Getting Started with the Control Design Assistant

June 2008, 371245B-01

This help file contains exercises to teach you how to load and save models, perform analysis on systems, and design controllers using the LabVIEW Control Design Assistant. You can learn how to complete the following procedures:

- Launch the LabVIEW Control Design Assistant in the LabVIEW
 SignalExpress environment
- Load models and view model information
- Perform <u>pole-zero</u>, <u>time domain</u>, and <u>frequency domain</u> analysis on systems
- Design and verify controllers
- Save models to file

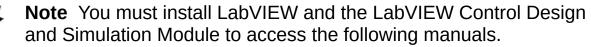
The exercises in this help file take a short amount of time to complete. However, each section of this help file builds upon previous sections so you should complete each exercise in order.

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

Next: Control Design Assistant Related Documentation

Control Design Assistant Related Documentation

The following documents contain information that you might find helpful as you use the LabVIEW Control Design Assistant.



- LabVIEW Control Design User Manual—Use this manual to learn more about the main steps in the control design process and how to develop a control design system using the LabVIEW Control Design and Simulation Module. Access this manual by navigating to the labview\manuals directory and double-clicking CD_User_Manual.pdf.
- Getting Started with LabVIEW—Use this manual as a tutorial to familiarize yourself with the LabVIEW graphical programming environment and the basic LabVIEW features you use to build data acquisition and instrument control applications. Access this manual by navigating to the labview\manuals directory and double-clicking LV_Getting_Started.pdf.
- LabVIEW Fundamentals—Use this manual to learn about LabVIEW programming concepts, techniques, features, VIs, and functions you can use to create test and measurement, data acquisition, instrument control, datalogging, measurement analysis, and report generation applications. Access this manual by navigating to the labview\manuals directory and double-clicking LV_Fundamentals.pdf.

Previous: Getting Started with the Control DesignNext: Before YouAssistantBegin



Before You Begin

System Requirements

The LabVIEW Control Design Assistant requires a minimum of 256 MB of RAM and a Pentium III or later or Celeron 866 MHz or equivalent processor. National Instruments recommends 512 MB of RAM and a Pentium 4/M or equivalent processor.

You can install the Control Design Assistant without installing LabVIEW on the host computer. However, to <u>convert projects created with the</u> <u>Control Design Assistant into LabVIEW block diagrams</u>, you must install and be a properly licensed user of the LabVIEW 8.6 or later Full or Professional Development System.

Refer to <u>Activating Your Software</u> for information about activating this software.

Printing the Help File

You can print this help file if you want to refer to a printed copy while completing the exercises. Complete the following steps to print this help file.

- 1. Click the **Contents** tab of the LabVIEW SignalExpress Help to display the contents.
- 2. Expand the Getting Started book.
- 3. Right-click the Getting Started with the Control Design Assistant book and select Print from the shortcut menu to open the **Print Topics** dialog box.
- 4. Select Print the selected heading and all subtopics.
- 5. Click the **OK** button.

Previous: Control Design Assistant Next: 1. Launching the Control **Related Documentation**

Design Assistant

1. Launching the Control Design Assistant

You can use the LabVIEW Control Design Assistant to develop mathematical models that describe a physical system, analyze the models to learn about their dynamic characteristics, and create controllers to achieve desired dynamic characteristics. In the following exercises, you will load and analyze a model and create a controller that improves the stability of the closed-loop system.

Complete the following steps to launch the Control Design Assistant in the default configuration if you have not done so already.

 Launch the Control Design Assistant by selecting Start»All Programs»National Instruments»LabVIEW Control Design Assistant»LabVIEW Control Design Assistant. The Control Design Assistant launches within the LabVIEW SignalExpress environment.

To launch the Control Design Assistant from LabVIEW, select **Tools**»Control Design and Simulation»Launch Control Design Assistant.

- 2. The LabVIEW SignalExpress environment opens in the default configuration. The Project View is the blank white rectangle on the left side of the LabVIEW SignalExpress environment. The Data View is the green coordinate grid on the black background in the middle of the LabVIEW SignalExpress environment. The context help window is the window on the right side of the LabVIEW SignalExpress environment that displays the How do I get help? and Context Help topics.
- 3. If the graph does not appear in the Data View, select **View»Data View** to display the Data View.
- 4. If the context help window does not appear, select **Help»Context Help** to display the context help window.

Previous: Before You Begin

Next: 2. Loading Models

2. Loading Models

You can use the LabVIEW Control Design Assistant to load linear timeinvariant, or .lti, files into the <u>LabVIEW SignalExpress environment</u>. A .lti file stores control design linear time-invariant models.

Complete the following steps to load a sample model in the LabVIEW SignalExpress environment.

