Simplorer Getting Started Guide

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 18.2
July 2017

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2008 companies.

Table of Contents

1 - Introduction

This *Getting Started* guide is written for Simplorer beginners as well as experienced users who are using Simplorer for the first time.

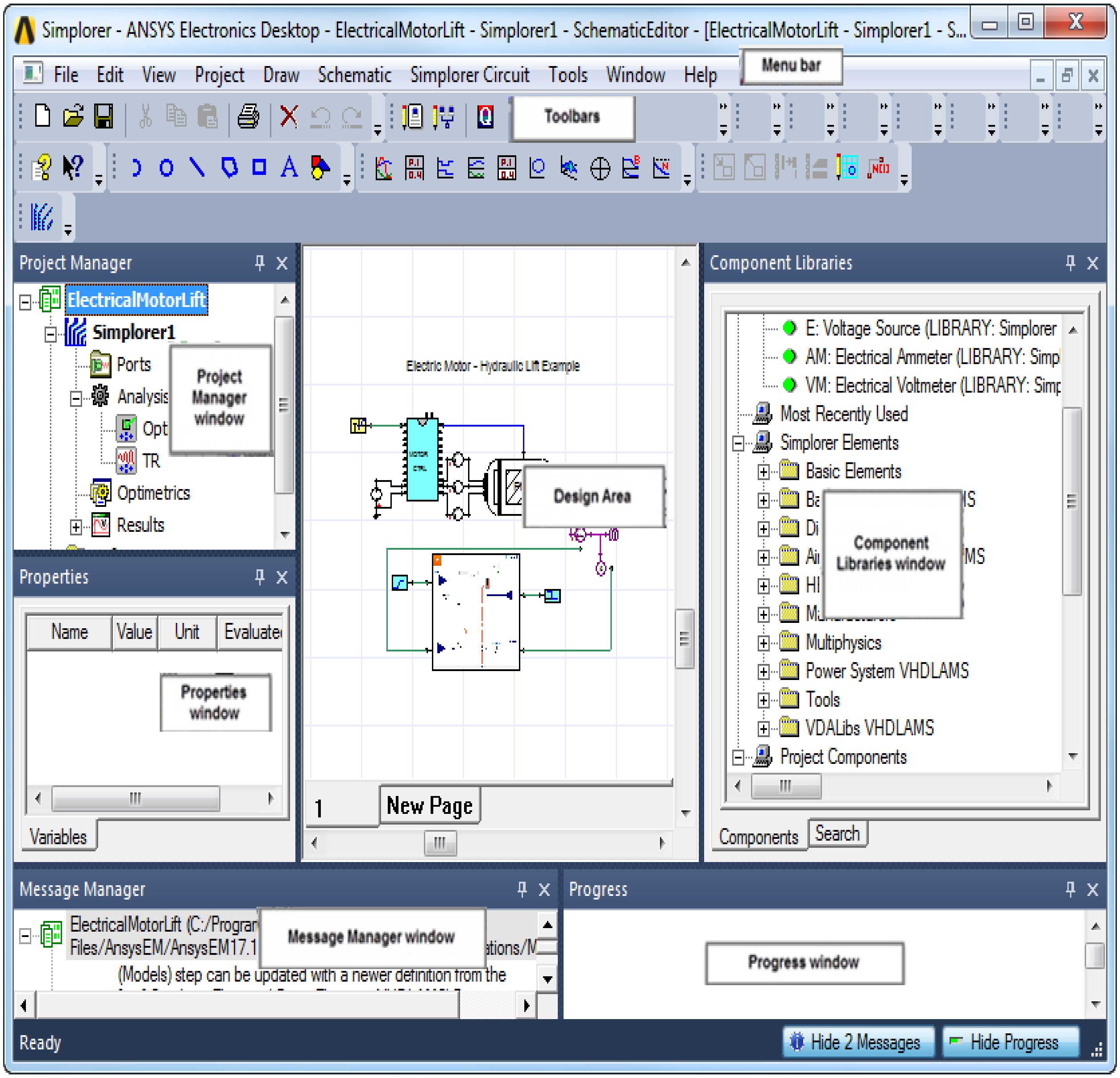
Simplorer is a simulation package for multi-domain designs that are commonly found in automotive, aerospace, power electronics, and electric drive systems.

Simplorer provides an approach for the virtual prototyping of large-scale systems by letting you to develop a design that combines predefined basic and industry-specific components with user-defined models. You can create models in common programming languages or standard modeling languages such as VHDL-AMS. The basic and industry-specific model libraries available for Simplorer provide ready-to-use parameterized components, making Simplorer accurate, and flexible.

- Chapter 2 of this guide shows you how to create and save a new Simplorer project.
- Chapter 3 leads you step-by-step through creating, solving, and analyzing the results of a Three-Phase Rectifier.
- Chapter 4 modifies the example of Chapter 3 by replacing the resistive/inductive load with a real machine model. The example is then expanded to include a control circuit modeled first using discrete components, then using state graph components.
- In Chapter 5 the state graph controller is replaced with a PI (proportional-integral) controller implemented using block components.
- Chapter 6 substitutes the use of VHDL-AMS opponents for modeling the DC motor.
- Chapter 7 explores several different methods for modeling a PWM Controller to demonstrate Simplorer's versatile modeling capabilities.
- Chapter 8 leads you through the process of importing a legacy Simplorer 7 Three-Phase Rectifier schematic into Simplorer Release 16.2, saving it, migrating it into Simplorer 2017.2, and then solving and analyzing the translated model.

Overview of the Simplorer Interface

Below is an overview of the major components of the Simplorer interface.



Project Manager window and Project Tree	The Project Manager window shows all the components, models, and other elements of each design in the project. Each project has its own expandable *Project Tree*. Many operations on the design elements can be performed directly from the Project Manager window.
Message Manager window	Displays error, informational, and warning messages for the active project.
Progress window	Displays solution progress information.

Properties window	Displays the attributes of a selected object in the active model, such as the object's name, electrical or other associated physical quantities, orientation, and color. Also displays information about a selected command that has been carried out. For example, if a circle was drawn, its command information would include the command's name, the circle's center position coordinates, and the size of its radius.
Design area window	Displays one or more editor windows such as the Schematic Editor, model editors, and symbol editor. It also displays various report windows.
Component Libraries window	Displays, on the Components tab, the component categories, including Favorites, Most Recently Used, Simplorer Elements, and Project Components. You can pin the window to make it remain visible or make it visible only when it is being used. The following elements are defined as favorites by default: • R (resistor) • L (inductor) • C (capacitor) • D (diode) • E (voltage source) • AM (ammeter) • VM (voltmeter) The Project Components section lists the elements that are active in your projects. If you have created any personal libraries, a Personal Libraries section displays them. The Component Manager window also provides a search feature on the Search tab.
Menu bar	Provides various menus that enable you to perform Simplorer tasks, such as managing project files, designs, and libraries; customizing desktop components; drawing objects; and setting and modifying project parameters and options.
Toolbars	Provides buttons that act as shortcuts for executing various commands.
Status bar	Shows current actions and provides instructions.

2 - Creating a New Project

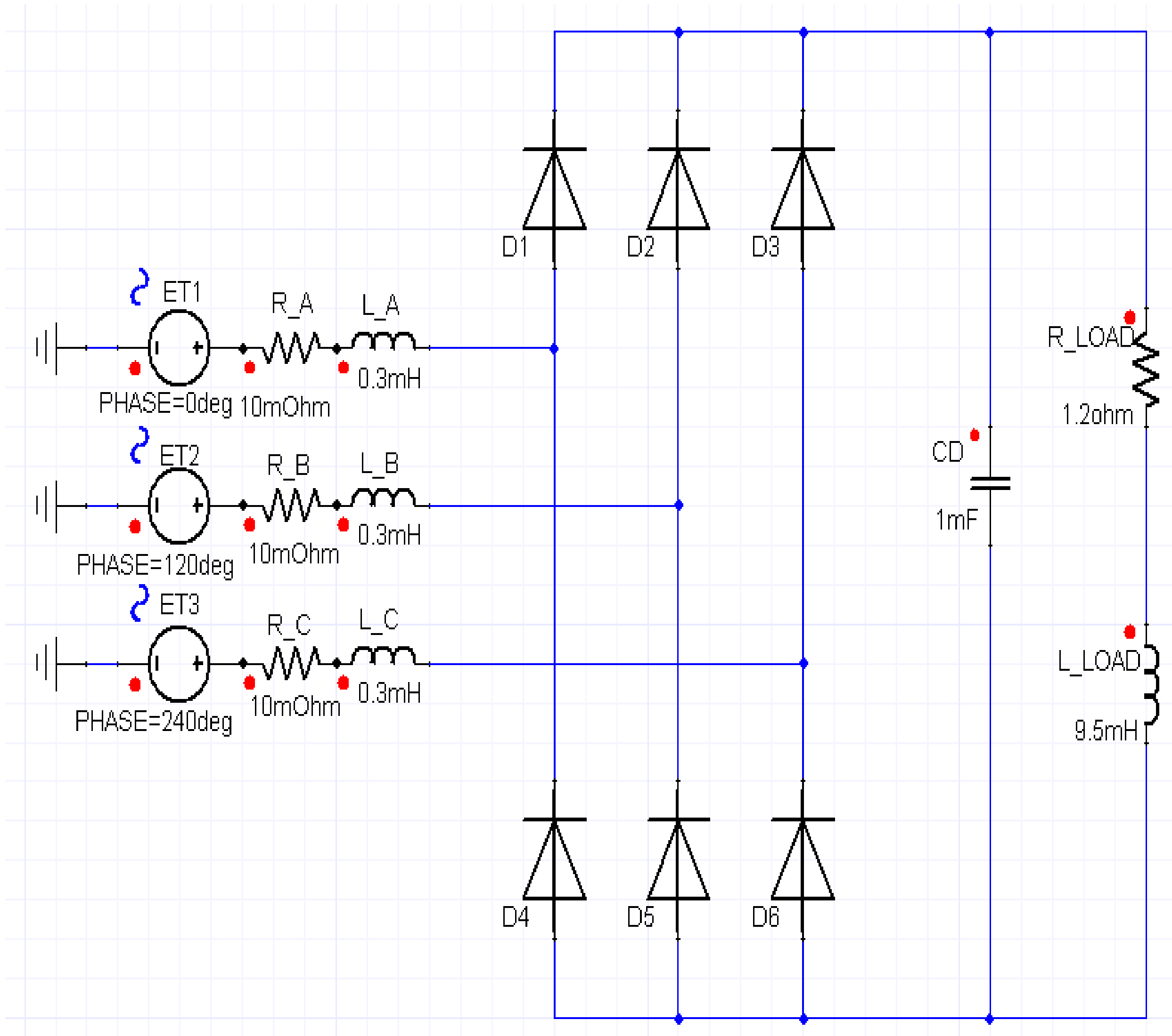
This guide assumes that you have installed Simplorer.

In this chapter you will complete the following tasks:

- Create and save a new project.
- Add and rename a Simplorer design in the project.

About the Three-Phase Rectifier

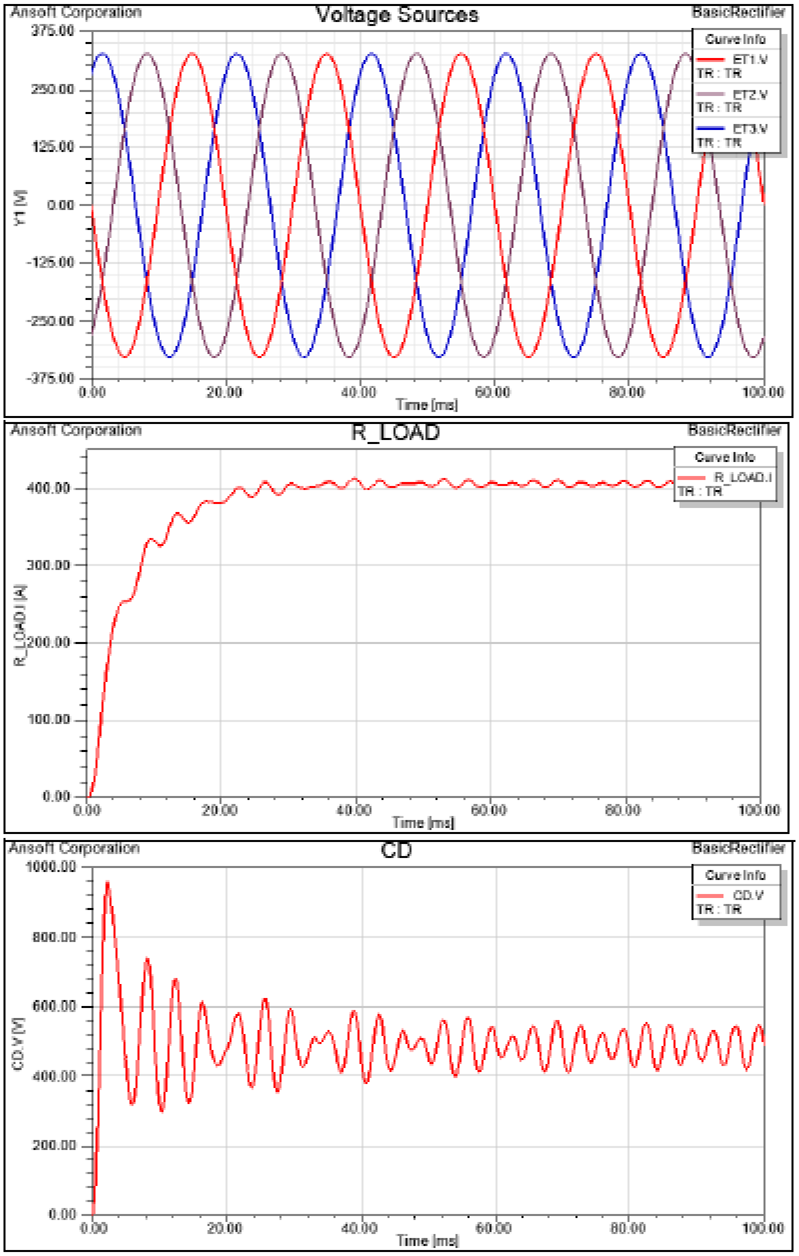
The following example describes a three-phase power supply and a rectifier bridge with resistive/inductive load. For the input signals, time-controlled voltage sources will be used. The diodes, the capacitor, and the resistor are ideal components; the diodes are determined by an exponential function (their characteristic).



Expected Results

After the simulation is run, the results are displayed in Report windows. For example, the first simulation variation demonstrates these results:

- Voltages of the sources ET1.V, ET2.V, and ET3.V
- Voltage of the smoothing capacitor CD.V
- Current of the load resistor R_LOAD.I



Using Simplorer to Create and Improve the Design

As you step through this *Getting Started* guide, you will be introduced to several key concepts:

- *There are numerous ways to perform most tasks.*

  For example, several methods are presented for selecting and for assigning design parameter values.

- *There is no required sequence of events when creating a design.* — A convenient method for creating the three-phase rectifier will be demonstrated, but the design setup steps can be completed in any logical order.
- *You can quickly modify design properties at any time.*

  For example, you can draw a box freehand, then specify its exact dimensions in the **Properties** window.

- *You can easily manage your design in the project tree.*

  The branches of the project tree in the **Project Manager** window provide access to setup dialogs, where you can modify design properties.

- *You can modify the model view at any time.*

  You will learn shortcut keys like **Ctrl+D**, which fits the model in the view window.

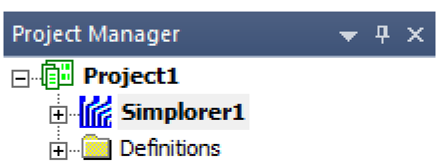
- *You can save time by parameterizing design properties.*

  This enables you to quickly modify design properties and generate new results.

- *You can use Simplorer's extensive post-processing features to evaluate solution results*.

Creating the New Project

The first step in using Simplorer to solve a problem is to create a project in which to save all of the data associated with the problem. A project is a collection of one or more designs saved in a single *.aedt file. By default, opening Simplorer creates a new project named Project*n* and inserts a new design named Simplorer*n*, where *n* is the order in which each was added to the current session.



Add the New Project

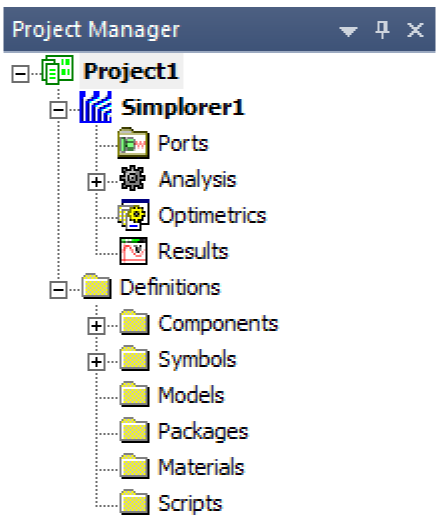
To add a new Simplorer project:

- Click **File>New**.

A new project named Project*n* containing a new design named Simplorer*n* is added in the project tree in the **Project Manager** window.

Hint	• If the Project Manager does not appear, click **View>Project Manager**. • If a new design does not appear automatically, click **Tools>Options> General Options**. Under **Project Options**, select **Insert a design of type**. • To automatically expand the project tree when an item is added to the project, click **Tools>Options> General Options**. Under **Project Options**, select **Expand Project Tree on Insert**.

The new project contains a file structure that organizes design elements such as: Ports, Analysis, Optimetrics, and Results. Project Definitions such as Components, Symbols, Models, Packages, Materials, and Scripts are also listed.



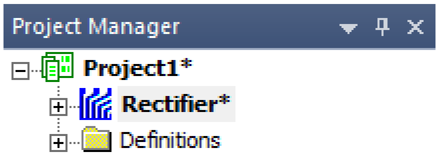
Rename the Design

You can rename the default Simplorer design name as follows:

1. Right-click Simplorer*n* in the project tree, and choose **Rename** on the shortcut menu. You can also click the name to select it, then press **F2**.

   This enables the text cursor for the design name.

2. Type **Rectifier** (or some other name of your choosing), and then press **Enter** to complete the change.



Add Design Notes (Optional)

You can optionally include notes about your design, such as a description of the design being modeled, with the project. These notes can be a useful tool for keeping a running log of your designs.

To add notes to your design:

1. Click **Simplorer Circuit>Edit Notes**.

   The **Design Notes** window appears.

2. Click in the window and type your notes.
3. Click **OK** to save the notes in the project tree under the current design.

Note	To edit existing design notes, double-click **Notes** in the project tree to open the **Design Notes** window in which you can edit the notes.

Save the New Project

To save the new project:

1. Click **File>Save**.

   The **Save As** dialog box appears.

2. Use the file browser to find the directory where you want to save the file.
3. Type the desired file name in the **File name** text box.
4. In the **Save as type** list, ensure that **Simplorer Project Files (*.aedt)** is chosen. Simplorer project files are given an .aedt extension by default.
5. Click **Save** to save the project to the specified location.

Note	For further information on any Simplorer topic, schematic editor commands or windows, you can view Simplorer's context-sensitive online help in one of the following ways:

- Click the **Help** button in a pop-up window.
- Press **Shift+F1**. The cursor changes to **?**. Click on the item for which you need help.
- Press **F1** to open the **Help** window. If you have a dialog box open, the **Help** window opens to a page that describes that dialog box.
- Use the commands in the **Help** menu.

Now you are ready to create the Rectifier model.

