</b> is the position of the model in the file; <b>Model Type</b> is the type of model in the file; <b>Model Dimension</b> displays the number of inputs, outputs, and states; <b>Model Name</b> is the name of the model; <b>Continuous/Discrete</b> displays whether the file model is continuous or discrete, and if discrete, displays the sampling time; and <b>Notes</b> displays additional information related to the file.

### **Save Control Design Model**

Creates a new .lti file or appends to an existing .lti file, writes the specified number of records to the .lti file, then closes the .lti file and checks for errors. A .lti file stores control design and linear time-invariant models. Each record contains a separate model. To retrieve all records in the file, <Ctrl>-click or <Shift>-click the records in the **Information in file** table.

Parameter	Description
File name	Specifies the name and path where you want to save the file.
Model	Specifies the model you want to save to file.
Append to file	Specifies that the new data appends to the file you specify in <b>File name</b> . If the <b>Append to file</b> checkbox does not contain a checkmark, the step saves the new record on the first record and erases all previous records.
Save	Saves the model to file.
Information in file	Displays a list of models available in the .lti file. <b>Record</b> is the position of the model in the file; <b>Model Type</b> is the type of model in the file; <b>Model Dimension</b> displays the number of inputs, outputs, and states; <b>Model Name</b> is the name of the model; <b>Continuous/Discrete</b> displays whether the file model is continuous or discrete, and if discrete, displays the sampling time; and <b>Notes</b> displays additional information related to the file.

### **Model Analysis**

Use the Model Analysis steps to analyze the model you create.

# **Frequency Domain Analysis**

Analyzes the system model in the frequency domain.

Parameter	Description
Bode Magnitude	Displays the linear or decibel magnitude of the given model plotted against a set of frequencies. <b>Bode</b> <b>Magnitude</b> appears if you select <b>Bode</b> from the <b>Analysis Type</b> pull-down menu on the <b>Configuration</b> tab.
Bode Phase	Displays the phase (in degrees) of the model plotted against a set of frequencies. <b>Bode Phase</b> appears when you select <b>Bode</b> from the <b>Analysis Type</b> pull-down menu on the <b>Configuration</b> tab.
Nyquist Plot	Displays the Nyquist plot of the model. <b>Nyquist Plot</b> appears if you select <b>Nyquist</b> from the <b>Analysis Type</b> pull-down menu on the <b>Configuration</b> tab.
Nichols Plot	Displays the Nichols plot of the model. <b>Nichols Plot</b> appears if you select <b>Nichols</b> from the <b>Analysis Type</b> pull-down menu on the <b>Configuration</b> tab.
Input Data	<ul> <li>Contains the following option:</li> <li>Model—Specifies the model you want to analyze.</li> </ul>
Configuration	<ul> <li>Contains the following options:</li> <li>Analysis Type—Specifies the type of analysis you want to perform. The default is Bode. You can select one of the following options: <ul> <li>Bode—Produces the Bode magnitude and Bode phase plots of the system model.</li> <li>Nyquist—Produces the Nyquist plot of the system model in which the imaginary part of the frequency response is plotted against the real part.</li> <li>Nichols—Creates a Nichols plot of the system model in which the magnitude, in</li> </ul> </li> </ul>

dB, of the frequency response is plotted against the phase.

- Magnitude Scale—Specifies how to scale the magnitude of the frequency response. The Magnitude Scale option appears if you select Bode from the Analysis Type pull-down menu. You can select one of the following:
  - linear
  - dB (default)
- Frequency Unit—Specifies the unit of the frequency. You can select one of the following units:
  - **Hz** (default)
  - rad/s
- **Min Number of Points**—Specifies the minimum number of points used in calculating the frequency response and producing the plots. The default is 500.
- Initial Frequency—Specifies the initial frequency used in calculating the frequency response and producing the plots. If you place a checkmark in the Initial Frequency checkbox, you can change the initial frequency value the step uses to calculate the maximum and minimum frequencies. If you do not place a checkmark in the Initial Frequency checkbox, the step uses an algorithm based on the system dynamics to calculate the maximum and minimum frequencies.
- Final Frequency—Specifies the final frequency used in calculating the frequency response and producing the plots. If you place a checkmark in the Final Frequency checkbox, you can change the final frequency value the step uses to calculate the maximum and minimum frequencies. If you do not place a checkmark in the Final Frequency checkbox, the step uses an