1. Click the **Add Step** button, shown below, to open the <u>Add Step</u> menu.



A step is a configurable function that acquires, generates, analyzes, loads, or stores signals and system data. Steps process input data and produce output data. You can configure the operation of steps by specifying values in the <u>configuration</u> <u>view</u>, or **Step Setup** tab, for each step. You create projects by adding steps to the <u>Project View</u>.

- 2. Select Control Design»Import/Export Model»Load Control Design Model to add a Load Control Design Model step to the Project View. The Step Setup tab for the Load Control Design Model step appears.
 - Tip When you move the cursor over a step in the Add Step menu, a description of that step appears in the help window at the bottom of the Add Step menu.
- 3. In the **Step Setup** tab, click the **Browse** button, shown below, next to the **File name** text box and navigate to the National Instruments\Control Design Assistant\Examples directory.

4. Double-click the cdmodel.lti data file to load the model.

The cdmodel.lti file contains an open-loop model.

5. Click Record **0** in the **Information in file** table.

The Information in file table displays information about the

loaded model. For example, the model in the cdmodel.lti file is a zero-pole-gain SISO model.

- 6. Click the **Data View** tab to display the **Data View**.
- 7. In the Project View, right-click the **Imported: 0** output and select **Rename** from the shortcut menu to give the signal a more intuitive name. Change the name of this output to Model Data.

You can create a <u>probe</u> to obtain information about a signal. In the Project View, right-click the **Model Data** output and select **Probe** from the shortcut menu to create the probe. Notice that the probe displays information about the output, including which step you used to generate the output. Close the probe.

- 8. Click the **Model Data** output of the Load Control Design Model step and drag it from the Project View to the Data View to display a diagram of the model.
- Right-click the Data View and select Model Type»State-Space from the shortcut menu to view the model as a state-space model. Right-click the Data View again and select Model Type»Zeros and Poles to view the model as a zero-pole-gain model.
 - Note Changing the model type in this way does not convert the model type as the <u>Type Conversion</u> step does. Only the display changes to show the model as a different type.
- 10. Select **File**»**Save Project As** and save this project as My Sample Project.seproj to an easily accessible location. A .seproj file is a LabVIEW SignalExpress project, which refers to the LabVIEW SignalExpress environment in which the Control Design Assistant resides.

Previous: 1. Launching the Control Design Assistant Next: 3. Performing a Pole-Zero Analysis

3. Performing a Pole-Zero Analysis

You can use the LabVIEW Control Design Assistant to analyze models to learn about the dynamic characteristics of a system.

Complete the following steps to analyze the poles and zeros of the model.

- Right-click the empty white area of the <u>Project View</u> and select Control Design»Model Analysis»Pole-Zero Analysis from the shortcut menu to add a <u>Pole-Zero Analysis</u> step.
 - Note In the Project View, a blue arrow indicates an output of a step. A red arrow indicates the input of a step. The blue and red arrows are shown below.
 - **I** step output
 - 🕨 step input
- 2. Click the **Input Data** tab on the **Step Setup** tab and verify that **Model Data**, the loaded model, is the input in the **Model** pull-down menu. You can use any outputs from previous steps in the project as inputs for later steps.
- 3. Click the **Pole-Zero Analysis** tab and notice that the **Stability** control indicates the system is stable.
- 4. Remove the checkmarks from the Autoscale Real Axis (X) and Autoscale Imaginary Axis (Y) checkboxes.
- 5. Select **0.5** at the top of the Imaginary axis of the **Pole-Zero Map** graph and enter 1.0 to change the axis on the graph.
- 6. Select –0.5 at the bottom of the Imaginary axis of the Pole-Zero Map graph and enter –1.0 to display the entire unit circle of the model.

Notice that the poles of the model are less than 1 and within the unit circle. The numerical values of the zeros and poles locations appear in the **Zeros** and **Poles** tables.

Previous: 2. Loading Models Next: 4. Performing a Time Domain Analysis

4. Performing a Time Domain Analysis

Complete the following steps to analyze the time domain response of the model.

1. From the LabVIEW SignalExpress menu bar, select Add Step»Control Design»Model Analysis»Time Domain Analysis to add a <u>Time Domain Analysis</u> step.

Notice that the **Step Setup** tab changes from displaying the <u>configuration view</u> of the <u>Pole-Zero Analysis</u> step to displaying the configuration view of the Time Domain Analysis step.

- 2. On the **Configuration** tab of the **Step Setup** tab, verify that **Step** is the type of analysis in the **Time Analysis** pull-down menu.
- 3. Click the **Parametric Data** tab to view the row of parametric data, which indicates system characteristics such as peak time and settling time.
- 4. Click the **Data View** tab to view the **Data View**.
- 5. In the <u>Project View</u>, right-click the **Signal0** output and select **Rename** from the shortcut menu. Change the name of this output to Step Response.
 - **Tip** When you move the cursor over a renamed input or output in the Project View, a dialog bubble appears with both the new name and the original name of the input or output. You also can use this bubble to view the entire name of the input or output if the width of the Project View is too small to view the full name.
- 6. Click the **Step Response** output of the Time Domain Analysis step and drag it from the Project View to the Data View to display the step response on the graph.