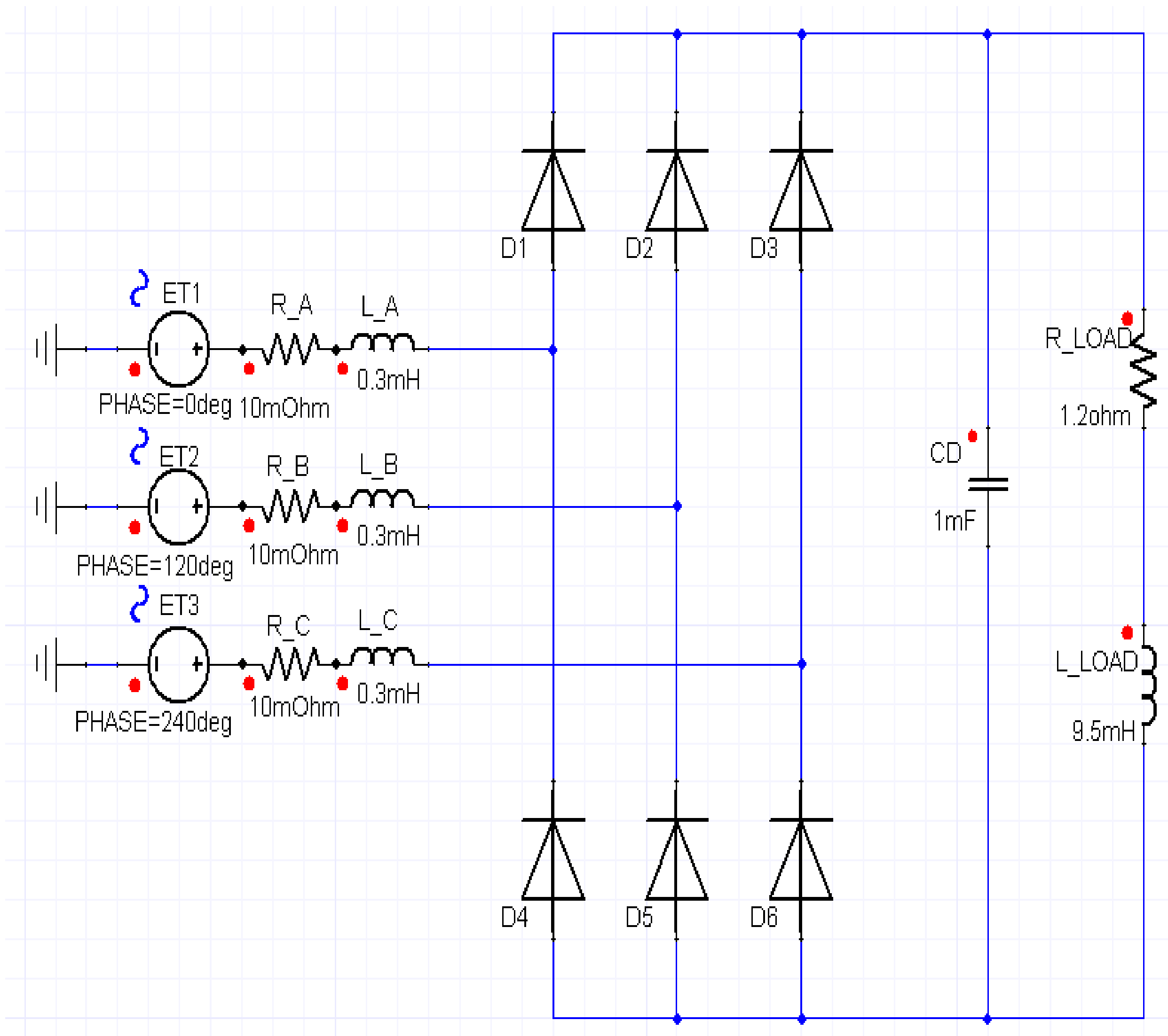
3 - Create the Rectifier Model

In this chapter you will complete the following tasks:

- Basic Simplorer functions
- Choosing Simplorer components from a library
- Placing and arranging components
- Connecting components on the sheet
- Controlling the display of component properties
- Modeling with electric circuit components
- Modeling time controlled sources
- Setting up and running an analysis (simulation)
- Using Reports for displaying simulation results

Create the Three-Phase Rectifier Schematic

The following example contains a three-phase power supply and a rectifier bridge with resistive/inductive load. For the input signals, time controlled voltage sources will be used. The diodes, the capacitor, and the resistor are ideal components; the output characteristics of the diodes are determined by an exponential function.

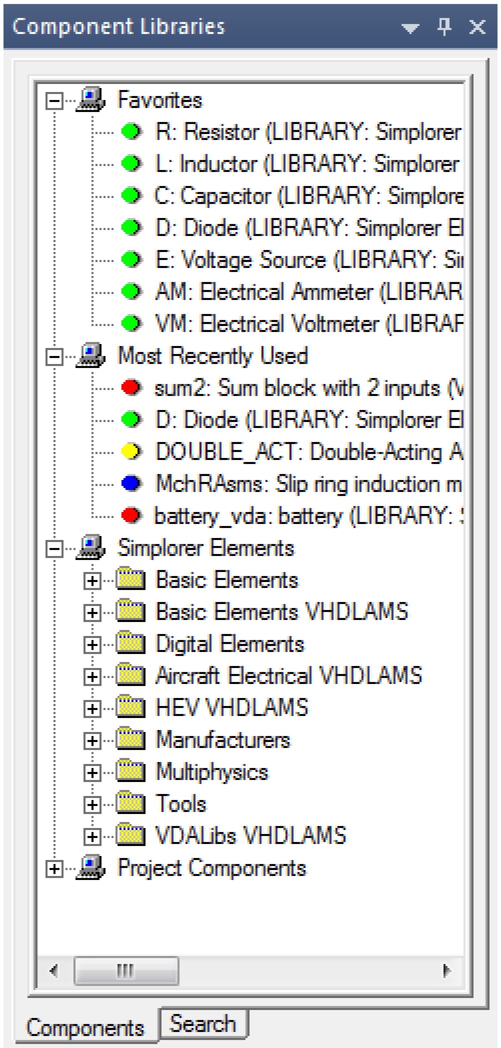


Note	If you make a mistake, click Rectifier in the project tree, and then click **Undo** on the **Edit** menu to undo design operations. Simplorer lets you undo every command performed since the last save.

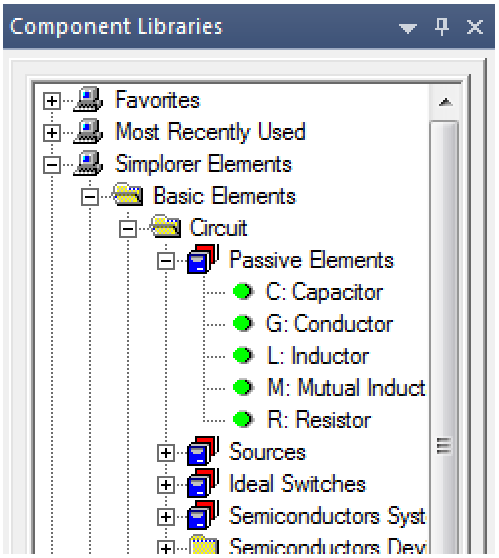
Choosing, Placing, and Arranging Components on the Schematic Sheet

First, you need to locate and choose the components you want to use in the simulation model, and then place and arrange the components on the schematic sheet.

1. In the **Component Libraries window**, select the **R: Resistor** element. The **R: Resistor** element is listed under **Favorites** by default.



Note	An alternative path is to click the “+” symbol to expand the **Simplorer Elements** tree, and then continue to click the “+” symbols next to the **Basic Elements** folder, the **Circuit** folder, and the **Passive Elements**.The **R: Resistor** element is listed under **Passive Elements**.

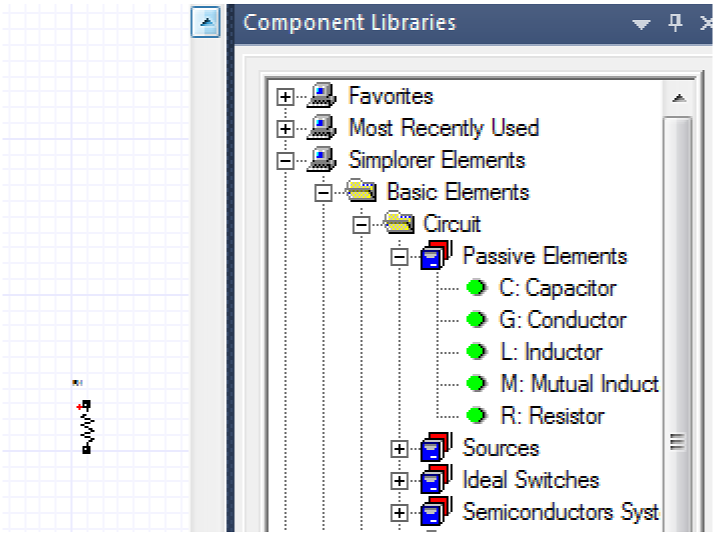


Note	You can also click the toggle triangle at the upper right of the Component Libraries window to undock the window and move it to another location. You can also expand the size of the Component Libraries window.

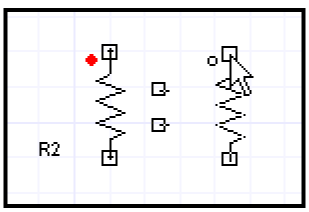
2. To place the resistor onto the sheet, select **R: Resistor**, hold the mouse button down, and drag the selection onto the sheet.

   As you drag a library component over the **Schematic Editor** window, the symbol for that component is displayed. At this point, you can press the **R** key to rotate the component before placing it on the sheet. The component rotates 90 degrees counter-clockwise each time you press the **R** key.

3. Release the mouse button to place the component on the sheet.

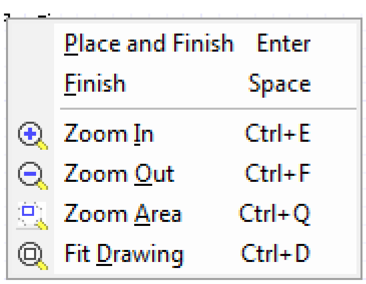


After placing one resistor, notice that the cursor retains a selected resistor symbol.



This feature permits you to place several instances of a component by clicking without having to reselect the component from the library.

4. Right-click to display the short-cut menu for a selected component.



The **Finish** command exits the “place” mode without placing a component and frees the cursor for selecting additional library components or other actions.

The **Place and Finish** command places an additional component before exiting “place” mode.

Note	You can also press the **Esc** key to exit “place” mode without placing a component.

5. To continue the design, repeat the process outlined above and place the following components from the **Basic Elements** library onto the **Schematic Editor** sheet.

Module	**Group**	**Component**	**Quantity**
Circuit	Passive Elements	R: Resistor	4
		L: Inductor	4
		C: Capacitor	1
	Sources	E: Voltage Source	3
	Semiconductors System Level	D: Diode	6

6. When you have placed the elements, use the *Three-Phase Rectifier* schematic at the start of this chapter as a guide to arrange the components.

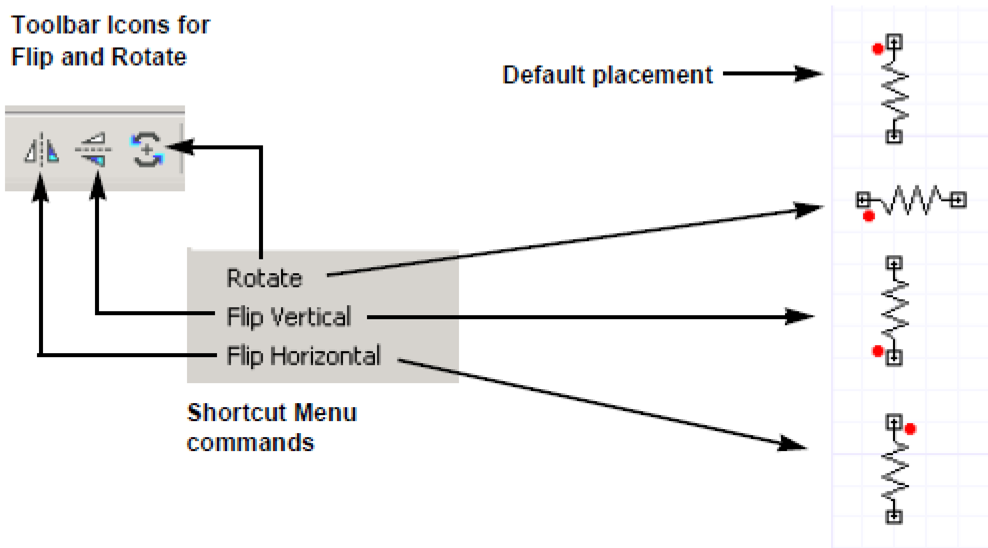
Note	Please note the “counting” direction when arranging components. This direction is marked by the red “dot” or the plus sign on the symbol of electrical components.

Once components are placed, you can select, move, copy, delete, rotate, or flip them. You can select elements:

- By clicking on them individually.
- By holding the mouse button down to draw a selection rectangle around multiple instances.
- By holding down the **Ctrl** key and clicking on multiple instances.

Selected instances are highlighted.

Commands for flipping and rotating selected elements appear in the toolbar icons and also in the schematic editor shortcut menu.



Hint	Save your project frequently: Click **File>Save**.

7. To align specific components, click and hold the mouse button, and drag the cursor to specify a selection area. Selected elements are highlighted.

   Use **Draw>Align Horizontal** to horizontally align the components, and **Draw>Align Vertical** to vertically align components.

8. A ground node is necessary for each separate circuit on a sheet. To place a ground node, click **Draw>Ground** (or click the ground symbol on the toolbar).

Hint	• You can zoom in and out by using the **View** menu commands, the Zoom icon functions in the toolbar, or **Ctrl+E** (Zoom In) and **Ctrl+F** (Zoom Out). **Ctrl+D** (Fit Drawing) scales the drawing to include all of its components in the current window. • To move the sheet and its contents around within the window, while pressing **Shift**, click and hold the mouse button (the cursor changes to a “hand” icon) and drag the sheet to the desired position.

   The ground symbol will “stick to” the mouse pointer.

9. Position the terminal of the ground node over the terminal of the device you are grounding and click to place the ground node and connect it to the device. You may rotate the ground node to the desired position if necessary.

All of the components required for the Three-Phase Rectifier simulation model should now be on the sheet and placed in the appropriate positions. In the next section, you will connect the components.

Connecting the Components

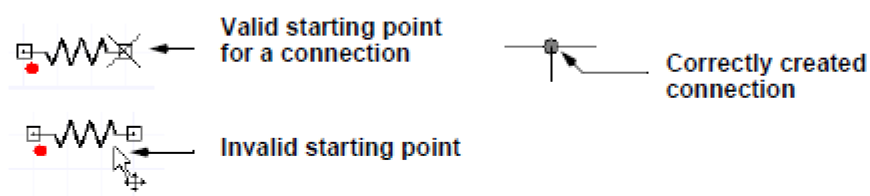
When you have arranged all of the components, you are now ready to connect them as required for this example.

1. To connect components, activate wire drawing mode by choosing **Draw>Wire**. The cursor changes to crossed wires.

Note	You can also activate wire drawing mode by pressing **Ctrl+W** or simply by placing your cursor over an element pin (the cursor automatically changes to cross wires) and clicking on the pin.

2. Connect the components as required for the example circuit.
   a. Place the cursor on the element pins and set the beginning, the corners, and the end of a wire by clicking and dragging.
   b. Press **Esc** to exit wire drawing mode. The cursor changes back to a pointer.

.



Defining Component Properties

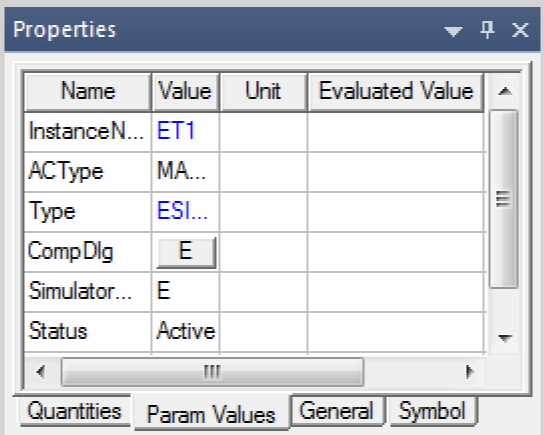
The components that you have placed and connected still have their default parameter values, as defined in the **Basic Elements** model library. You will now assign values to these components to match the schematic at the beginning of this chapter by performing the following steps:

1. First, define the parameters of the three voltage sources. All sources are time-controlled sine function generators with a phase shift of 120 degrees with respect to each other.
   a. Double-click the first Voltage Source symbol to open the component's **Parameters** dialog box.
   b. Change the **Name** parameter to **ET1**.
   c. Select the **Time Controlled** radio button and ensure that **Sine** is selected in the drop-down list. Keep the **Phase** value as-is at ***0*** (zero) degrees. Keep all other values at their default values.
   d. Click **OK** to apply the changes.

2. Double-click the second voltage source symbol to open its **Parameters** dialog box.
   a. Change the **Name** parameter to **ET2**.
   b. Select the **Time Controlled** radio button.
   c. Set the **Phase** value to **120** degrees. Keep all other values at their default values.
   d. Click **OK** to apply the changes.
3. Double-click the third voltage source symbol to open its **Parameters** dialog box.
   a. Change the **Name** parameter to **ET3**.
   b. Select the **Time Controlled** radio button.
   c. Set the Phase value to **240** degrees. Keep all other values at their default values.
   d. Click **OK** to apply the changes.

Note

- Properties of the currently selected element are also displayed in the **Properties** window similar to the one shown below.



- Many component properties can be modified in this window. Refer to the Simplorer help for details.
- Clicking the **Info** button opens detailed online help for the current component.

4. Next, define the parameters of the phase resistors.
   a. Double-click the top-most phase resistor symbol to open the resistor's **Parameters** dialog box.
   b. Change the **Name** to **R_A**.
   c. Change the **Resistance** from **1000 ohm** to **10 mOhm**.
   d. Click **OK** to apply the changes.
   e. **Repeat** steps **a** through **d** for the other two phase resistors, naming them **R_B** and **R_C**, respectively.
5. Define the parameters for the phase inductors.

a. Double-click the top-most phase inductor symbol to open its **Parameters** dialog box.
b. Change the **Name** to **L_A**.
c. Change the **Inductance** from **0.001 H** to **0.3 mH**.
d. Check **Initial Value** and set the value to **0** (zero).
e. Click **OK** to apply the changes.
f. **Repeat** steps **a** through **d** for inductors **L_B** and **L_C**.

6. Define the parameters of the diodes. In this example, all diodes are static models using an exponential function as their characteristic.

Note	For most applications static semiconductor models supply sufficient simulation data. If switching, losses, thermal analysis, and other properties are targets of your simulation, you need dynamic elements. However, using many dynamic elements in a simulation model can increase the simulation time.

a. Double-click the upper left diode symbol to open its **Parameters** dialog box.
b. Change the **Name** to **D1**.
c. Change the **Type** by clicking the **Type** radio button, then choosing **Exponential Function** from the drop-down menu.
d. Keep all other values as they are and click **OK** to apply the changes.
e. **Repeat** steps **a** through **d** for the remaining diodes **D2** through **D6**.

7. Define the parameters of the smoothing capacitor.
   a. Double-click the capacitor symbol to open its **Parameters** dialog box.
   b. Change the **Name** to **CD** and change the **Capacitance** from **1e-006 F** to **1 mF**.
   c. Click **OK** to apply the changes.
8. Define the parameters of the load resistor.
   a. Double-click the resistor symbol to open its **Parameters** dialog box.
   b. Change the **Name** to **R_LOAD**.
   c. change the **Resistance** from **1000 ohm** to **1.2 ohm**.
   d. Click **OK** to apply the changes.
9. Define the parameters of the load inductor.
   a. Double-click the inductor symbol to open the **Parameters** dialog box.
   b. Change the **Name** to **L_LOAD**.
   c. Change the **Inductance** from **0.001 H** to **9.5 mH**.
   d. Check **Initial Value** and set the value to **0** (zero).
   e. Click **OK** to apply the changes.

All components of the simulation model should now have their correct values. The table below lists the components of the simulation model and their parameter values.