	algorithm based on the system dynamics to calculate the maximum and minimum frequencies.
Frequency Analysis	<ul> <li>Contains the following options:</li> <li>Gain Margin—Displays the smallest gain margin of the system.</li> <li>Phase Crossover Frequency—Displays the crossover frequency that is an integer multiple of –180 degrees and that corresponds to the smallest gain margin.</li> <li>Phase Margin—Displays the smallest phase margin of the system.</li> <li>Gain Crossover Frequency—Displays the 0 dB crossover frequency that corresponds to the smallest phase margin.</li> <li>Bandwidth—Displays the frequency, relative to the DC gain, at which the magnitude of the frequency response of the system falls below the magnitude drop.</li> </ul>

## **Pole-Zero Analysis**

Plots the poles and zeros of a system model on an XY graph that represents a complex plane.

Parameter	Description
Pole-Zero Map	Plots the poles and zeros of a system model on an XY graph that represents a complex plane.
Autoscale Real Axis (X)	Automatically scales the x-axis of the graph to display all the data.
Autoscale Imaginary Axis (Y)	Automatically scales the y-axis of the graph to display all the data.
Input Data	<ul> <li>Contains the following option:</li> <li>Model—Specifies the model you want to analyze.</li> </ul>
Pole-Zero Analysis	<ul> <li>Contains the following options:</li> <li>Stability—Determines if the input system is stable, unstable, or marginally stable.</li> <li>Zeros—Returns all the system zeros.</li> <li>Poles—Returns all the system poles.</li> <li>Natural Freq. (Damping Ratio)—Returns the natural frequencies and damping ratios for each pole in the system.</li> </ul>

# Time Domain Analysis

Creates generic linear simulations and time domain plots for step inputs, impulse inputs, and initial condition responses.

Parameter	Description
Output	Displays the output of the time domain analysis. The graph displays the individual trajectory responses of each state of the system.
	<ul> <li>One of the following graphs appears:</li> <li>Step Response—Plots the output of the system when it is excited by a step input. The initial states of the system are assumed to be zero. This graph appears if you select Step from the Time Analysis pull-down menu on the Configuration tab.</li> </ul>
	• Impulse Response—Plots the output of the system when it is excited by an impulse (delta) or by a pulse (Kronecker) signal. The step uses an impulse for a continuous model and a pulse for a discrete model. This graph appears if you select Impulse from the Time Analysis pull-down menu on the Configuration tab.
	• Initial State Response—Plots the calculated response of the natural or zero-input response of the system to a specific initial state value. This graph appears if the input Model is a state-space model and you select Initial State from the Time Analysis pull-down menu on the Configuration tab.
	<ul> <li>Initial Conditions Response—Plots the calculated response of the natural or zero-input response of the system to a specific initial output condition. This graph appears if you select Initial Conditions from the Time Analysis pull-down menu on the Configuration tab.</li> <li>Simulated Response—Plots the calculated</li> </ul>