Previous: 3. Performing a Pole-Zero Analysis Next: 5. Performing a Frequency Domain Analysis

5. Performing a Frequency Domain Analysis

Complete the following steps to analyze the frequency response of the model.

- In the Project View, right-click the <u>Time Domain Analysis</u> step and select Insert After»Control Design»Model Analysis»Frequency Domain Analysis from the shortcut menu to add a <u>Frequency Domain Analysis</u> step after the Time Domain Analysis step.
- 2. On the **Configuration** tab of the **Step Setup** tab, verify that **Bode** is the type of analysis in the **Analysis Type** pull-down menu.
- 3. Verify that Hz is the unit of frequency selected in the Frequency Unit pull-down menu.
- 4. Click the **Frequency Analysis** tab to view the analysis data of the model.
- 5. Click the **Data View** tab to view the **Data View**.
- 6. In the <u>Project View</u>, right-click the **Magnitude0** output and select **Rename** from the shortcut menu. Change the name of this output to Bode Magnitude.
- 7. Rename the **Phase0** output to Bode Phase.
- 8. Click the **Bode Magnitude** output of the Frequency Domain Analysis step and drag it from the Project View to the Data View to display the Bode Magnitude plot in the Data View. You can repeat this step with the **Bode Phase** output.
 - Note You can remove displays from the Data View by right-clicking the display that you want to remove and selecting **Data View**»**Remove Display** from the shortcut menu.
- Right-click the Bode Magnitude graph and select X Scale»Logarithmic from the shortcut menu to change the X scale to logarithmic format. You can repeat this step with the Bode Phase graph.

<u>Previous: 4. Performing a Time Domain</u> <u>Analysis</u> Next: 6. Designing Controllers

6. Designing Controllers

You can use the LabVIEW Control Design Assistant to create controllers to achieve desired dynamic characteristics in a system.

Complete the following steps to use the <u>PID Synthesis</u> step to design a controller for the model to cause the model to stabilize faster.

- In the Project View, right-click the Frequency Domain Analysis step and select Insert After»Control Design»Controller Design»PID Synthesis from the shortcut menu to add a PID Synthesis step after the Frequency Domain Analysis step.
- 2. On the **Controller Synthesis** tab of the **Step Setup** tab, verify that a checkmark is placed in the **Gain** checkbox.
- 3. Set the scale control to 5.
 - Note You also can enter 5 in the Gain numeric control under the scale.
- 4. Place a checkmark in the **Integral(s)** checkbox and enter 0.02 in the **Integral(s)** numeric control.
- 5. Click the **Model Output** tab to view a diagram of the model.

Notice that **Closed Loop (r-y)** is one of the systems that the PID Synthesis step creates. Configuring the PID Synthesis step creates a controller that changes the system to a closed-loop system. The **Step Response** graph displays the response of the closed-loop system.

- 6. Click the **Data View** tab to view the **Data View**.
- 7. Right-click the diagram of the **Model Data** output in the Data View and select **Add Display**»**Right** from the shortcut menu to add a new display.
- 8. Click the **Closed Loop Model** output of the PID Synthesis step and drag it from the Project View to the added display in the Data View to display information about the model, including its name, type, and structure.

Previous: 5. Performing a Frequency Domain Analysis Next: 7. Verifying Controllers

7. Verifying Controllers

Complete the following steps to use the <u>Time Domain Analysis</u> step to verify the controller for the model.

- 1. Click the Add Step button to open the Add Step menu.
- 2. Select **Control Design»Model Analysis»Time Domain Analysis** to add a Time Domain Analysis step to the <u>Project</u> <u>View</u>.
- 3. On the **Input Data** tab of the **Step Setup** tab, select **Closed Loop Model** from the **Model** pull-down menu to analyze the **Closed Loop Model** input from the <u>PID Synthesis</u> step.
- 4. On the **Configuration** tab, verify that **Step** is the type of analysis in the **Time Analysis** pull-down menu.

The **Step Response** graph displays the same response data as the **Step Response** graph in the **Step Setup** tab of the PID Synthesis step.

5. Click the **Parametric Data** tab to view the parametric data.

The settling time and the rise time are lower for this closed-loop model than for the previous open-loop model in the first <u>Time</u> <u>Domain Analysis</u> step. These lower settling and rise times indicate that the system stabilizes faster.