Name	Type	Quantities
ET1	**Sine** (Time-controlled)	Amplitude [V]=**326**; Frequency [Hz]=**50**; Initial Delay [s]=**0**; Phase [deg]=**0**; Offset [V]=**0**
ET2	**Sine** (Time-controlled)	Amplitude [V]=**326**; Frequency [Hz]=**50**; Initial Delay [s]=**0**; Phase [deg]=**120**; Offset [V]=**0**
ET3	**Sine** (Time-controlled)	Amplitude [V]=**326**; Frequency [Hz]=**50**; Initial Delay [s =**0**; Phase [deg]=**240**; Offset [V]=**0**
R_A **R_B** **R_C**	**R** (Linear)	Resistance [mOhm]=**10**
L_A **L_B** **L_C**	**L** (Linear)	Inductance [mH]=**0.3**; Initial Current [A]=**0**
D1…D6	**DEXP** (Exponential Func- tion)	Saturation Current [pA]=**1**; Thermal Voltage[mV]=**35**; Reverse Resistance[kOhm]=**100**
CD	**C** (Linear)	Capacitance [mF]=**1**; Initial Voltage [V]=**0**
R_ LOAD	**R** (Linear)	Resistance [Ohm]=1.2
L_ LOAD	**L** (Linear)	Inductance [mH]=**9.5**; Initial Current [A]=**0**

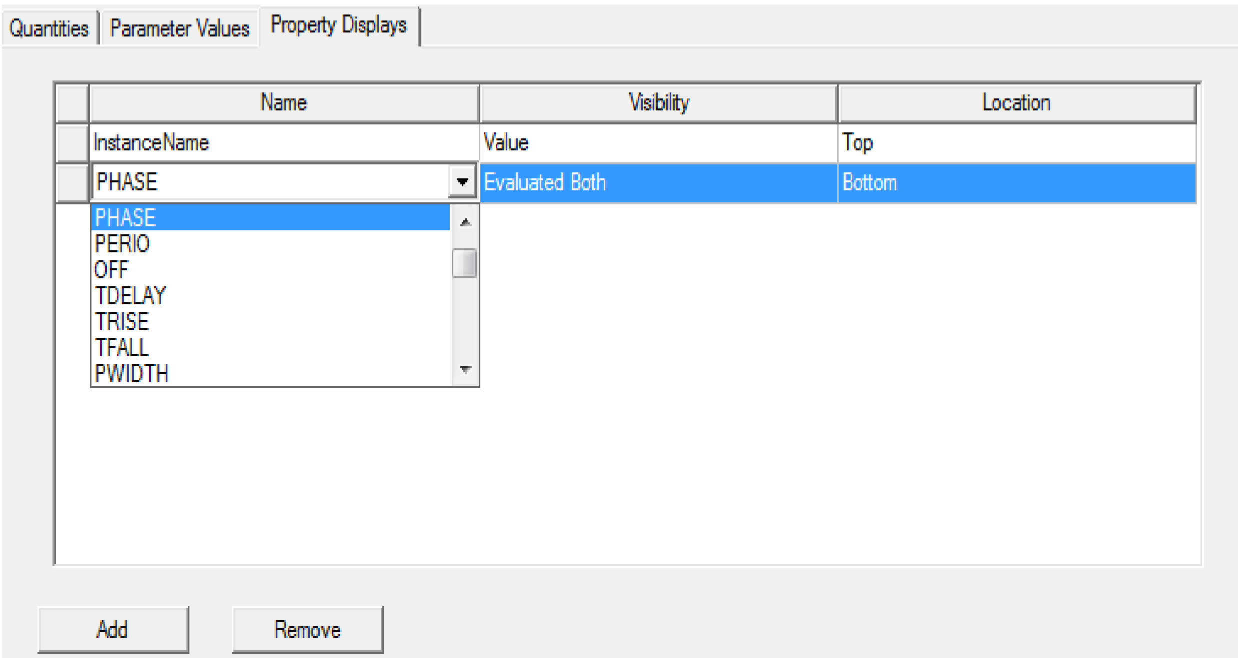
Next you will define the component properties that will be displayed on the schematic sheet.

Property Displays for Components

While most component properties are accessible in the dockable **Properties***window*, you control the ***display*** of component properties via the **Property Displays** tab in the **Properties***dialog boxes.*

1. For example, select Voltage Source 1 (**ET1**) and open its **Properties** dialog box. (Right-click on the component and select **Properties** from the context menu.)
2. On the **Property Displays** tab, click the **Add** button.

   This adds a line with three fields: **Name**, **Visibility**, and **Location**. Each of these fields contains drop-down selection menus.



- For the **Name** field, select the **Label ID** of interest.

The drop-down menu for the **Name** field includes all available property names for the selected instance. For each kind of component in the schematic, you will select another property value. For now, start with the name.

- The **Visibility** field allows you to specify what to display about the selected Label ID on the schematic. This can be **None**, the property **Name**, the property's **Value**, the **Evaluated Value**, **Both** (the name and value) or **Evaluated Both**. Select **Value** from the menu.
- The **Location** field lets you select *initial* locations relative to the *default* orientation of the component instance.

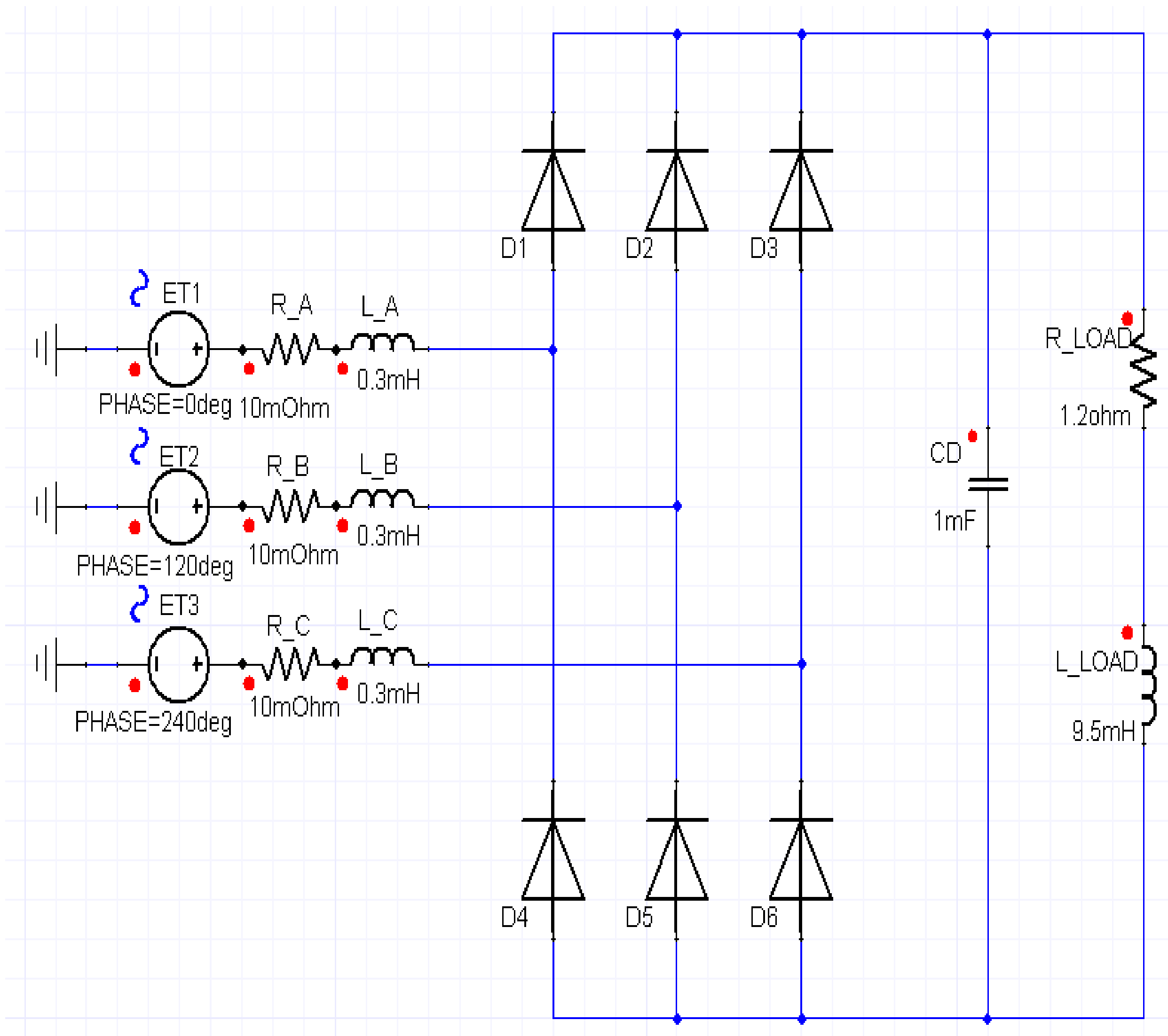
3. For this example, **OK** the dialog box, and locate and click to select the **ET1** label in its initial placement.

   Selecting the label highlights it.

4. Drag the label to a position above the component instance.
5. With the label highlighted, you can click the **Rotate** icon on the toolbar, or use **Draw>Rotate** to rotate the label as needed.
6. Open the **Properties** dialog again, and select the **Property Displays** tab again.

   Note that the **Location** field for the **Name** property now shows **Custom**.
7. Now click the **Add** button again.
8. This time, use the drop-down menu in the **Name** field to locate and select **PHASE**.
9. For **Visibility**, select **Evaluated Both** and **OK** the dialog.

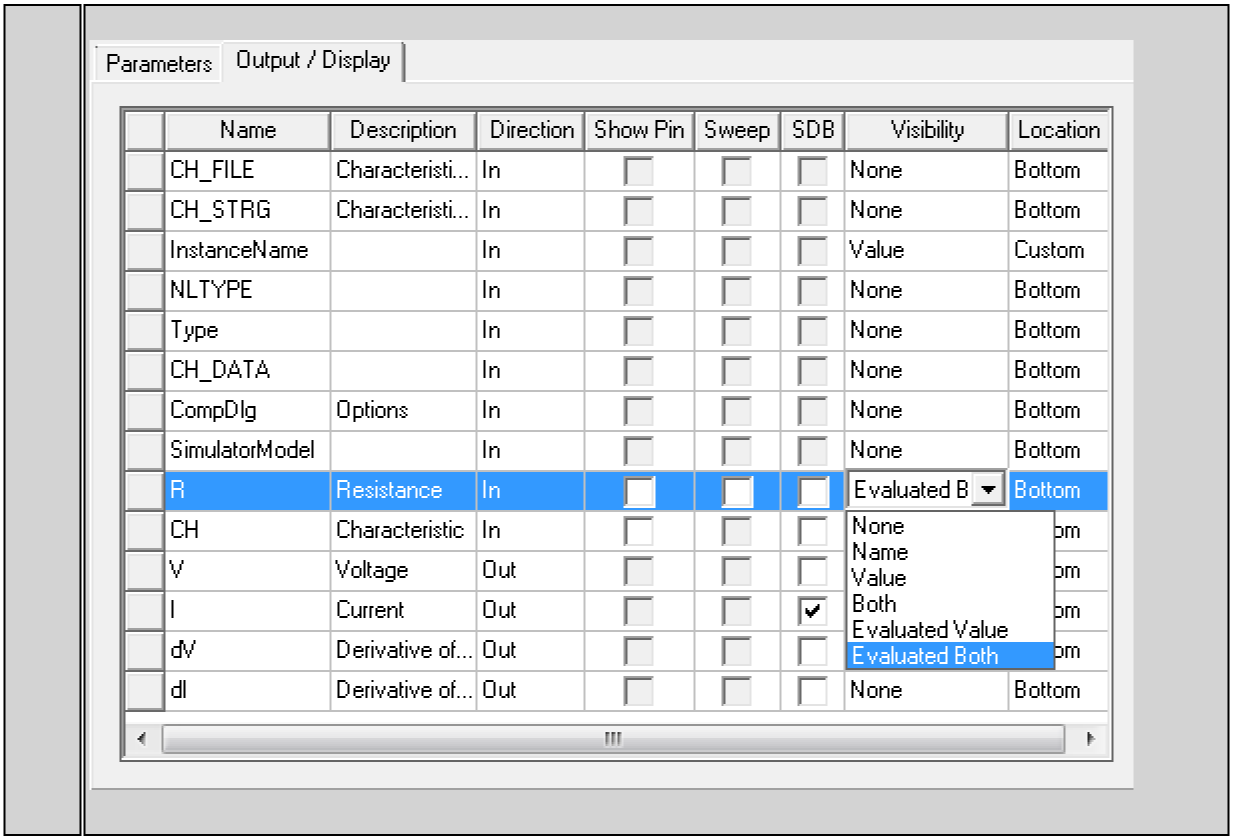
   The label **PHASE=0** should now be visible on the schematic.
10. Drag the **PHASE=0** label to a position to the left of the component as shown in the figure.
11. Use the foregoing techniques to activate labels for the rest of the components to match the schematic figure.



- For the Resistors, add “Name” and “R” name.
- For the Inductors, add “Name” and the “L” name.
- For the Diodes, add only “Name” to the display.
- For the Capacitor, add “Name” and “C”.

12. Save your work before continuing to the next step.

Note	You can also set the visibility and initial location for properties you wish to display by double-clicking on a component to open its Parameters dialog box. The Output/Display tab has drop-down menus for setting the **Visibility** and **Location** for each property.



Specifying Simulation Outputs

During a simulation many types of data are generated based on outputs that you specify. These can include quantities such as voltage, current, frequency, phase angle, torque, and displacement.

Using the outputs you define, you can also create reports of simulation results, or plot output quantities specified by probes that you place in the schematic.

To define simulation outputs for the example model:

1. Click **Simplorer Circuit>Output Dialog**. This displays the **Output** dialog. The **Add/Remove** list shows all of the schematic elements. Checkboxes control which quantities are visible in the selection window.
2. Type **ET** in the **Find** field.

   This moves the displayed list to the **ET** elements, and opens a list of check boxes that show the kinds of outputs you can select.

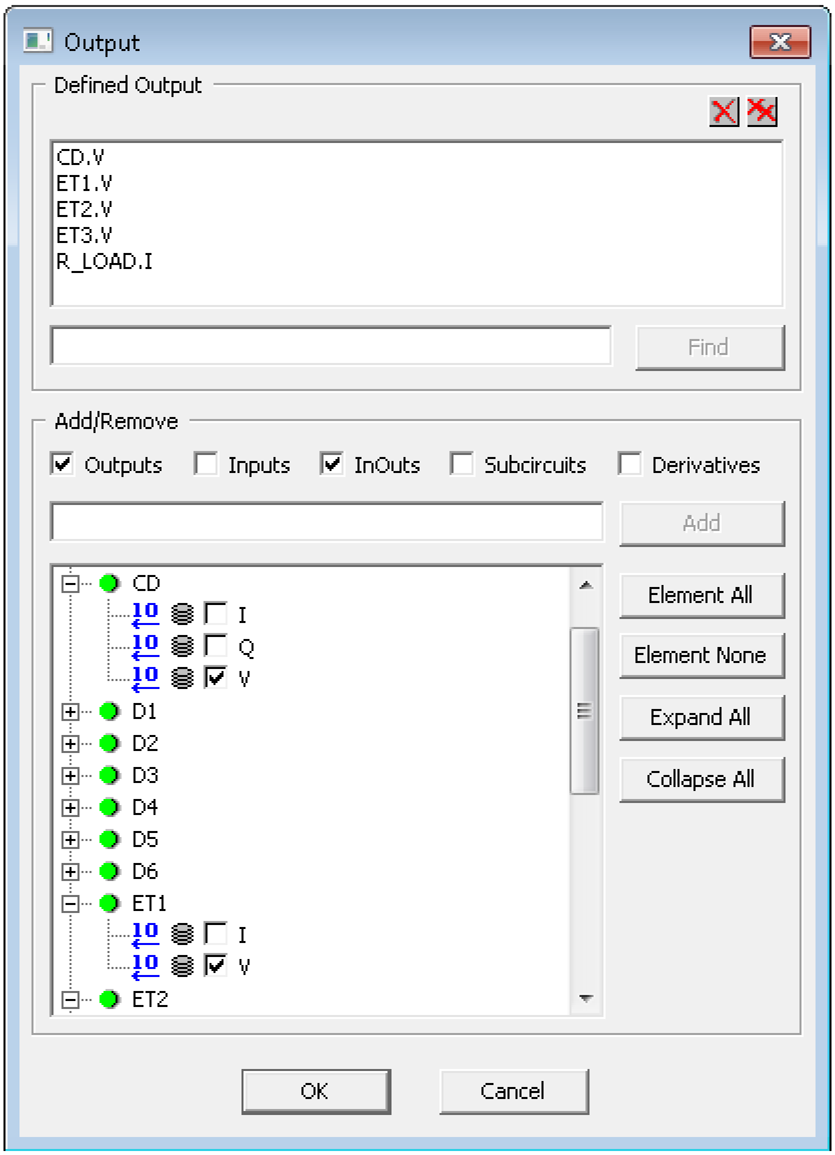
3. For **ET1**, check the **V** box.

   ET1.V is now listed as a **Defined Output**.

   Similarly, check the **V** boxes for **ET2** and **ET3** to add **ET2.V** and **ET3.V** to the **Defined Output** list.

4. Use the **Find** field to locate the **R_LOAD** item and check the **I** box.
5. Use the **Find** field to locate the **CD** element, and check the **V** box.

   When you have finished, the **Defined Output** field will list the items you selected.



6. Click **OK** to close the dialog box.

Next, you will define the Solution Options and Analysis Parameters to be used when analyzing the circuit.

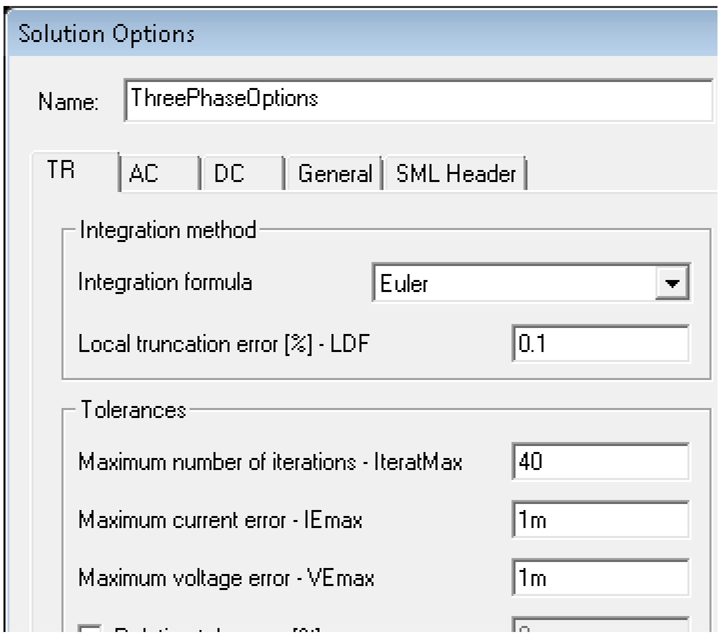
Defining Solution Options and Analysis Parameters

Simulation parameters control the simulation process. The choice of simulation parameters is important for a successful simulation. There are general and circuit simulator parameters. The values obtained during a simulation provide valuable information about the quality of a simulation result.

To set up solution options:

1. Right-click on **Analysis** in the **Project Manager** window, and select **Add Solution Options** from the short-cut menu.

   This displays the **Solution Options** dialog box.

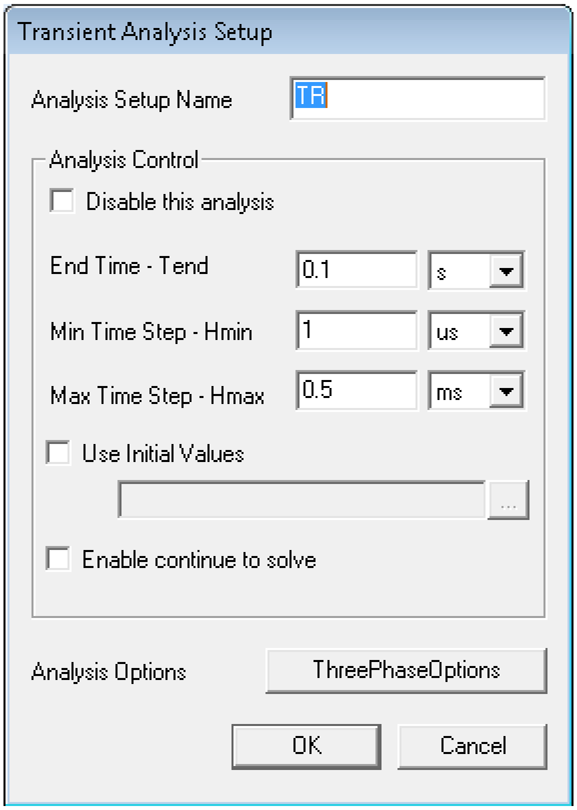


2. On the **TR** tab, change the **Integration formula** to **Euler**, the **Maximum number of iterations** from **40** to **20** and the **Local truncation error** from **1** to **0.1**. Change the **Name** to **ThreePhaseOptions**. Leave all other settings unchanged.
3. Click **OK** to apply the changes.

   An icon for the new options named **ThreePhaseOptions** appears under **Analysis** in the **Project Manager** window.

4. Right-click on **Analysis** in the **Project Manager** window, and select **Solution Setup>Add Transient** from the shortcut menu.

   This opens a **Transient Analysis Setup** dialog box.



5. Change simulation **End Time** from **40ms** to **0.1s**; **Min Time Step** from **10us** to **1us**, and **Max Time Step** from **1ms** to **0.5ms**.
6. Click the **Analysis Options** button and select **ThreePhaseOptions** from the **Select Solution Options** dialog box.
7. Click **OK** to close the **Select Solution Options** dialog box. Click OK again to close the **Transient Analysis Setup** dialog box.

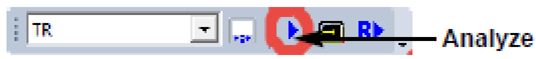
Now that you have set up the necessary conditions for finding solutions for the example circuit, the next step is to start an analysis.

Starting an Analysis

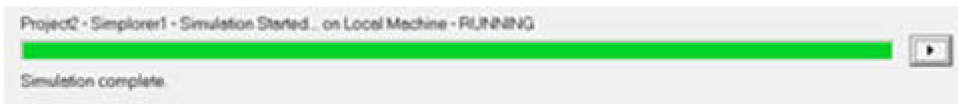
To start an analysis:

1. Start the analysis (simulation) by selecting the **Simplorer Circuit>Analyze** menu command.

   You can also right-click on **Analysis** in the **Project Manager** window and click **Analyze** on the short-cut menu, click the **Analyze** icon on the **Simplorer Simulation** toolbar, or press **F12** on your keyboard to start the analysis.