	response of the output when the given system is excited with <b>Input Signal</b> . This graph appears if you select <b>Linear Simulation</b> from the <b>Time</b> <b>Analysis</b> pull-down menu on the <b>Configuration</b> tab.
State Trajectories	Displays the state trajectories of the model. The <b>State</b> <b>Trajectories</b> tab appears if you select a state-space model from the <b>Model</b> pull-down menu on the <b>Input</b> <b>Data</b> tab.
Input Data	<ul> <li>Contains the following option:</li> <li>Model—Specifies the model you want to analyze.</li> </ul>
Configuration	<ul> <li>Contains the following options:</li> <li>Time Analysis—Specifies the type of time analysis you want to perform. You can select one of the following options:         <ul> <li>Step—Calculates the output of the system when it is excited by a step input. The initial states of the system are assumed to be zero.</li> <li>Impulse—Calculates the output of the system when it is excited by an impulse (delta) or by a pulse (Kronecker) signal. The step uses an impulse for a continuous model and a pulse for a discrete model.</li> <li>Initial State—Calculates the natural or zero-input response of the system to a specific initial output state. This option appears if you select a state-space model from the Model pull-down menu on the Input Data tab.</li> <li>Initial Conditions—Calculates the natural or zero-input response of the system to a system to a specific initial output state.</li> </ul> </li> </ul>

	<ul> <li>select a transfer function or zero-polegain model from the Model pull-down menu on the Input Data tab.</li> <li>Linear Simulation—Calculates the output when the given system is excited with the inputs.</li> <li>Input Signal—Specifies the input signal the step uses to obtain the response of the system model through linear simulation. This option appears when you select Linear Simulation from the Time Analysis pull-down menu.</li> <li>Initial Time (s)—Specifies the initial time (or t0) for the responses. For linear simulation, the Control Design Assistant obtains the value of Initial Time (s) from the waveform.</li> <li>Interval (s)—Specifies the interval time. If you select Continuous, this time specifies the time between samples for simulation. If you select Discrete or Linear Simulation, this time is the discrete sampling time, which you cannot modify.</li> <li>Final Time (s)—Specifies the final time, in seconds, you want to simulate. If you select Linear Simulation, you cannot use the Final Time (s) control.</li> <li>Initial Conditions—Defines the initial condition of the simulation. This option appears if you select a transfer function or zero-pole-gain model from the Model pull-down menu on the Input Data tab.</li> </ul>
	simulation. This option appears if you select a state-space model from the <b>Model</b> pull-down menu on the <b>Input Data</b> tab.
Parametric Data	<ul> <li>Contains the following options:</li> <li>Parametric Data Step Response—Displays the parametric data of the step response model. The table includes data for Signal, Peak Time (s),</li> </ul>

Peak Value/Overshoot (Mp) [%], Settling Time (s), Steady State Value/Error, and Rise Time (s). Peak Time (s) is the time required for the dynamic system response to reach the peak value of its first overshoot. Overshoot (Mp) [%] is the dynamic system response value that most exceeds unity, expressed as a percent. Settling Time (s) is the time required for the response to reach 1% of its final value. Steady State Value is the final value of the signal after transient responses have decayed. Rise Time (s) is the time required for the dynamic system response to rise from 10% of its final value to 90% of its final value.
<ul> <li>Parametric Data Impulse Response—Displays the parametric data of the impulse response model. The table includes data for Signal, Peak Time (s), Peak Value, and Settling Time (s). Peak Time (s) is the time required for the dynamic system response to reach the peak value of its first overshoot. Settling Time (s) is the time required for the response to reach 1% of its final value.</li> </ul>
<ul> <li>Parametric Data Initial Response—Displays the parametric data of the initial response model. The table includes data for Signal, Peak Time (s), and Peak Value. Peak Time (s) is the time required for the dynamic system response to reach the peak value of its first overshoot.</li> <li>Parametric Data Simulated Response— Displays the parametric data of the simulated response model. The table includes data for Signal, Minimum Value, and Maximum Value.</li> </ul>

### **Model Transformation**

Use the Model Transformation steps to transform the models you create.

## **Connect Models**

Performs different types of linear system interconnections. You can build large system models by connecting smaller system models together.