- 6. Click the **Data View** tab to view the **Data View**.
- 7. In the Project View, right-click the **Signal0** output in the Project View and select **Rename** from the shortcut menu. Change the name of this output to Controlled Response.
- 8. Click the **Controlled Response** output of the Time Domain Analysis step and drag it from the Project View to the **Step Response** graph in the Data View to display the analysis data on the same graph as the **Step Response** output.

Notice that the **Controlled Response** output shows faster settling and rise times than the **Step Response** output.

Previous: 6. Designing Controllers

Next: 8. Saving Models

8. Saving Models

Complete the following steps to save the closed-loop model.

- 1. Click the Add Step button to open the Add Step menu.
- 2. Select Control Design»Import/Export Model»Save Control Design Model to add a <u>Save Control Design Model</u> step to the <u>Project View</u>.
- 3. Click the **Browse** button next to the **File name** text box and navigate to the directory in which you want the LabVIEW Control Design Assistant to save the model.
- 4. Enter closedloopmodel.lti in the **File name** text box and click the **OK** button.
- 5. Select Closed Loop Model from the Model pull-down menu.

The Control Design Assistant saves the closed-loop model as closedloopmodel.lti in this directory each time you run this project. The **Information in file** table displays information about the saved model. You also can use this saved model in LabVIEW.

- 6. Click the **Data View** tab to view the **Data View**.
- 7. Run the project once by clicking the down arrow next to the **Run** button, shown below, on the <u>LabVIEW SignalExpress toolbar</u> and selecting **Run Once** from the shortcut menu.

🔁 Run 🔻

The steps in a project do not execute until you run the project.

- 8. Verify that closedloopmodel.lti is saved in the directory that you specified. You now have a saved, accurate model of the closed-loop system. You can use the Load Model step to load this model to a project.
- 9. Select File»Save Project to save the project.

Previous: 7. VerifyingNext: Extending Control Design AssistantControllersProjects with LabVIEW

Extending Control Design Assistant Projects with LabVIEW

In this help file, you learned how to complete the following procedures:

- Launch the LabVIEW Control Design Assistant in the LabVIEW
 SignalExpress environment
- Load models and view model information
- Perform <u>pole-zero</u>, <u>time domain</u>, and <u>frequency domain</u> analysis on systems
- Design and verify controllers
- <u>Save models</u> to file

Not only can you use LabVIEW SignalExpress projects with the Control Design Assistant, but you also can extend the projects to interact with the LabVIEW environment.

Converting to Block Diagrams

Without prior knowledge about programming in LabVIEW, you can use the Control Design Assistant to analyze models that describe a physical system and synthesize controllers to achieve desired dynamic characteristics. The LabVIEW Control Design and Simulation Module also provides LabVIEW VIs that you can use to complete the entire control design process.

You can use LabVIEW to <u>convert Control Design Assistant projects to</u> <u>LabVIEW block diagrams</u> to continue developing an application. You first can use the interactive tools in the Control Design Assistant to analyze models and design controllers. You then can convert the sequence into a LabVIEW block diagram to enhance the capabilities of the application.

The Control Design Assistant converts projects into block diagrams using the VIs that ship with the Control Design and Simulation Module. The <u>Control Design steps</u> in the <u>Add Step</u> menu do not convert into Express VIs.

Unlike creating a project with the Control Design Assistant, creating a LabVIEW application using the Control Design VIs requires basic knowledge about programming in LabVIEW. Refer to the *LabVIEW Fundamentals* manual and the *Getting Started with LabVIEW* manual for more information about the LabVIEW programming environment. In LabVIEW, access these manuals by clicking the corresponding links in the **Getting Started** window. Refer to the *LabVIEW Help* for more information about LabVIEW VIs and functions. In LabVIEW, access this help file by selecting **Help**»Search the LabVIEW Help.

Sweeping

The <u>Sweep</u> step repeatedly performs measurements with different parameter values. The Control Design Assistant converts the Sweep step into a For Loop in a LabVIEW block diagram. The Control Design steps in the **Add Step** menu can exist within a sweep, but they do not provide parameters that you can use within a sweep.

Using User-Defined Steps

You can <u>build a user-defined VI</u> to use in a <u>Run LabVIEW VI</u> step. The Run LabVIEW VI step runs a VI in the LabVIEW SignalExpress environment. When you import a LabVIEW VI, the Control Design Assistant maps the inputs of the VI as parameters and the outputs of the VI as outputs by default.

Further Examples

The Control Design Assistant provides several other example projects that demonstrate more capabilities of the Control Design Assistant. These projects are located in the National Instruments\Control Design Assistant\Examples directory. You can modify an example project to fit an application, or you can copy and paste from one or more examples into a project that you create.

You can find the solution project, My Sample Project Solution.seproj, to the exercises in this help file, as well as an example of the closed-loop model you created, closedloopmodel.lti, in the National Instruments\Control Design Assistant\Examples directory.

Previous: 8. Saving Models