The simulation model is compiled and outputs are calculated. During the simulation run, the name of the model is visible in the simulation toolbar in the **Progress** window, and a button to stop the simulation is available.



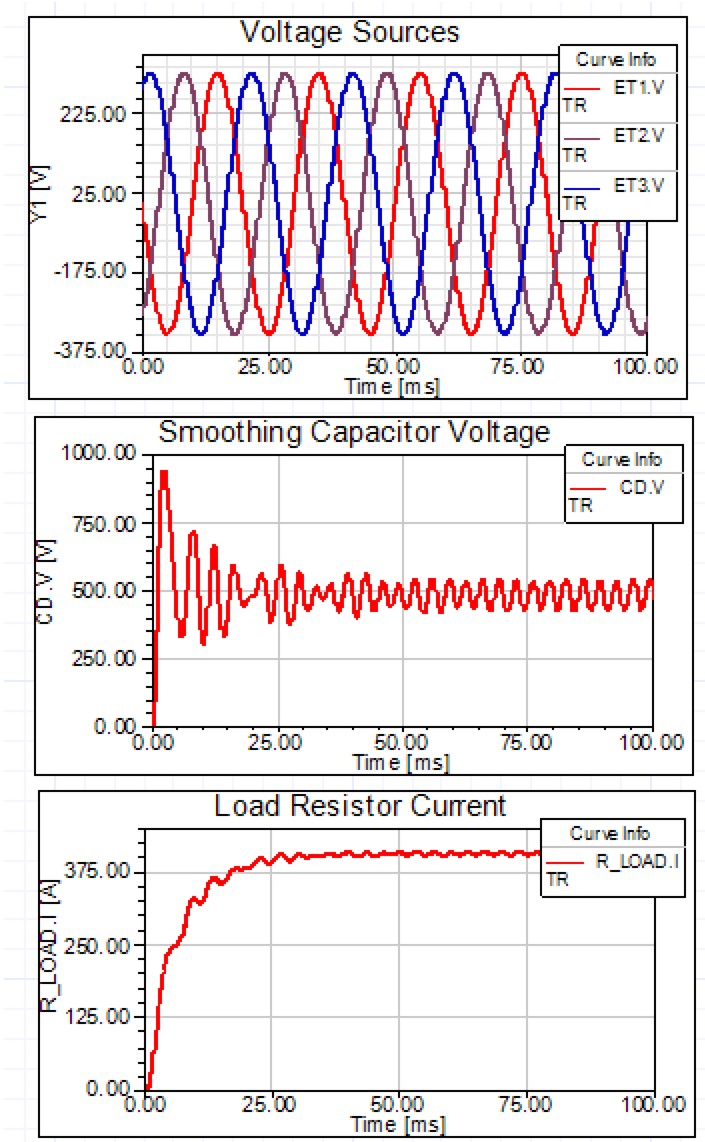
2. After the simulation, the output quantities are displayed in the plots on the sheet or in the report windows. Output data are also saved in an .sdb file (Simplorer Database).

Plotting Rectifier Model Simulation Results

In this section, you will create reports that graph the following outputs:

- Voltages of the sources ET1.V, ET2.V, and ET3.V
- Voltage of the smoothing capacitor CD.V
- Current of the load resistor R_LOAD.I

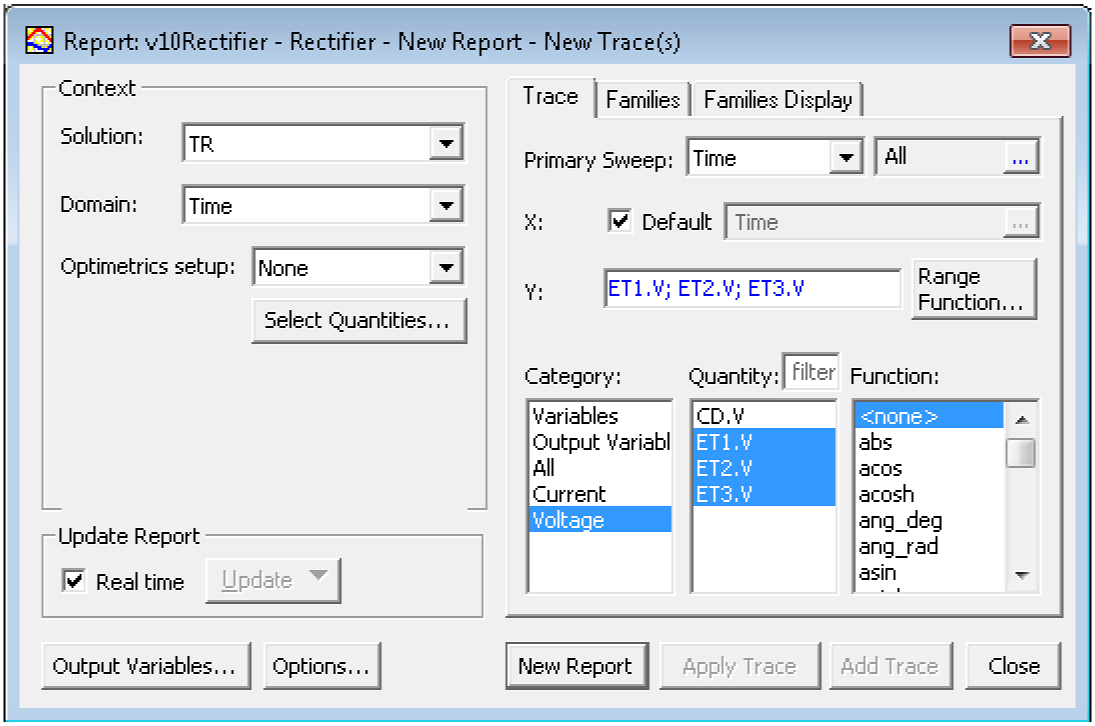
The resulting plots should resemble those shown below.



Begin by creating a 2D rectangular report that plots the three-phase voltage source outputs.

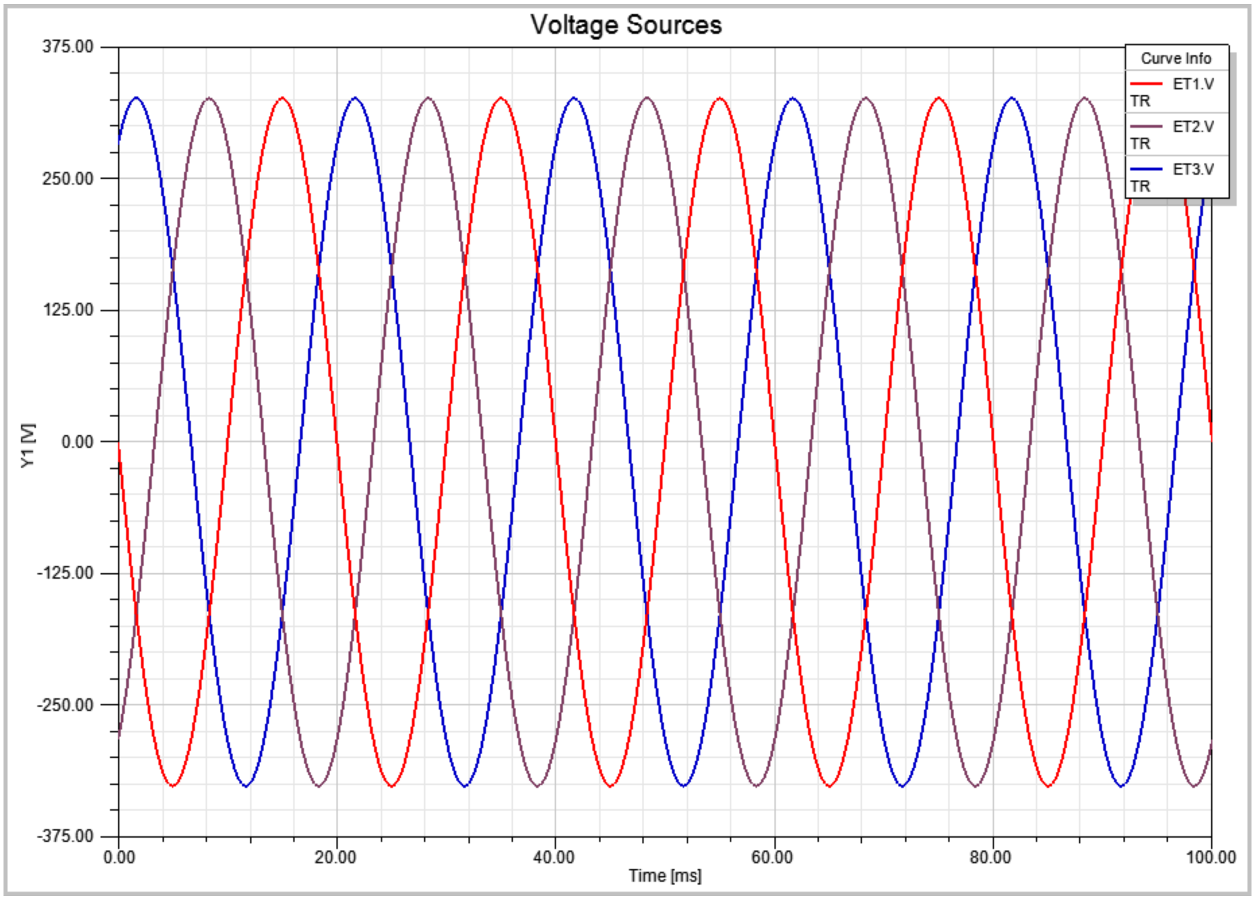
1. Right-click on **Results** in the Project tree and select **Create Standard Report>Rectangular Plot** from the context menu.

   A new **Report** window appears.



2. Under Category, select **Voltage**. While pressing the **Ctrl** key, select ET1.V, ET2.V, and ET3.V in the **Quantity** list. Keep all other settings unchanged.

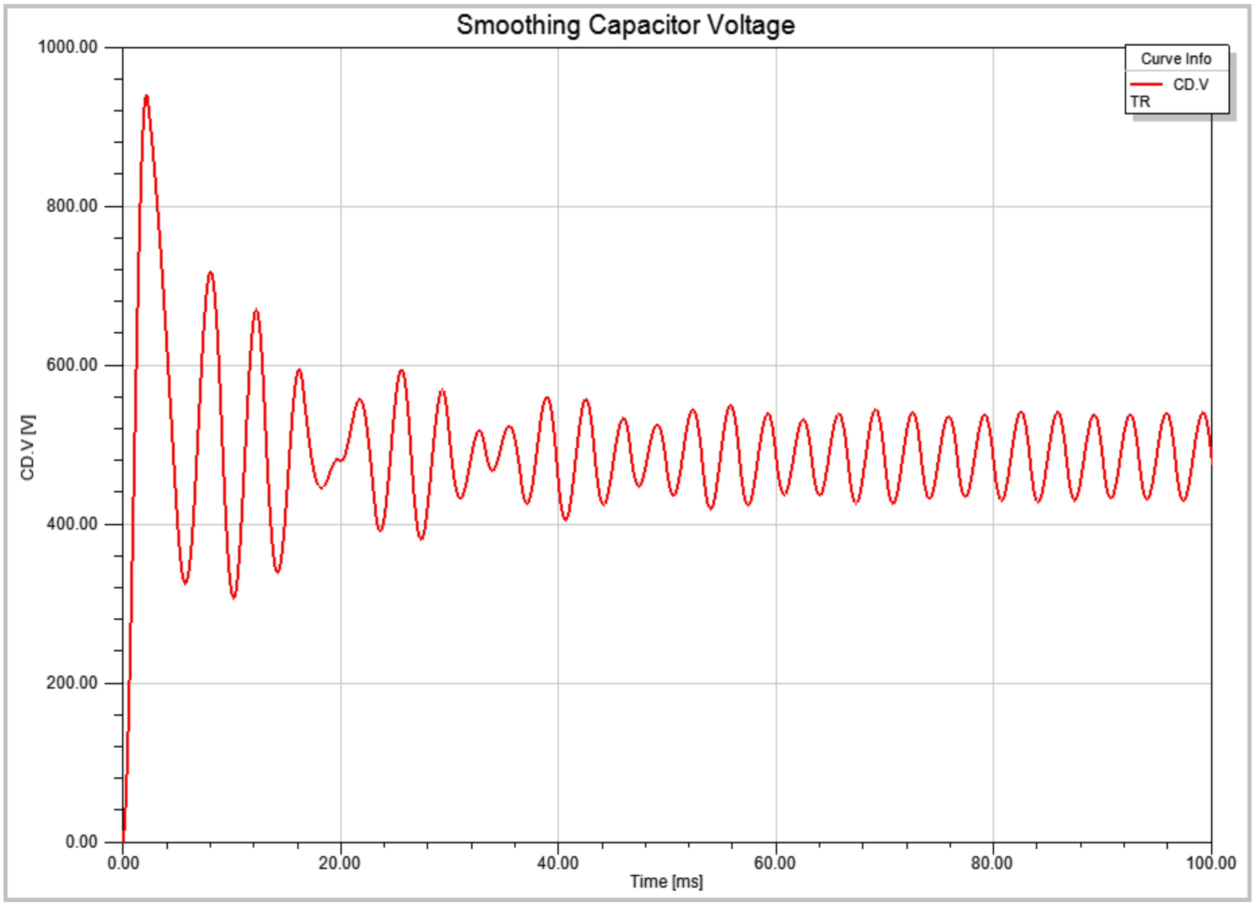
3. Click the **New Report** button to create a report similar to the following:



A **Result** named *XY Plot 1* appears in the **Project Manager** tree.

4. Without closing the **New Report** dialog, select CD.V in the Quantity list.
5. Click **New Report** to create a report showing the smoothing capacitor voltage similar to the

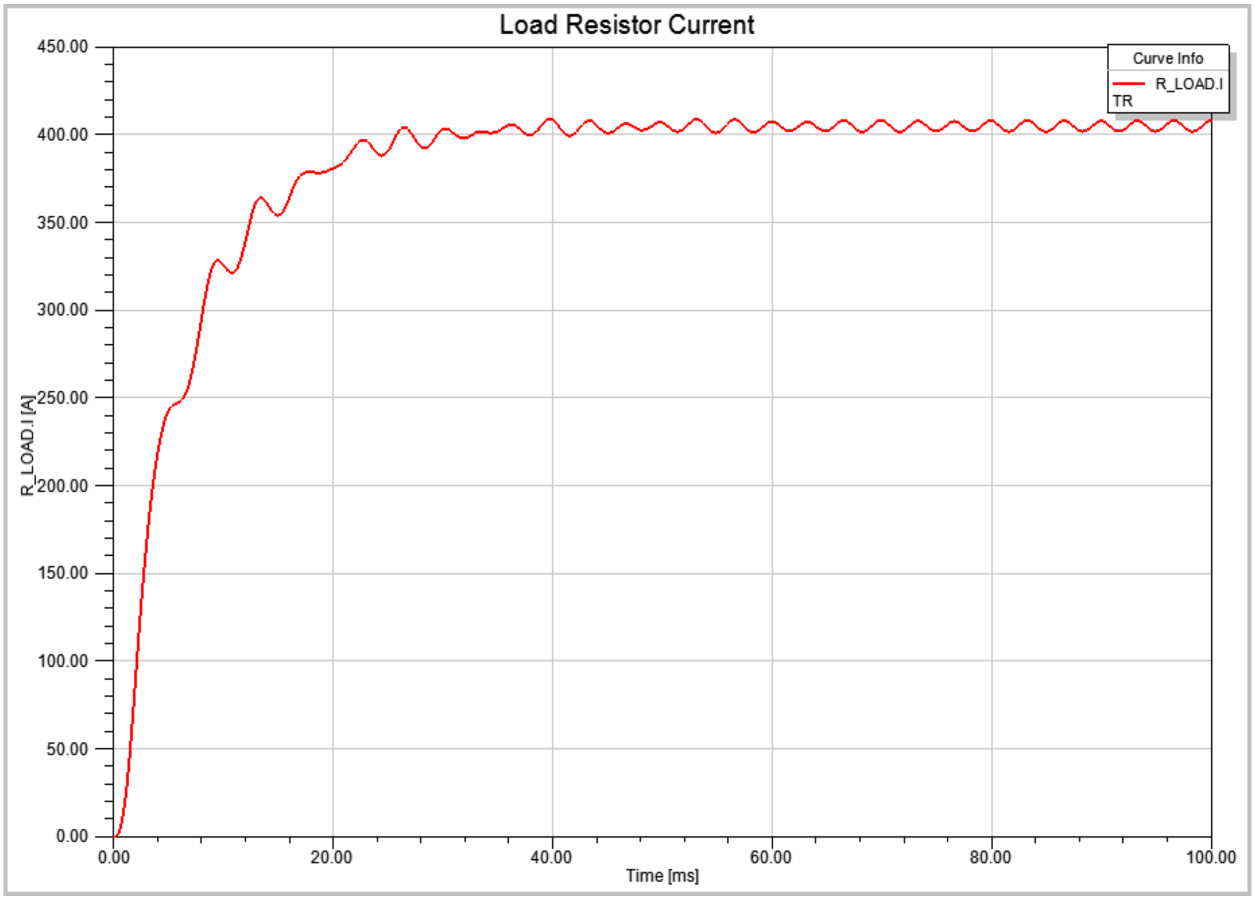
following:



A **Result** named *XY Plot 2* appears in the **Project Manager** tree.

6. Without closing the **New Report** dialog, select Current in the Category list; then select R_ LOAD.I in the Quantity list.
7. Click **New Report** to create a report showing the load resistor current similar to the fol-

lowing:



A **Result** named *XY Plot 3* appears in the **Project Manager** tree.

8. Click **Close** to close the **New Report** dialog box.

Simplorer allows you to customize reports such as the one you created in this example in many ways. For example, you can rename plots by right-clicking on a plot name in the **Project Manager** tree, and selecting **Rename** from the context menu. You can also adjust the background color; grid scale and color; text font, size, and color; trace color, line style, and thickness; legend text; add various data markers, etc.

You can also use the tools in Simplorer's **Draw** menu to add your own custom elements and text to your reports. Detailed information on *Generating and Modifying Reports* is available in the online help.

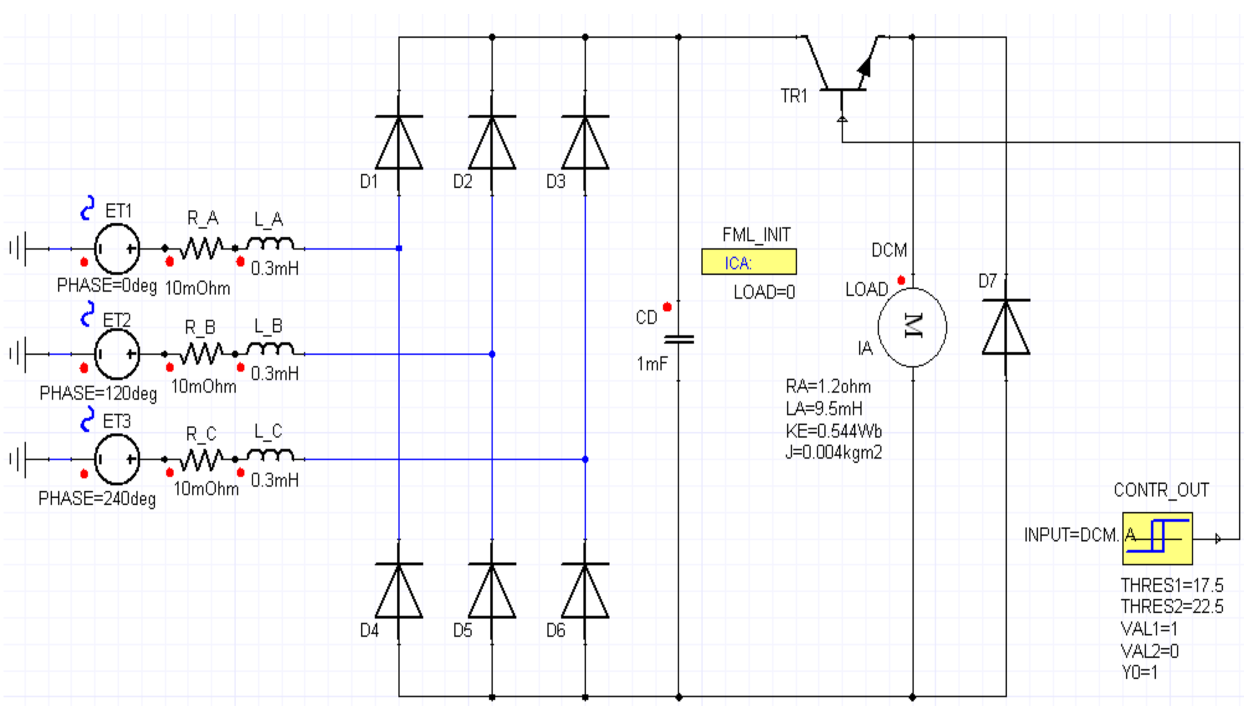
4 - Hysteresis Current-Controlled DC-Motor Start-Up

In this chapter, the example from chapter 3 is modified as follows:

- The resistive/inductive load is replaced with a real machine model (DC motor with permanent excitation).
- The example is then expanded to create a simple, current-controlled machine model by adding a two-point hysteresis element, a chopper transistor, and a freewheeling diode. The other parts of the circuit — the three-phase power supply with rectifier bridge — remain unchanged.
- In the second example, the controller is modeled with state graph components.