Parameter	Description
Resulting Model	Displays the mathematical expression that defines the model you create. If you select a state-space model, <b>Resulting Model</b> displays the matrices of the state-space model.
Input Data	Contains the following options:
	<ul> <li>Model 1—Specifies the first system model the Control Design Assistant uses to create the connected model.</li> </ul>
	<ul> <li>Model 2—Specifies the second system model the Control Design Assistant uses to create the connected model.</li> </ul>
	<ul> <li>Sampling time 1 (s)—Displays the sampling time of the model you select in Model 1. This option appears if Model 1 is a discrete-time model.</li> </ul>
	• Sampling time 2 (s)—Displays the sampling time of the model you select in Model 2. This option appears if Model 2 is a discrete-time model.
Configuration	Contains the following option:
	<ul> <li>Operation—Specifies the method the step uses to connect the models you specify on the Input Data tab. You can select one of the following options:</li> </ul>
	<ul> <li>Series—Connects Model 1 and Model 2 such that the series model represents the two input models in series topology. The system models must be either continuous-time models or have the same sampling time if they are discrete- time models.</li> </ul>

<ul> <li>Feedback—Connects Model 1 with Model 2 in feedback configuration. The system models must be either continuous-time models or have the same sampling time if they are discrete- time models.</li> </ul>
<ul> <li>Parallel—Connects Model 1 and Model</li> <li>2 such that the parallel model represents the two input models in parallel topology. The system models must be either continuous-time models or have the same sampling time if they are discrete- time models.</li> </ul>
<ul> <li>Series + Unit Feedback—Applies the series operation to the two input models and closes the system using unity feedback. The resulting model is a system of Model 1 and Model 2 in series in a feedback configuration.</li> </ul>
<ul> <li>Feedback Sign—Specifies the sign of the feedback loop. This option appears when you select an Operation of Feedback.</li> </ul>
<ul> <li>Parallel Sign 1—Specifies the sign of the first input model in the parallel structure. This option appears when you select an Operation of Parallel.</li> </ul>
• <b>Parallel Sign 2</b> —Specifies the sign of the second input model in the parallel structure. This option appears when you select an <b>Operation</b> of <b>Parallel</b> .
<ul> <li>Series Sign—Specifies the sign of the unit feedback loop. This option appears when you select an Operation of Series + Unit Feedback.</li> </ul>

### **Discretize Model**

Converts a continuous-time model to a discrete-time model, converts a discrete-time model to a continuous-time model, or changes the sampling time of a discrete-time system model.

#### **Details**

Parameter	Description
Bode Magnitude	Displays the decibel magnitude of the given model plotted against a set of frequencies.
Bode Phase	Displays the phase (in degrees) of the model plotted against a set of frequencies.
Autoscale magnitude	Automatically scales the y-axis of the magnitude on the display graph.
Autoscale phase	Automatically scales the y-axis of the phase on the display graph.
Input Data	<ul> <li>Model—Specifies the model you want to transform from continuous to discrete or discrete to continuous, or to change the sampling time.</li> </ul>
Configuration	<ul> <li>Input Model—Contains the following options:         <ul> <li>Sampling time in (s)—Displays whether the system model represents a continuous-time system or a discrete-time system. If the model represents a continuous-time system, Sampling time in (s) equals zero. If the model represents a discrete-time system, Sampling time in (s) is greater than zero and equal to the sampling time, in seconds, of the discrete system.</li> <li>Nyquist frequency in—Displays the half maximum frequency in the spectrum of the input system model.</li> </ul> </li> </ul>

you want to perform. If you select a continuous model from the **Model** pull-down menu on the **Input Data** tab, you can select only **Discretize** from this pull-down menu. If you select a discrete model from the **Model** pull-down menu on the **Input Data** tab, you can select one of the following options:

- Discretize (default)
- Re-discretize
- Make Continuous
- **Method**—Specifies the <u>algorithm</u> used to calculate the discrete equivalent of the continuous-time system model or the continuous equivalent of the discrete-time system model. You can select one of the following options:
  - Zero-Order-Hold (default)
  - Tustin (Bilinear)
  - Tustin with Prewarping
  - Forward
  - Backward
  - Z-Transform
  - First-Order-Hold
  - Matched Pole-Zero
- Frequency unit—Specifies the frequency unit of the system model. You can select one of the following options:
  - Hz
  - rad/s (default)
- **Output Model**—Contains the following options:
  - Sampling time out (s)—Defines whether the system model represents a continuous-time system or a discretetime system. If the model represents a continuous-time system, Sampling time out (s) must equal zero. If the model represents a discrete-time system,