Modify the Rectifier Model Design

The figure below shows a Schematic sheet of the simulation model you will design in this section. The model consists of a three phase power supply, a rectifier bridge with static diodes and their characteristics, a smoothing capacitor, a DC machine model, Chopper transistor, and freewheeling diode.



Deleting the Resistive/Inductive Load

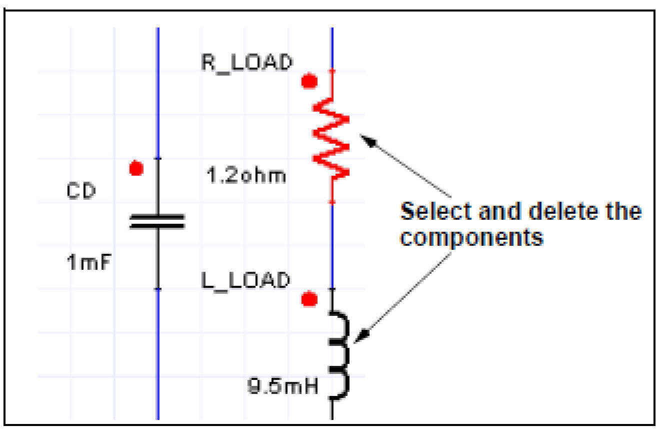
First, you need to delete the components which are not used in the modified simulation model. In place of the resistive/inductive load, you will add a DC machine model, a freewheeling diode, and a chopper transistor.

1. Select and delete resistor **R_LOAD** and inductor **L_LOAD**.

   You can select them one at a time (selected elements turn red); or drag the mouse around them to select both elements.

   To delete the selected components you can:

   - Choose **Edit>Delete.**
   - Right-click and choose **Delete** from the short-cut menu.
   - Press **CTRL+X**.
2. Select and delete the remains of the interconnecting wires.



Saving the Sheet with a New Name

1. Choose **File>Save As**.
2. Enter a new file name.
3. Click **OK**.

Placing and Arranging the New Block Components on the Sheet

First, place and arrange the new components needed for the PI Controller in the extended simulation model. The **Component Libraries** provides access to the library containing the Simplorer basic components, which are used for this example.

Module	Group	Component	Quantity
Blocks	Continuous Blocks	GAIN: Gain INTG: Integrator	2 1
	Sources Blocks	CONST: Constant Value	1
	Signal Processing Blocks	LIMIT: Limiter SUM: Summation	1 3
Tools	Time Functions	DATAPAIRS: 2D Look-up Table	1

1. Open the Simplorer **Basic Elements** model library folder by clicking the **+** symbol next to the folder. You can also double-click the **Basic Elements** folder icon or name to open it.
2. **Place the component onto the sheet.** Select the folder *Blocks* and then the folder *Continuous Blocks*. Select the component *Gain* and drag the component onto the sheet.
3. **Arrange the component on the sheet.** To connect the components, you must place them in appropriate positions. See the simulation model figure on "Current and Speed Controlled DC Motor" on page 5-1. Orient each component as needed so that it can be easily connected with the other components.
4. Repeat steps 2 and 3 until all new components used in this example are placed on the sheet.

Connecting the New Components

When all the components are arranged, you can connect them as required for this example.

Connect the components as required for the controller. Using the simulation model shown on "Current and Speed Controlled DC Motor" on page 5-1 as a guide, place the cursor on the element pins and set the beginning, the corners, and the end of a wire with the mouse. Press **Esc** to exit the wire mode.

Note	Connections to the motor's **LOAD** (load torque), **IA** (armature current), and **N** (rotor speed) parameters are not visible on the schematic. These connections will be added later.

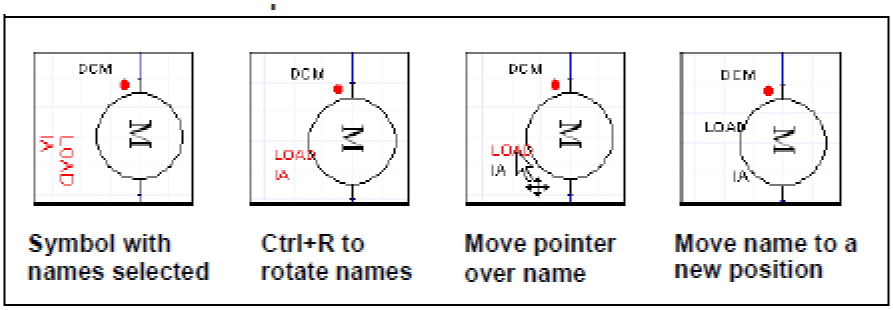
Defining DC Machine Values

1. Define component properties. Double-click the DCMP machine symbol to open its **Parameters** dialog box.
2. On the **Parameters** tab, change the **Name** from **DCMP…** to **DCM**., and define the following machine parameters:
   - **RA: Armature Resistance:***1.2 ohm*
   - **LA: Armature Inductance:***9.5 mH*

- **KE: Back EMF Constant:** *0.544 Wb*
- **J: Rotor Moment of Inertia:***0.004 kgm*^{2}

3. In the **Default Outputs** section of the **Parameters** tab, check **N** and **IA** to enable them as outputs. You will use these later when creating reports.
4. Make the names of the mechanical load and armature current properties visible on the sheet. (These two properties will be used later.)
   a. Click **the Output/Display** tab. In the **Name** column, select **LOAD** (Load Torque). Under **Visibility**, select **Name**. Leave **Location** as-is.
   b. Similarly set visibility and location for **IA** (Armature Current).
   c. Click **OK** to apply the changes.
5. Move the property names to appropriate positions.

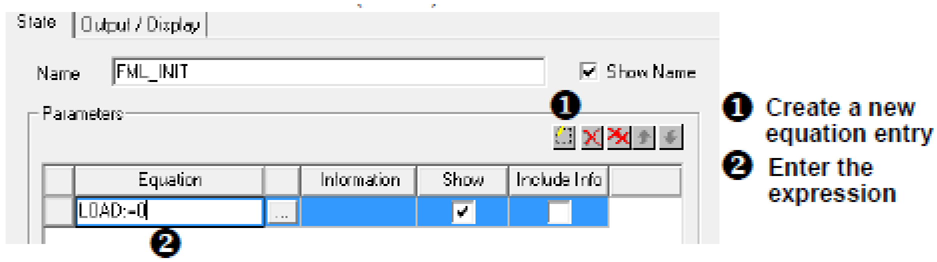
   You can rotate property names displayed on the sheet by clicking the name to highlight it, then pressing **Ctrl+R** repeatedly until the name is oriented as desired. Activate the move mode by moving the mouse pointer over the name you want to move until the mouse-pointer changes into a pointer with a four-headed arrow. Click and drag the name to the new position.



Defining Mechanical Load

The mechanical load of the machine model can be set in many different ways. The variant used in this example is based on an initial value (ICA) component. This component is useful when initial values are used for different models on the sheet. In this example, the equation defined within the ICA component is used to “connect” an initial value, **0**, to the load torque parameter of the machine component. Values in the ICA component are set only once at the simulation start.

1. **Define a new entry within the initial value list.**
   a. Double-click the ICA symbol to open its **Parameters** dialog box.
   b. On the **State** tab, change the **Name** to **FML_INIT** and check **Show Name**.
   c. Create a new **Equation** entry with the **Add** (yellow lightning-bolt) button.
   d. Click in the **Equation** field and type **LOAD:=0**. This creates a new ICA parameter named **LOAD** and assigns it a value of **0** (zero).



e. To display the **LOAD** parameter name and value, check the **LOAD** parameter's **Show** check box.

f. Click **OK** to apply the changes. The new parameter is now defined and is available for connecting with a quantity.

2. Connect the ICA initial value to the LOAD property of the DC machine.

   a. Double-click the DC machine symbol to open its **Parameters** dialog box.

   b. On the **Parameters** tab, type **LOAD** in the **Value** cell for the LOAD property, and click **OK.** This "connects" the DC machine LOAD property to the LOAD value specified in the ICA component.

Name	Value	Unit	Description
LOAD	LOAD	Newto...	Load Torque
RA	1.2	ohm	Armature Resistance

Freewheeling Diode

Define the parameters of the diode.

1. Double-click the diode symbol to open its **Parameters** dialog box and set parameters.
2. Change the **Name** from D*n* to **D7**.
3. Select **Type** and choose **Exponential Function** from the list.
4. Leave all other values as they are. Click **OK** to apply the changes.

Chopper Transistor

The transistor turns on and off depending on the machine current, **IA**. Initially, the transistor is set "on" to start the process.

Define the parameters of the transistor.

1. Double-click the transistor symbol to open its **Parameters** dialog box.
2. Change the **Name** from BJT*n* to **TR1**.
3. Select **Type** and choose **Exponential Function** from the list.
4. Leave all other values as they are. Click **OK** to apply the changes.

Controller Modeling Using Block Elements

Initially, the current-controller is designed using a two-point element with hysteresis. The block element input signal is the DC machine current, **IA**. The output signal controls the chopper transistor, TR1.

1. **Define component properties of the Two-point element.** Double-click the symbol to open its **Parameters** dialog box. On the **Parameters** tab, change the **Name** from TPH*n* to **CONTR_OUT**, then define the following parameters:
   - **THRES1 (Threshold T1)**: **17.5**
   - **THRES2 (Threshold T2):22.5**
   - **VAL1 (Value A1):1**
   - **VAL2 (Value A2):0**
   - **Y0 (Initial Value):1**
2. **Define the Block sample time, TS.** The smaller the block sample time, the more precise the current control of the machine. In this example, the system sample time is used, i.e., the block is calculated using the same sample time that the circuit models use. To ensure that the system sample time is used, set the **TS** (Sample Time) parameter to **0** (zero).
3. **“Connect” the IA parameter of the DC machine (DCM) to the input of the hysteresis block.**
   a. First, on the **Output/Display** tab, uncheck **Show Pin** for the **INPUT** parameter. For **Visibility**, select **Both** to display the parameter and its value.
   b. Next, on the **Parameters** tab, type **DCM.IA** in the **INPUT** parameter **Value** field.
   c. Click **OK** to apply the changes. Position the text as needed.
4. **Connect the output pin of the block with the input pin of the control signal of the bipolar junction transistor.** The cursor changes to cross wires. Connect both pins by clicking them. Press **ESC** to finish the wire mode.

All parameters of the modified simulation model should now have the correct values. The table below lists all new components of the simulation model and their parameter values.

Name	Type	Parameters
DCM		RA (Armature Resistance)=**1.2 ohm** LA (Armature Inductance)=**9.5 mH** KE (Back EMF Constant)=**0.544 Wb** J (Rotor Moment of Inertia)=**0.004 kgm2**
FML_INIT1		LOAD:=0
D7	Exponential Function	ISAT (Saturation Current)=**1e-012 A** VT (Thermal Voltage)=**0.035 V**

Name	Type	Parameters
		RR (Reverse Resistance)=**100k ohm**
TR1	Exponential Function	ISAT (Saturation Current)=**1e-012 A** VT (Thermal Voltage)=**0.035 V** RR (Reverse Resistance)=**100k ohm** CTRL (Control Signal)=**CONTR_OUT.VAL** (pin)
CONTR_OUT		THRES1 (Threshold T1)=**17.5** THRES2 (Threshold T2)= **22.5** VAL1 (Value A1)=**1** VAL2 (Value A2)=**0** Y0 (Initial Value)=**1** TS (Sample Time)=**0** (System time) INPUT (Input Signal)=**DCM.IA**

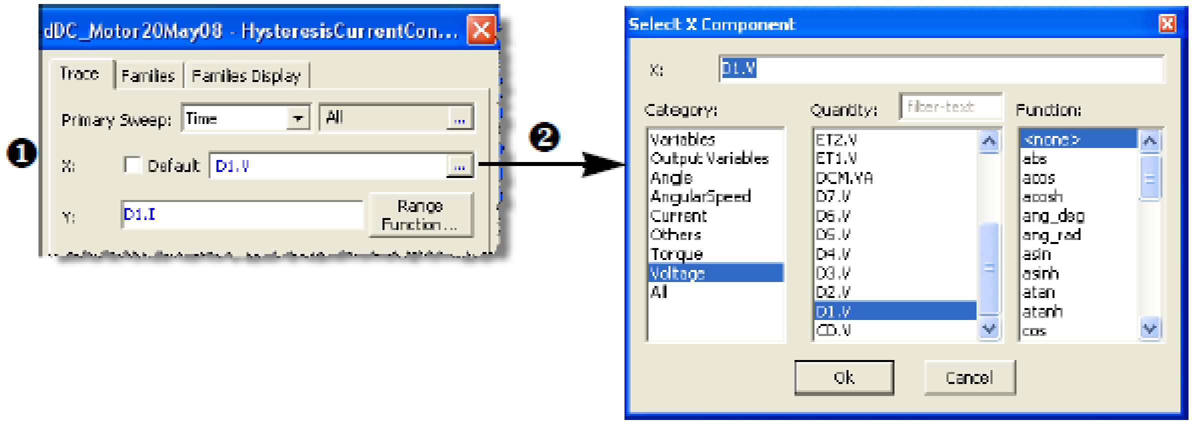
Modifying Report Elements

1. **Define the model output DCM.N.**
   a. In the Project Manager window, under Results on the Project tab, double-click the **XY Plot 1** report icon to open the report window.
   b. **Ctrl**+click each of the traces to select them all, then press **Delete** to remove the traces. You can also right-click and select **Edit>Delete** from the context menu.
   c. Right-click the **XY Plot 1** report icon and select **Modify Report** to open the report dialog box.
   d. In the **Category** list, click **All**, then select **DCM.N** in the **Quantity** list.
   e. Click the **Add Trace** button to display the output.
   f. Click **Close** to close the dialog box.
   g. In the **Project Manager** window, right-click **XY Plot 1** and select **Rename** from the context menu. Change the title of the plot to *Speed*. Press **Enter** to apply the change.
2. **Define the model output DCM.IA.**
   a. Double-click the **XY Plot 2** symbol to open the report window.
   b. Select the **CD.V** trace and use the **Delete** button to delete it.
   c. Create a new output for **DCM.IA**.
   d. Change the title of the plot to **Current**. See also Plotting Rectifier Model Simulation Results.

Display Diode Characteristic

Plots can also be used to display component characteristics. For such cases, the X-axis is defined in some quantity other than simulation time.

1. Define the axes quantities.
   a. In the Project Manager window, under Results on the Project tab, double-click the **XY Plot 3** symbol to open the report dialog box.
   b. Select the **R_LOAD.I** trace and **Delete** it.
   c. Create a new Y-axis output for **D1.I**. Remember to use **Simplorer Circuit>Output Dialog** to define output quantities as needed.
   d. Select a new X-axis quantity by unchecking **Default**. Then click [**...**] and select **D1.V** on the Select X Component dialog box.



e. Click **OK** to close the Select X Component dialog box.

f. In the report dialog box, click **Apply Trace** to apply the changes; then click **Close** to close the dialog box.

g. By default, the trace type is a continuous (solid) line. To edit the trace properties so that you can see the individual simulation points on the curve, click the trace to select it (The cursor changes color when hovering over a selectable display element). In the dock-able Properties window, change the **Trace Type** to **Discrete**.

To view the area of interest on the characteristic curve, you need to manually edit the X- and Y-axis properties.

a. Double-click the X-axis to select it and open its **Properties** dialog box.

b. On the **Scaling** tab, uncheck **Auto Units** and select *V* as the **Units** value. Check **Specify Min**, then enter *0.55* in the **Min Value** field. Similarly, check **Specify Max** and enter *0.9* for the **Max** value. Next check **Specify Spacing** and enter *0.05* in the **Spacing** field to set the interval between major tick marks on the X-axis. Leave the rest of the settings as they are and click **OK** to apply the changes and close the dialog box.

c. Double-click the Y-axis to select it and open its **Properties** dialog box.

d. On the **Scaling** tab, uncheck **Auto Units** and select *A* as the **Units** value. Check **Specify Min** and enter *0* in the **Min Value** field. Check **Specify Max** and enter *0.1* for the **Max** value. Check **Specify Spacing** and enter *0.025* in the **Spacing** field to set the interval between major tick marks on the Y-axis. Leave the rest of the settings as they are and click **OK** to apply the changes and close the dialog box.

e. Change the title of the plot to **Characteristic**.

Defining Simulation Parameters

Simulation parameters control the simulation process. The values chosen for a simulation determine the success, and affect the quality, of a simulation result.

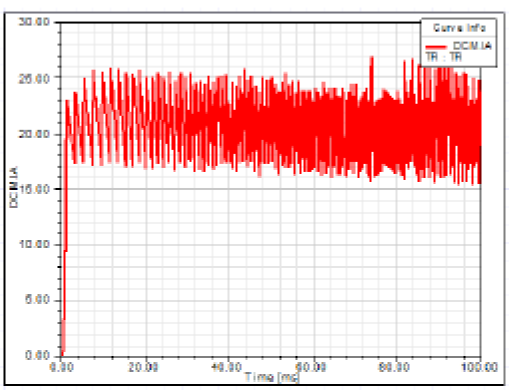
1. Choose **Simplorer Circuit>Add Solution Setup>Transient** to define the simulation parameters.
2. Change the default values for simulation **End Time** from 40ms to 0.5ms, for **Min Time Step** from 10us to 1ns, and for **Max Time Step** from 1ms to 1us. Click **OK** to apply the changes.

Starting Simulation (Block Components)

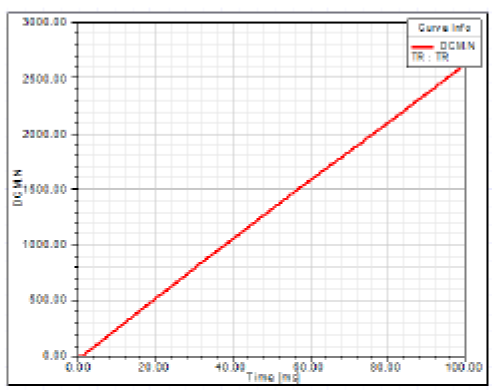
Start the simulation with the **Simplorer Circuit>Analyze** menu command or right-click on the **Analysis** icon in the **Project Manager** window and select **Analyze** from the short-cut menu. The simulation model is compiled and calculated.

Simulation Results (Block Components)

The Speed and Current plots display the simulation results for the machine armature current (**DCM.IA**) and speed (**DCM.N**). Depending on the armature current, the two-point element with hysteresis controls the switching behavior of the chopper transistor. The speed for the DC motor no-load starting torque approaches 2613 rpm.

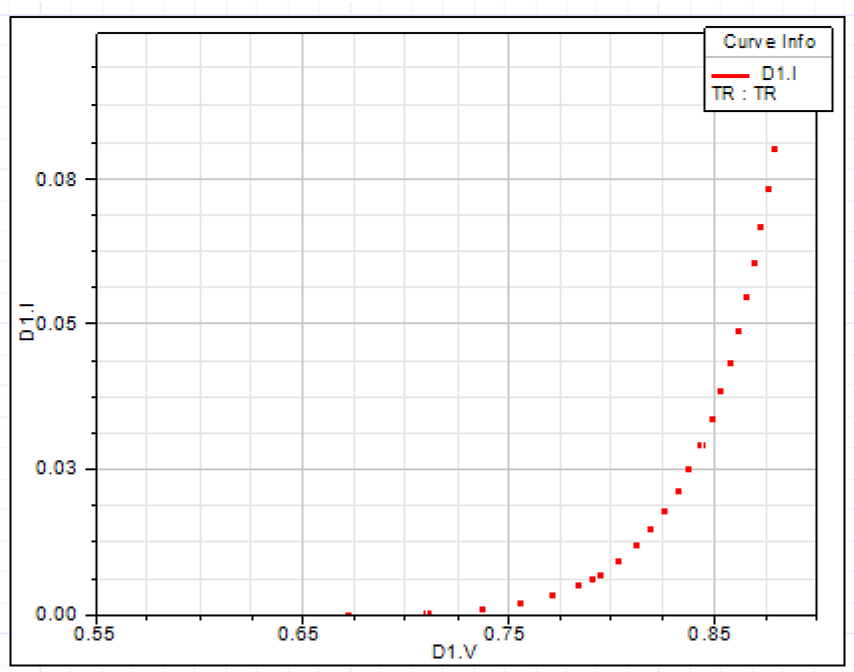


Armature Current of the DC machine



Rotor speed of the DC machine

The Characteristic plot displays diode D1's current as a function of its voltage and should look similar to the image below.