<b>Sampling time out (s)</b> must be greater than zero and equal to the sampling time, in seconds, of the discrete system. The default is 100m.
<ul> <li>Nyquist frequency out—Specifies the half maximum frequency in the spectrum of the output system model. The Control Design Assistant correlates the Nyquist frequency out to the Sampling time out (s) by using the following equations: w<sub>n</sub>=2*pi/T (rad/s) or</li> </ul>
f <sub>n</sub> =1/T (Hz) The default is 2*pi/0.1, or 31.4159.

### **Discretize Model Details**

Refer to the LabVIEW Control Design User Manual for information about the equations this step uses for each Method.



Note You must install the LabVIEW Control Design and Simulation Module to access this manual.

Access this manual by navigating to the labview\manuals directory and double-clicking CD\_User\_Manual.pdf.

# **Type Conversion**

Converts a system model into a transfer function, pole-zero, or state-space model.

**Details** 

Parameter	Description
Resulting Model	Displays the mathematical expression that defines the model you create. If you select a state-space model, <b>Resulting Model</b> displays the matrices of the state-space model.
Data Input	Contains the following option:
	<ul> <li>Model—Specifies the system model whose representation you want to change.</li> </ul>
Configuration	Contains the following options:
	<ul> <li>Model Type Conversion—specifies the type of model to which you want to change the representation. Contains the following options:</li> <li>State-Space—Converts a transfer function or zero-pole-gain model to a state-space model. You can specify a full or minimum realization using the Realization type option.</li> </ul>
	<ul> <li>Transfer Function—Converts a state- space or zero-pole-gain model to a transfer function model.</li> </ul>
	<ul> <li>Zero-Pole-Gain—Converts a state- space or transfer function model to a zero-pole-gain model.</li> </ul>
	<ul> <li>Realization type—Specifies the type of realization to use. You can select one of the following types of state-space realization:         <ul> <li>Minimum (default)—Provides a system model after removing zero-pole cancellations and all states that do not affect the output of the system.</li> </ul> </li> </ul>

- Full—Provides a system model without reducing any states or canceling any zero-pole pairs.
- Tolerance—Determines a circle where zero-pole cancellations can occur. If the difference between the location of a pole and a zero is within the Tolerance, this step removes the zero-pole pair if the Realization type is Minimum. This option is available only when you select Minimum from the Realization type pull-down menu. The default is 1E–16.
- Apply canonical form—Transforms the model to a canonical form. You can place a checkmark in the Apply canonical form checkbox if you select State-Space or Transfer Function from the Model Type Conversion pull-down menu. If you select State-Space and place a checkmark in the Apply canonical form checkbox, the Form type option appears. If you select Transfer Function and place a checkmark in the Apply canonical form checkbox, the Transformation type option appears.
- Form type—Specifies the type of canonical form you want to apply to the state-space model you create. The Form type option appears if you select State-Space from the Model Type Conversion pull-down menu and place a checkmark in the Apply canonical form checkbox. You can select one of the following form types:
  - Modal (default)
  - Controllability
  - Controller Companion
  - Observability
  - Observer Companion
- **Transformation type**—Specifies the type of canonical form you want to apply to the transfer

function you create. The **Transformation type** option appears if you select **Transfer Function** from the **Model Type Conversion** pull-down menu and you place a checkmark in the **Apply canonical form** checkbox. You can select one of the following transformation types:

- Highest Term (default)
- Lowest Term

### **Type Conversion Details**

Refer to the LabVIEW Control Design User Manual for more information about converting models.



Note You must install the LabVIEW Control Design and Simulation Module to access this manual.

Access this manual by navigating to the labview\manuals directory and double-clicking CD\_User\_Manual.pdf.