Controller Modeling Using State Graph Components

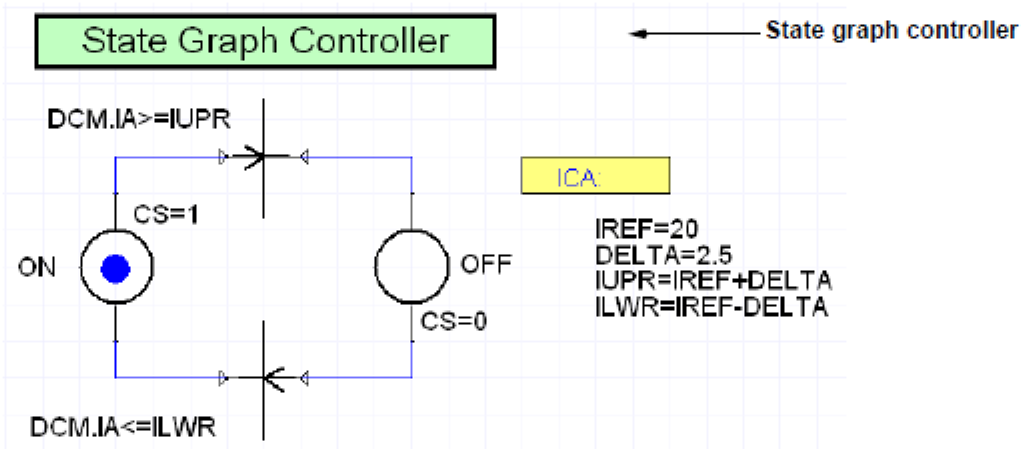
Simplorer's state graph module, which is based on the *Petri Net* theory, allows you to model event-driven, discontinuous processes. The theoretical basis of the modeling divides a system into significant states and events, and transitions from one state to the other. The following procedure explains the modeling of the two-point hysteresis controller with state graph components.

First, place and arrange the state graph components used in the modified simulation model. Any unused area of the sheet may be used.

1. Choose the Simplorer model library *Basic Elements*.
2. Place the component onto the sheet. Select the folder *States.* Select the component *State 11*and drag it onto the sheet.
3. Arrange the component on the sheet. Orient the component so that it can be connected with the other components in the manner show in figure below.
4. Repeat steps **2** and **3** until all new components used in the state graph are placed on the sheet

Module	**Group**	**Component**
States		State 11 2x
		Transition 2x
Tools	Equations	Initial Values 1x

5. All of the required components for the state graph are now on the sheet. To connect the components, you must place them in appropriate positions as shown in the figure below. Note in particular the directions of the arrows on the ***Transition*** components and rotate or flip components as needed.



Connecting the State Graph Components

When all of the state graph components are arranged, you can connect them as required for this example. Make certain to consider the direction of the transition components.

1. **Activate the wire mode.** Choose **Draw>Wire** or use the **Ctrl+W** keyboard shortcut. The cursor changes to cross wires.
2. **Connect the components as required for the circuit.** Place the cursor on the element pins and set the beginning, the corners, and the end of a wire with the mouse. Press **Esc** to finish and exit the wire mode.

Defining the Properties of State Graph Components

A process sequence can be considered as a sequence of states. The current state is called *active*. Switching the activity from a given state to its successor state is called an *event*. An event occurs only if: all previous states are active, all following states are inactive, and the transfer condition — in the form of a logical expression — is true. At the beginning of the simulation, one state *must* be defined as *active*.

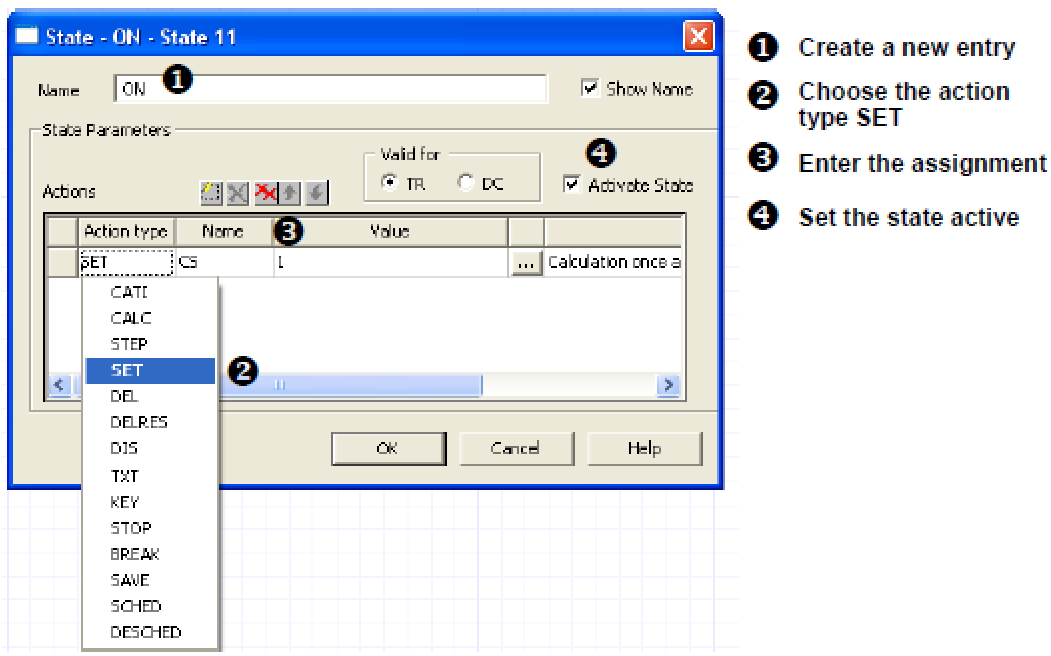
1. Define the parameters of the ***state ON.***
   a. Double-click the **STATE11** symbol on the left to open its Parameters dialog box.
   b. Change the **Name** from *STATE…* to **ON**.
   c. Create a new *SET* **Action Type** entry with the **Add** button.

d. Type **CS** in the **Name** field and **1** in the **Value** field. This entry means that the variable *CS* (control signal) is set to *1* if the state is active. You can click the **[...]** button to open the **Calculator** window and confirm the expression **CS:=1** is present. (Click **OK** or **Cancel** to close the window.)

e. Check the **Activate State** box to set the state *active* at the beginning of the simulation.

f. Ensure that the **TR** radio button is selected.

g. Click **OK** to apply the changes.



The blue circle in the symbol indicates the state is *active*. During a simulation, the state markers “run” through the state graph, depending on the active states in the simulation process.

2. Define the parameters of the OFF state.

   a. Double-click the state symbol on the right to open the property dialog and set parameters.

   b. Change the **Name** from *STATE…* to **OFF**.

   c. Create a new *SET* **Action Type** entry with the **Add** button.

   d. Type **CS** in the **Name** field and **0** in the **Value** field. This entry means that the variable *CS* (control signal) is set to *0* if the state is active.

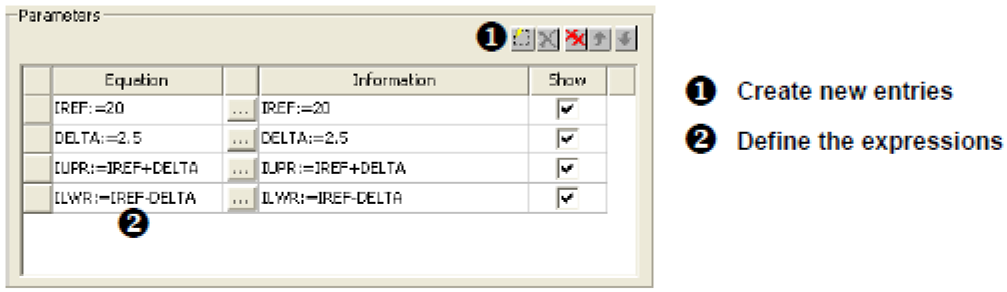
   e. Click **OK** to apply the changes.

3. Define the parameters of the **initial value component.**

   a. Double-click the **ICA** symbol to open the component dialog box and set parameters.

   b. Turn off the Name and then create four new entries with the **Add** button.

c. Click in the **Equation** field and type the *name***:=***value* corresponding to the picture.



You can also enter a description of each equation in the **Information** field and display the descriptions on the schematic by enabling **Show**.

d. Click **OK** to apply the changes.

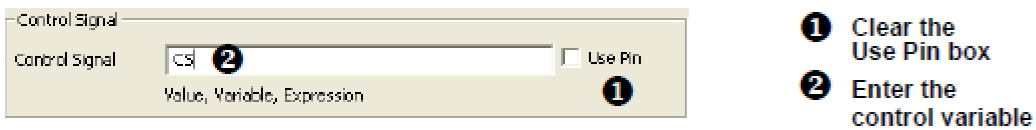
4. Define the parameters of the first transition.
   a. Double-click the top **Transition** symbol to open the parameter dialog box.
   b. Type **DCM.IA>=IUPR** in the **Condition for transition** field. This entry means that the condition becomes true if the machine armature current (**DCM.IA**) is greater-than or equal-to the variable **IUPR**. The variable **IUPR** is defined in the initial value component.
   c. Clear **Show Name**.
   d. Check **Show Condition** to display the condition on the schematic.
   e. Click **OK** to apply the changes.
5. Define the parameters of the second transition.
   a. Double-click the lower **Transition** symbol to open the parameter dialog box to define the transfer condition.
   b. Type **DCM.IA<=ILWR** in the **Condition for transition** field. This entry means that the condition becomes true if the machine armature current is less-than or equal-to the variable *ILWR*. This variable is also defined in the initial value component.
   c. Turn off the name, and check **Show Condition** to display the parameter value.
   d. Click **OK** to apply the changes.

Note	Using the “=” operator type forces the simulator to synchronize on the condition with the minimum time step. Because of this, the state graph works more precisely than the two-point hysteresis component, but the processing time of the simulation is longer.

Using Name References

To control the switching behavior of the transistor, **TR1**, you need to enter the control variable *CS* in the transistor dialog.

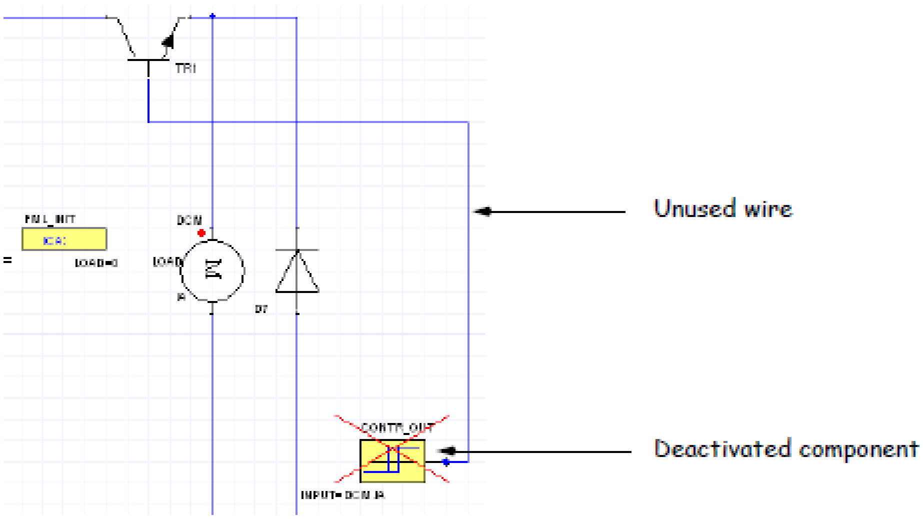
1. Double-click the transistor symbol to open the Parameters dialog to define the control parameter.
2. Clear the **Control SignalUse Pin** box, and enter the control variable **CS** in the **Control Signal** field.
3. Click **OK** to apply the changes.



Deactivating Components on the Sheet

You can deactivate separate components or parts of a model sheet for a simulation run. The components and all of their properties remain on the sheet, but the simulator ignores the deactivated components. The connection between the terminals is considered *open*. This feature is especially helpful when you want to test simulation models with several different elements and parameters. Components can be deactivated with the **Edit>Deactivate (Open)**menu command or the same command in the shortcut menu for the selected element. A deactivated (Open) component will have a large red **X** over it.

Select the hysteresis block *CONTR_OUT* and choose **Edit>Deactivate (Open)**. The wires, connected with the machine model and transistor, remain on the sheet.



All parameters in the modified simulation model now have the correct values. The table below lists all new components of the simulation model and their parameter values.

Name	Type	Parameters
TR1	Exponential Function	Saturation Current [A]=**1e-12** Thermal Voltage[V]=**0.035** Reverse Resistance[Ω]=**100k** Control Signal=**CS**
CONTR_OUT	***Deactivated***	Threshold T1=**17.5** Threshold T2= **22.5** Value A1=**1** Value A2=**0** Initial Value=**1** Sample Time=0 (System) Input**=DCM.IA**
TRANS1		**DCM.IA>=IUPR**
TRANS2		**DCM.IA<=ILWR**
ON		SET: **CS:=1**
OFF		SET: **CS:=0**
FML_INIT2		**IREF:=20**

Name	Type	Parameters
		DELTA:=2.5 **IUPR:=IREF+DELTA** **ILWR:=IREF-DELTA**

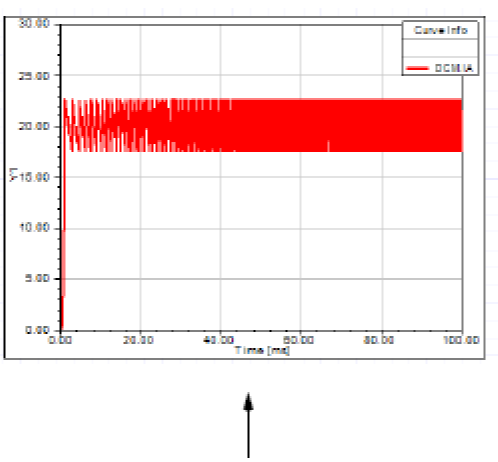
Starting Simulation (State Graph)

Start the simulation with the **Simplorer Circuit>Analyze** menu command or the right click on the Analysis icon on the Project window and select **Analyze** from the short-cut men. The simulation model is compiled and calculated.

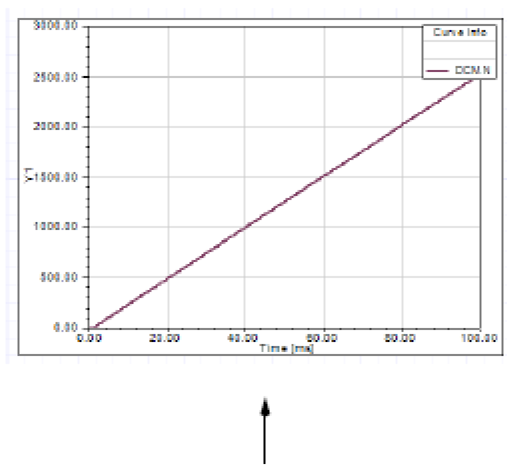
Simulation Results (State Graph)

The *Speed* and *Current* plots display the simulation results for the machine armature current (**DCM.IA**) and speed (**DCM.N**). Depending on the armature current, **DCM.IA**, the state graph controls the switching behavior of the chopper transistor. The speed for the DC motor no-load starting condition approaches 2550 rpm.

The tolerance band of the state graph controller is more precise than for the hysteresis controller, because the “=” operator type in the state graph forces the simulator to synchronize on the condition with the minimum time step. To force any of the block models to calculate more precisely, you can define a special sample time in the model’s property dialog.



DC machine Current



DC machine Start-up speed

5 - Current and Speed Controlled DC Motor

In this chapter the example from Chapters 3 and 4 is extended further:

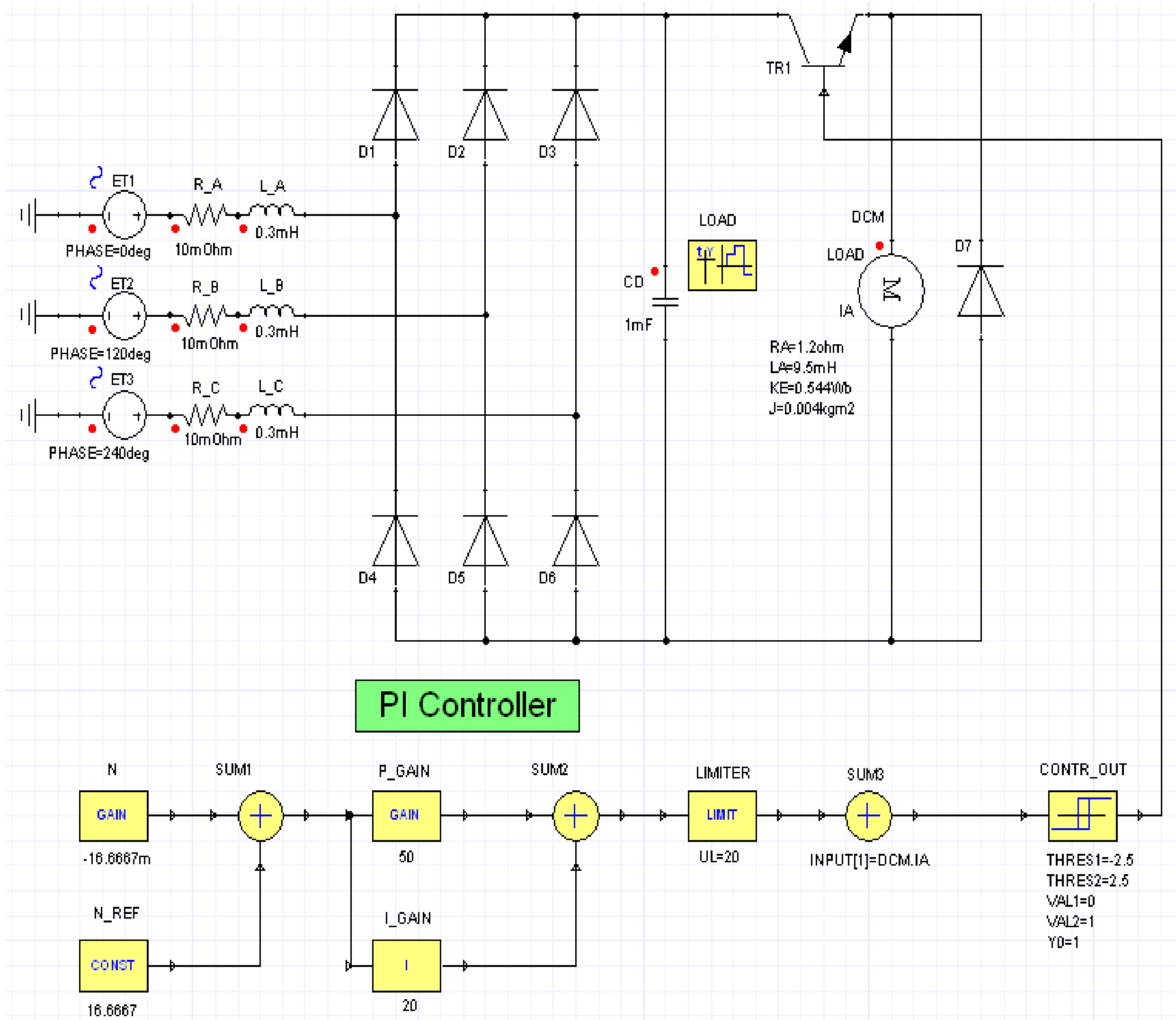
- The state graph/hysteresis controller is replaced with a PI (*proportional-integral*) controller implemented using block components.
  The other parts of the circuit, such as the three-phase power supply with rectifier bridge, remain unchanged.

In addition to basic functions (selecting, placing, arranging, and connecting components), this chapter introduces the following Simplorer features:

- Modeling with block components
- Examining Block Sequence
- Using Pins for parameter transfer
- Using Characteristic components

Modify the State Graph Design

The figure below shows the Schematic sheet of the simulation model with the corresponding values of components: three phase power supply, the rectifier bridge, smoothing capacitor, motor, and the PI controller.



Deleting the State Graph

First, you need to delete the components that are not used in the modified simulation model. Instead of the state graph or simple hysteresis controller, you will use a PI controller consisting of block diagram components.

1. Select the components to delete. Hold down the **Ctrl** key and select the state graph components: **FML_INIT1**, and **FML_INIT2**, **STATE_11** (both ON and OFF), and the two **Transition** components by clicking them to highlight the components.
2. Delete the components. Choose **Edit>Cut**. The components are removed from the sheet. You can also press **Ctrl** and the **X** key at the same time; or you can use the **Delete** key on your keyboard.

3. Delete the remaining wires. Select any remaining unused wires by holding down the **Ctrl** key and clicking to select them. Press **Ctrl+X** to remove the wires.

Saving the Sheet with a New Name

1. Choose **File>Save As**.
2. Enter a new file name.
3. Click **OK**.

Placing and Arranging the New Block Components on the Sheet

First, place and arrange the new components needed for the PI Controller in the extended simulation model. The **Component Libraries** provides access to the library containing the Simplorer basic components, which are used for this example.

Module	Group	Component	Quantity
Blocks	Continuous Blocks	GAIN: Gain INTG: Integrator	2 1
	Sources Blocks	CONST: Constant Value	1
	Signal Processing Blocks	LIMIT: Limiter SUM: Summation	1 3
Tools	Time Functions	DATAPAIRS: 2D Look-up Table	1

1. Open the Simplorer **Basic Elements** model library folder by clicking the **+** symbol next to the folder. You can also double-click the **Basic Elements** folder icon or name to open it.
2. **Place the component onto the sheet.** Select the folder *Blocks* and then the folder *Continuous Blocks*. Select the component *Gain* and drag the component onto the sheet.
3. **Arrange the component on the sheet.** To connect the components, you must place them in appropriate positions. See the simulation model figure on "Current and Speed Controlled DC Motor" on page 5-1. Orient each component as needed so that it can be easily connected with the other components.
4. Repeat steps 2 and 3 until all new components used in this example are placed on the sheet.

Connecting the New Components

When all the components are arranged, you can connect them as required for this example.

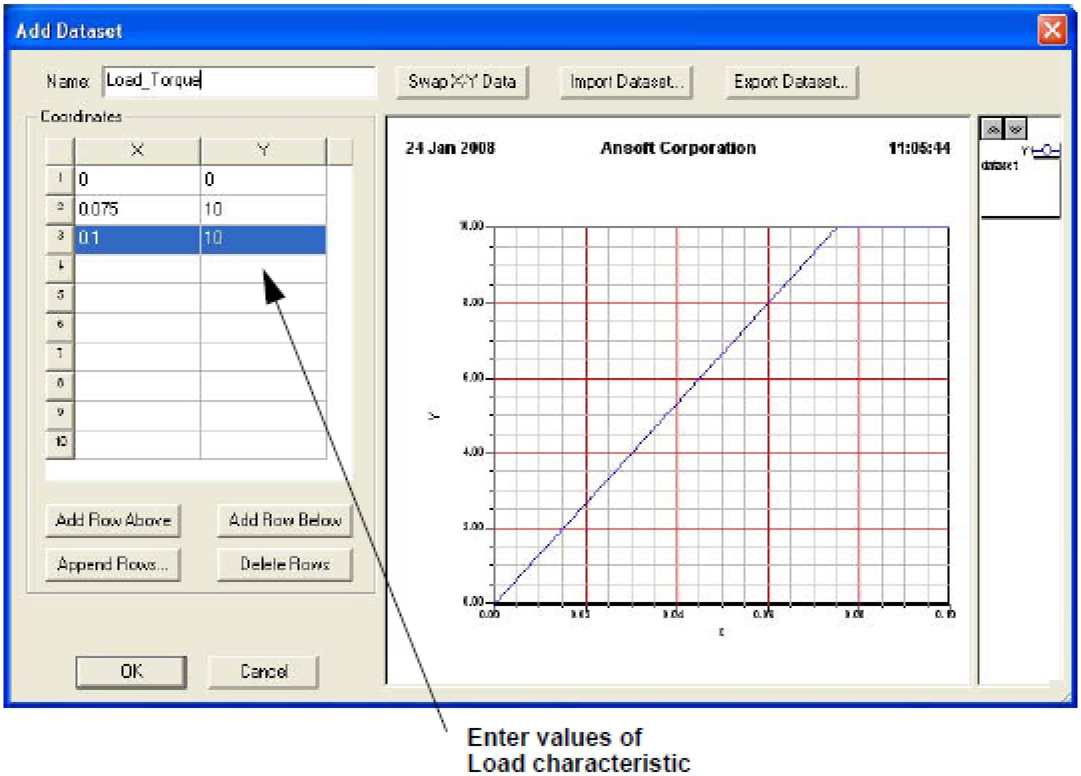
Connect the components as required for the controller. Using the simulation model shown on "Current and Speed Controlled DC Motor" on page 5-1 as a guide, place the cursor on the element pins and set the beginning, the corners, and the end of a wire with the mouse. Press **Esc** to exit the wire mode.

Note	Connections to the motor's **LOAD** (load torque), **IA** (armature current), and **N** (rotor speed) parameters are not visible on the schematic. These connections will be added later.

Defining Mechanical Load (Block)

The **DATAPAIRS: 2D Look-up Table** (already placed on the sheet) allows the definition of wave-forms from a set of fixed data points either with linear interpolation between them (straight lines from point-to-point), or rectangular lines between them (two orthogonal lines — parallel to the coordinate axes — from point-to-point). The X-values of the data-pairs must be monotonously increasing. The last slope is effective for all values outside the X range. If you want to have a constant value outside the X-range, you need to define two data-pairs with the same Y-value at the end. In this example, the 2D Lookup Table is used to define the mechanical load of the DC machine.

1. Define the load characteristic.
   a. Double-click the **2D Look-up Table** symbol to open the **Parameters** dialog box and set parameters.
   b. Change the **Name** to **LOAD**.
   c. Choose **Without** from the **Interpolation** list.
   d. Click **Characteristic** to open the **Characteristic** dialog box.
   e. Choose the **Internal Reference** radio button, then click **Datasets** to open the **Datasets** dialog box.
   f. Click **Add** to open an **Add Dataset** dialog box. Enter the values as shown in the figure below.
   g. A default unique name is assigned to each dataset. Change the **Name** to something more descriptive such as *Load_Torque*.



h. Click **OK** to save the dataset. The **Add Dataset** dialog box closes - returning you to the main **Datasets** dialog box.

i. Click **Done** to close the **Datasets** dialog box. The **Characteristic** dialog box should now show the name of the dataset you created in its **Dataset** field.

j. Click **OK** to close this dialog box.

k. On the 2D Look-up Table **Parameters** tab, check **Value** in the **Outputs** section.

l. Click **OK** to close the 2D Look-up Table **Parameters** dialog box.

2. **Connect the output parameter value of the *LOAD* characteristic with the load of the DC machine.**

   a. Open the *LOAD* characteristic **Parameters** dialog box.

   b. On the **Output/Display** tab, uncheck **Show Pin** for the *VAL* parameter, and click **OK** to close the dialog box.

   c. On the **Parameters** tab of the DC machine **Parameters** dialog box, type **LOAD.VAL** in the **Value** field for the *LOAD* parameter. This connects the **LOAD** element characteristic output to the machine's *LOAD* input parameter.

   d. In the **Default Outputs** section, ensure that **N** and **IA** are checked to enable them to be used later when creating reports.

   e. Click **OK** to close the dialog box.

Defining the PI Controller

The components that have been placed and connected still have their default parameter values. You will now assign proper values for the components that comprise the PI Controller. Use the simulation model figure "Current and Speed Controlled DC Motor" on page 5-1 as a guide.

1. Define the parameters of the first **gain block.**
   a. Double-click the **Gain** block symbol to open its **Parameters** dialog box.
   b. On the **Parameters** tab, change the **Name** to **N**.
   c. Type **-16.6667** for the *KP* (Proportional Gain) parameter **Value** and select **m** for its Units.
   d. Change the parameter **Value** to **DCM.N*1**. This "connects" the gain block input to the DC machine's rotor speed output parameter with a scaling factor of 1. On the Output/Display tab, uncheck **Show Pin** for the *INPUT* parameter.
   e. Click **OK** to close the dialog box.
2. Define the parameters of the constant value block.
   a. Double-click the **CONST** block symbol to open its **Parameters** dialog box.
   b. Change the **Name** to **N_REF** and the **Value** to **16.6667**. Ensure that **Use System Sample Time** is checked and that **Block Output Signal** is unchecked.
   c. Click **OK** to apply the changes.
3. Define the parameters of the summation of the N gain block and the N_REF constant block.
   a. Double-click the summation symbol to the right of the **N** gain block to open its **Parameters** dialog box.
   b. Ensure that **Use System Sample Time** is checked and that **Block Output Signal** is not checked.
   c. Click **OK** to apply the changes.
4. Define the parameters of the second gain block.
   a. Double-click the **GAIN** block symbol to open its **Parameters** dialog box.
   b. Change the **Name** to **P_GAIN** and the Proportional Gain (**KP**) to **50**.
   c. Click **OK** to apply the changes.
5. Define the parameters of the integrator block.
   a. Double-click the **I** block symbol to open its **Parameters** dialog box.
   b. Change the **Name** to **I_GAIN** and the Integral Gain (**KI**) to **20**. The specification of "**0**" for upper limit (**UL**) and lower limit (**LL**) means that there is no limitation.
   c. Click **OK** to apply the changes.
6. Define the parameters of the summation of **P_GAIN** and **I_GAIN**.
   a. Double-click the summation symbol to the right of the **P_GAIN** block to open its **Parameters** dialog box.

b. Ensure that **Use System Sample Time** is checked and that **Block Output Signal** is not checked.

c. Click **OK** to apply the changes.

7. **Define the parameters of the**limiter block.

   a. Double-click the limiter symbol to open its **Parameters** dialog box.

   b. Change the **Name** to **LIMITER** and the Upper Limit of Output Signal (**UL**) to **20**.

   c. Click **OK** to apply the changes.

8. Define the parameters of thesummation of machine current and limiter value.

   a. Double-click the summation symbol to the right of the **LIMITER** block to open its **Parameters** dialog box.

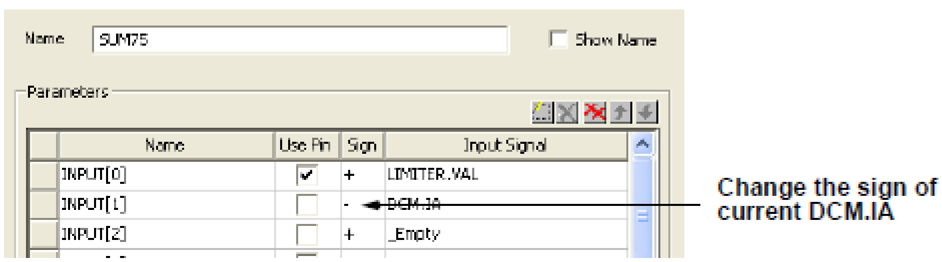
   b. For **INPUT[1]**, un-check **Use Pin**.

   c. Enter **DCM.IA** in the **Input Signal** field to link the DC machine armature current to **INPUT[1]**.

   d. Click in the **Sign** column and select "**–**" from the list. The sign is applied to the **DCM.IA** input signal.

   e. Ensure that **Use System Sample Time** is checked and that **Block Output Signal** is not checked.

   f. Click **OK** to apply the changes.



9. Modify the parameters of thehysteresis block.

   a. Right-click the hysteresis block symbol and choose Activate on the context menu.

   b. Double-click the hysteresis block symbol to open its **Parameters** dialog box.

   c. Define the parameters as follows:

- **THRES1 (Threshold T1):-2.5**
- **THRES2 (Threshold T2):2.5**
- **VAL1 (Value A1):0**
- **VAL2** (Value A2): **1**
- **Y0 (Initial Value):1**
- **TS (Sample Time):5**with units **us**

   d. Click **OK** to apply the changes.

10. Modify the parameters of transistor TR1.
   a. Right-click the TR1 symbol to open its **Parameters** dialog box.
   b. On the **Parameters** tab, clear the **Control Signal** field, then check **Use Pin** to reconnect the transistor's control pin to the output of the hysteresis block.
   c. Click **OK** to close the dialog box.

All parameters of the PI controller should now have the correct values. The table below lists all components of the PI controller and their parameter values.

Name	**Output To**	**Parameters**
N	SUM1	KP (Proportional Gain)=**-16.6667m** TS (Sample Time)=System (**0**) INPUT (Input Signal)=**DCM.N**
N_REF	SUM1	Value=**16.6667** Sample Time=System
SUM1	P_GAIN I_GAIN	Sample Time=System Input[0]=**N.VAL** INPUT[1]=**N_REF.VAL**
P_GAIN	SUM2	KP (Proportional Gain)=**50** TS (Sample Time)=System (**0**) INPUT (Input Signal)=**SUM1.VAL**
I_GAIN	SUM2	KI (Integral Gain)=**20** Y0 (Initial Value)=**0** UL (Upper Limit)=**0** LL (Lower Limit)=**0** TS (Sample Time)=System (**0**) INPUT (Input Signal)=**SUM1.VAL**
SUM2	LIMITER	Sample Time=System Input[0]=**P_GAIN.VAL** INPUT[1]=**I_GAIN.VAL**
LIMITER	SUM3	UL (Upper Limit)=**20** LL (Lower Limit)=**0**

Name	Output To	Parameters
		TS (Sample Time)=System (**0**) INPUT (Input Signal)=**SUM2.VAL**
SUM3	CONTR_OUT	Sample Time=System Input[0]=**LIMITER.VAL** INPUT[1]=**-DCM.IA**
CONTR_OUT	TR1.CTRL	THRES1 (Threshold T1)=**-2.5** THRES2 (Threshold T2)= **2.5** VAL1 (Value A1)=**0** VAL2 (Value A2)=**1** Y0 (Initial Value)=**1** TS (Sample Time)=**5us** INPUT (Input Signal)=**SUM3.VAL**
LOAD	DCM.LOAD	Interpolation=Without TPERIO (Period)= **0.1** PHASE (Phase)= **0** PERIO (Periodical)= **Yes** TDELAY (Initial Delay)= **0** CHARACTERISTIC DATASET= $Load_Torque x=0 y=0 x=0.075 y=10 x=0.1 y=10

Starting Simulation

Start the simulation with the **Simplorer Circuit>Analyze** menu command or the right click on the **Analysis** icon on the **Project Manager** window and select **Analyze** from the short-cut men. The simulation model is compiled and calculated.

Adding a Rectangular Plot (PI Controller)

1. **Add a Rectangular Plot.**
   a. In the Project Manager tree, right-click the Results icon and select **Create Standard Report>Rectangular Plot**.

b. A **Report** dialog box opens.

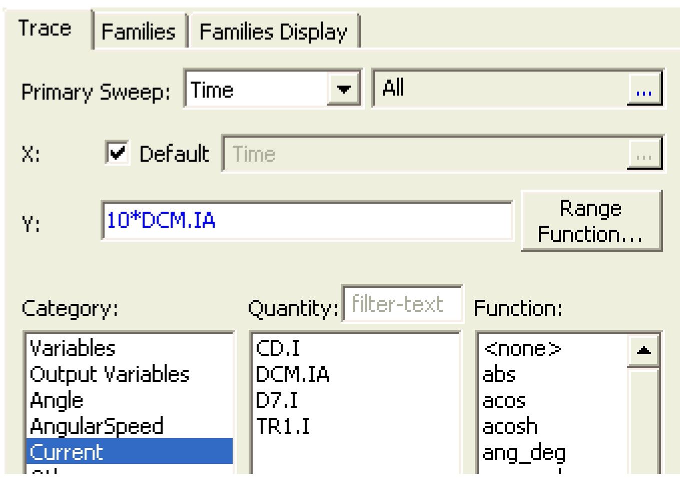
2. **Define the model outputs *DCM.IA*, *DCM.N*, *LOAD.VAL*.**

a. Under **Category**, select **All**.

b. In the Quantity list, select *DCM.N* and click **New Report** to create the report.

c. Without closing the **New Report** dialog, select *DCM.IA* in the **Quantity** list. **DCM.IA** appears in the **Y** trace field. Its blue color signifies that it is a valid expression.

d. Simplorer allows you to apply various functions such as a scale factor to the selected quantity. Place the cursor at the beginning of **DCM.IA** in the **Y** field and type **10*** to multiply **DCM.IA** by a factor of ten. As you are entering the factor, the color of the expression may change to red at times indicating that the expression as shown at that point is invalid. The finished expression should be: **10*DCM.IA** and be blue indicating that it is valid.

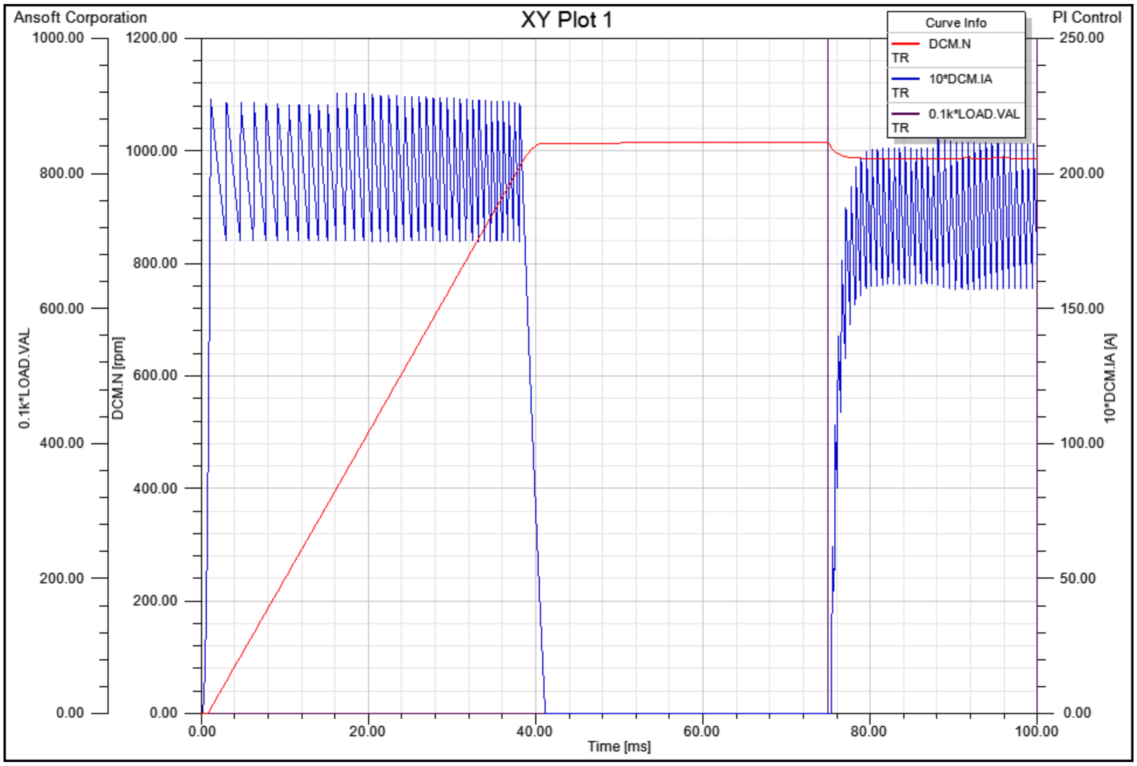


e. Click **Add Trace** to add the new scaled trace to the report.

f. Next select *LOAD.VAL* and apply a scale factor of **0.1k** to it. The finished Y expression should be **0.1k*LOAD.VAL**.

g. Click **Add Trace** to add the trace to the report.

h. Click **Close** to close the report dialog box. The plot should look similar to the following:



Adjusting Plot Properties

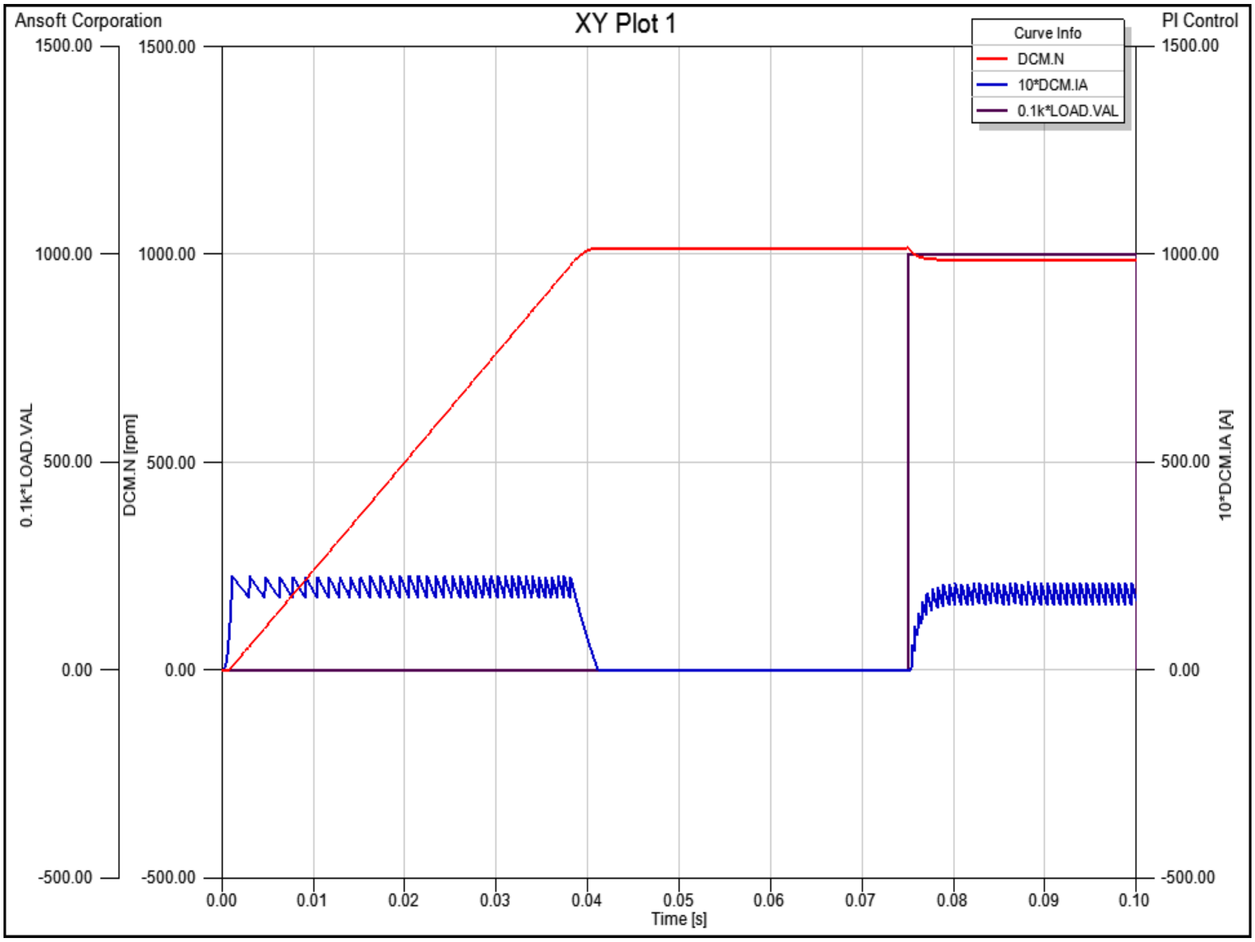
Many of a plot's properties can be adjusted. In the following procedure you will make changes to various properties of the X- and Y- axes.

1. **Change the Y-axis Properties.**
   a. Position the cursor over any part of the Y-axis labeled **0.1k*LOAD.VAL** on the plot. The arrow will turn green when hovering over a selectable area of the axis. Click the axis to select it for editing. The axis scale will become bold when selected. The axis name and scale numbers will also be underlined.
   b. In the Properties window, select the Scaling tab.
   c. Check the **Specify Min** check box and type **-0.5k** in the **Min Value** field below it.
   d. Check the **Specify Max** check box and type **1.5k** for the **Max Value**.
   e. Check the **Specify Spacing** box and change the **Spacing** value to **500**. This setting controls the interval between major "tick" divisions on the axis scale.
   f. Change **Minor Tick Divs** to **1**. This setting controls the number of divisions between major divisions.
   g. Repeat steps **a** through **f** for the Y- axis labeled **DCM.N**.

   h. For **10*DCM.IA** Y-axis, first clear the **Auto Units** check box and set **Units** to ***A***, then repeat steps **a** through **f**.

2. **Change the X-axis Properties.**
   a. Select the X-axis for editing.
   b. On the Scaling tab, uncheck **Auto Units** and select a **Units** setting of **s** (seconds).
   c. Set **Min** value to **0**, **Max** value to **0.1**, **Spacing** to **0.01**, and **Minor Tick Divs** to **1**.

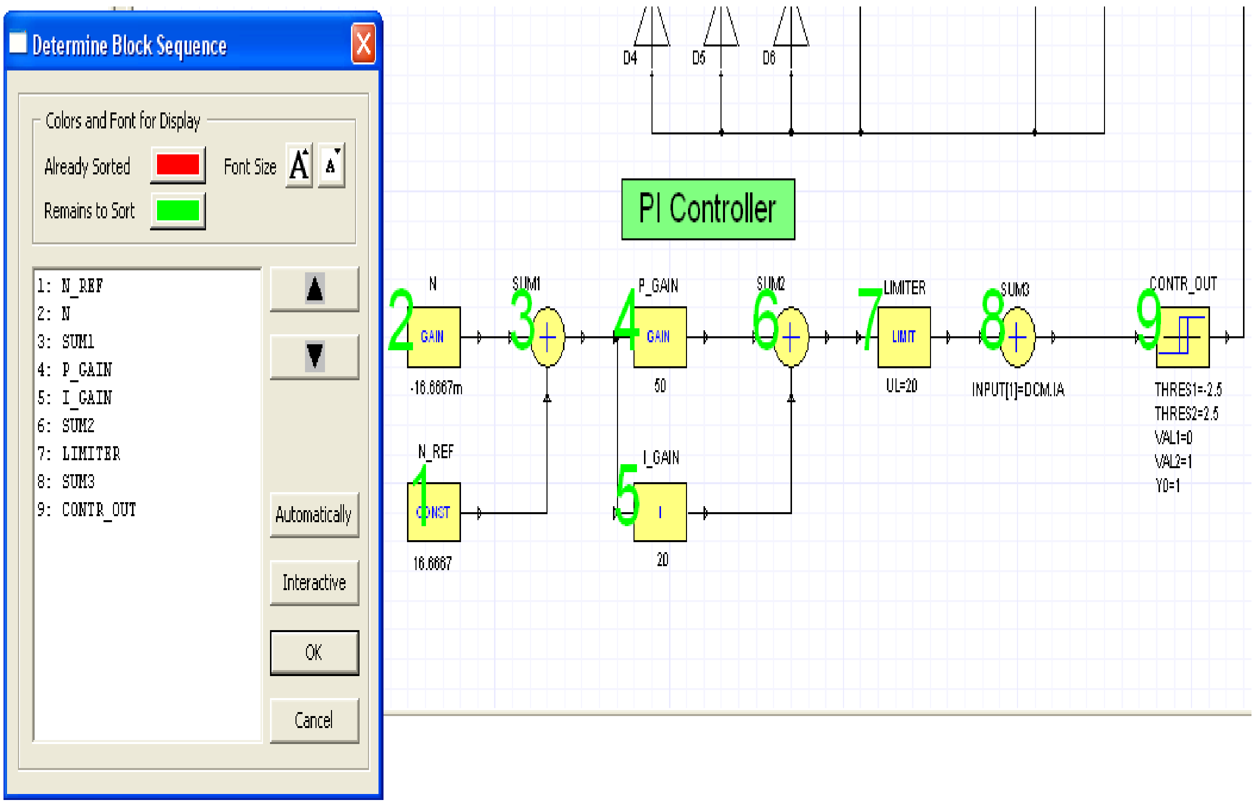
The finished plot should look similar to the following:



Refer to the online help for additional information on adjusting plot properties.

Checking the Block Sequence

The sequence of block computation can significantly influence the simulation result. You can change the block sequence with the **Schematic>Sort Components** menu command.



Using Automatic Block Sorting

When **Block Sequence** sorting is determined **Automatically**, the blocks are sorted according to their signal direction after simulation begins. Automatic block sequence sorting is the default mode used for simulation.

Using Manual Block Sorting

To sort blocks manually, you can select block items in the **Determine Block Sequence** dialog box list and use the up and down arrow buttons to change the sort order.

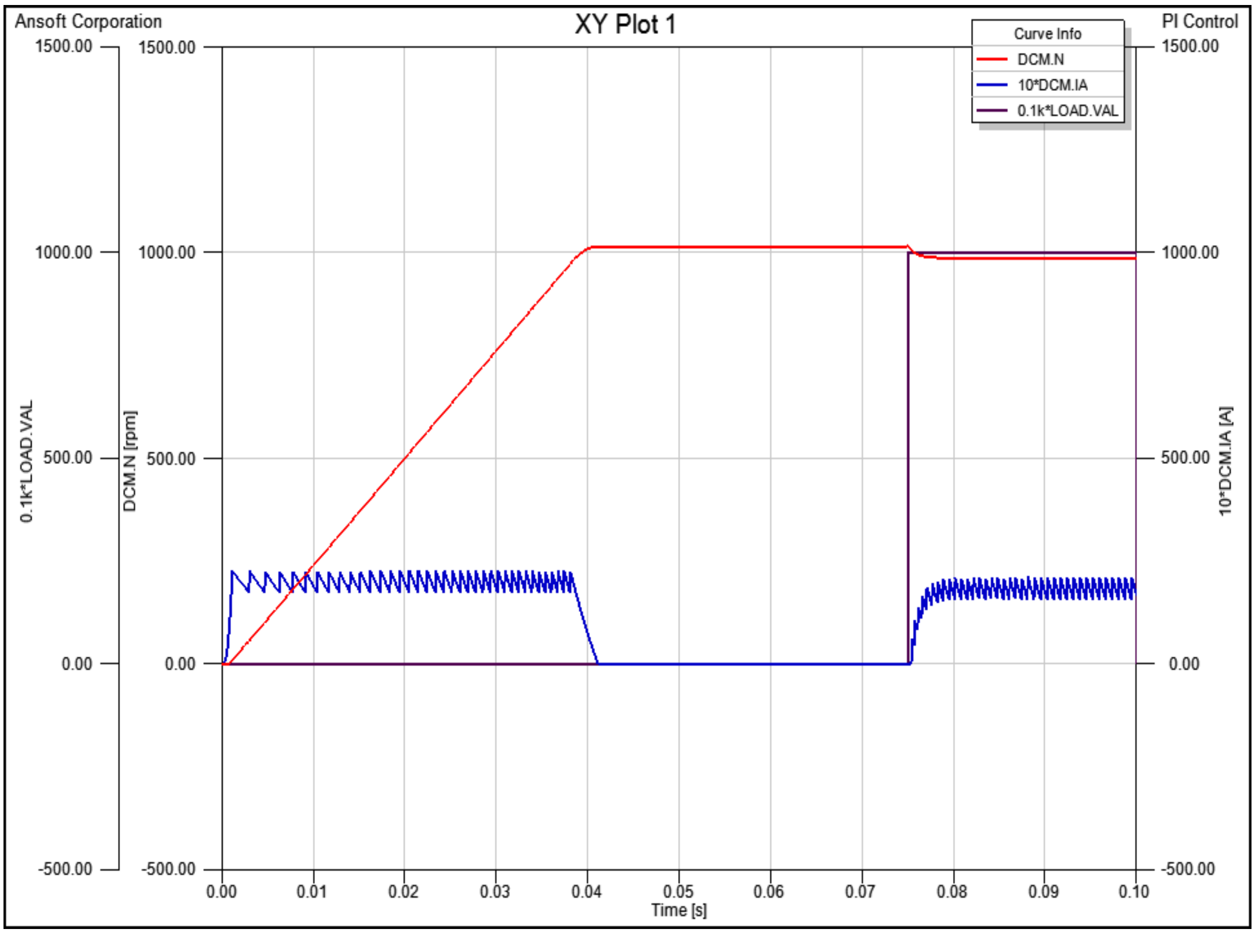
You can also click the **Interactive** button which closes the dialog box and displays the current sequence in digits next to the blocks. You can then change the sequence by clicking the blocks in the desired sequence on the sheet or by arranging the block list in a new sequence in the dialog.

Rerun the Simulation (PI Controller)

After changing the Block Sequence, rerun the simulation with the **Simplorer Circuit>Analyze** menu command. The simulation model is recompiled and recalculated.

Simulation Results (PI Controller)

The plot displays the simulation results for the machine armature current (**DCM.IA**), speed (**DCM.N**), and the **LOAD**. Depending on the armature current, the PI controller adjusts the switching behavior of the chopper transistor. The no-load starting speed for the DC motor approaches 1000 rpm.



6 - Using VHDL-AMS Components for Modeling

This example describes the modeling of a PI controlled DC motor system using a VHDL-AMS component for the DC machine Simplorer functions you will use in this example:

- Basic functions (selecting, placing, arranging, and connecting components)
- Modeling with VHDL-AMS components
- Using plots for displaying simulation results

VHDL-AMS Components

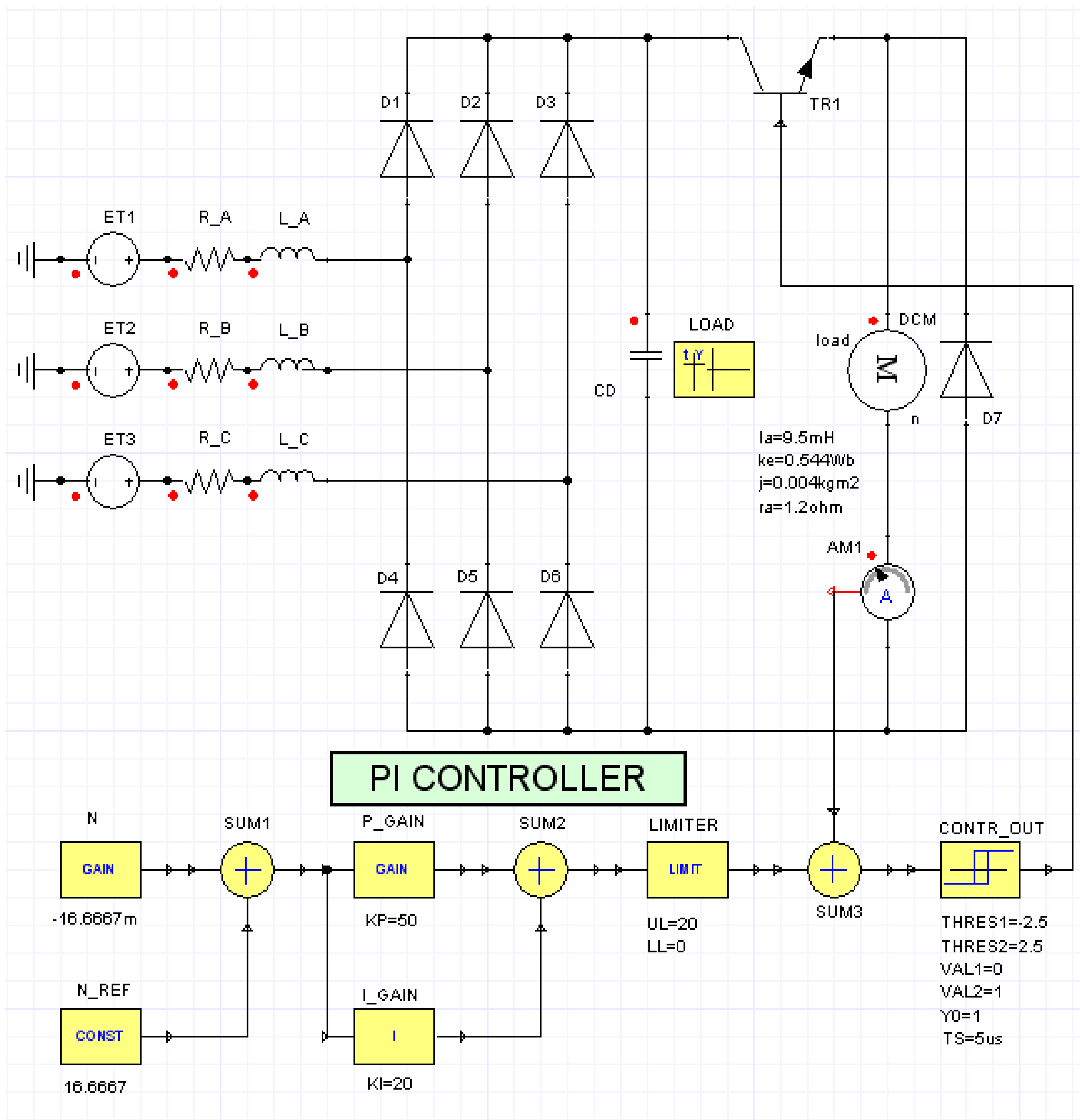
The ***Basic Elements VHDLAMS*** library provides the user with the most common basic functionality, circuit components and block elements as VHDL-AMS models developed according to IEEE 1076.1 (VHDL Analog and Mixed Signal Extensions Standard).

The functionality of the VHDL-AMS models is a subset of that available in the equivalent Simplorer SML (Simplorer Modeling Language) models provided in the ***Basic Elements*** library. VHDL-AMS models can be used in parallel with SML models. VHDL-AMS block and circuit models are simulated by the analog solver; while SML block elements are solved by a separate block diagram simulator. Consequently, unlike SML models. VHDL-AMS models will not display step delays along the simulator backplane. Numerical values of the VHDL-AMS data type *generic* are defined at *t = 0* and remain unchanged for the remainder of the simulation.

You can view the inputs and outputs of a VHDL-AMS model using the appropriate analog or digital report. You can also view internal values used within the architecture of a model on reports and use them as variables between models. However, if the VHDL-AMS design needs to be exported to other simulators then internal values of the model should not be used outside the model.

Modify the PI Controller Design

The figure below shows the Schematic sheet of the simulation model and corresponding values of the three-phase power supply, the rectifier bridge with static diodes and their characteristics, the smoothing capacitor, and the VHDL-AMS machine component, **DCM**.



Save the Project with a New Name

1. Choose **File>Save As**.
2. Enter a new file name, and click **OK**.

Delete the DC Machine Component

Delete the DC machine component that will be replaced with a VHDL-AMS machine component.

1. Select the DC machine component by clicking it.
2. Press the **Delete** key to delete the component.

Placing and Arranging the New VHDL-AMS Components on the Sheet

1. In the **Component Libraries** window, select the component *AM: Ammeter* from the Favorites folder and drag it onto the sheet.
2. Open the *Basic Elements VHDLAMS>Circuit>Electrical Machines* folder. Select the component *dcmp: DC Permanent Magnet Excitation* and drag it onto the sheet.
3. Arrange the components in appropriate positions. See the simulation model figure in"Modify the PI Controller Design" on page 6-1.

Connecting the New VHDL-AMS Components

1. Connect the VHDL-AMS DC machine and ammeter component pins as indicated on "Modify the PI Controller Design" on page 6-1.

Some pins are not available yet. Leave these connections out in this step. The connections are created later.

Defining VHDL-AMS DC Machine Values

1. Open the VHDL-AMS DC machine **Properties** dialog box.
2. On the **Parameter Values** tab, change **InstanceName** to **DCM**, **la** (Armature Inductance) to ***9.5 mH***, **ke** (Back EMF Constant) to ***0.544 Wb***, and **j** (Moment of Inertia) to ***0.004*** $\boldsymbol{kgm^2}$.
3. On the **Quantities** tab, change **ra** (Armature Resistance) to ***1.2 ohm***. For the **load** (Load Torque) property, type ***LOAD.VAL*** in its **Value** field to “connect” it to the **LOAD** characteristic component.

4. On the **Property Displays** tab add the following property and visibility settings:

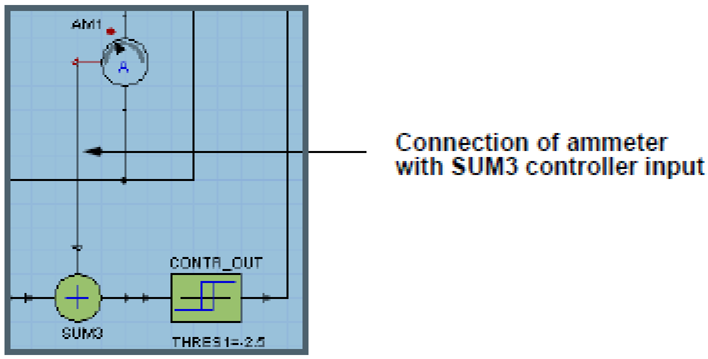
Name	Visibility
InstanceName	Evaluated Value
n	Name
load	Name
la	Both
ke	Both
j	Both
ra	Both

5. Click **OK** to apply the changes.
6. Open the **Properties** dialog box of **Gain** block **N**. Override the default **INPUTValue** by entering ***DCM.n*** to connect it to the VHDL-AMS motor speed property.

Defining Connections for Machine Current

1. Make the ammeter *Current* parameter pin available for connecting with the SUM3 component input pin.
   a. Double-click the ammeter symbol to open its **Parameters** dialog box.
   b. On the **Parameters** tab, change its **Name** to **AM1**.
   c. On the **Output/Display** tab, check **Show Pin** for the **I** (Current) property.
   d. Click **OK** to apply the changes.
2. Make the **SUM3** summation input pin available for connecting with the ammeter **Current** pin.
   a. Double-click the **SUM3** component to open its **Parameters** dialog box. If a message dialog displays informing you that **DCM.IA** is not a defined variable, just click **OK** to continue.
   b. On the **Parameters** tab, for **INPUT[1]**, delete **DCM.IA** in the Input Signal field and check **Use Pin**.
   c. Ensure that **Use System Sample Time** is checked and that **Block Output Signal** is unchecked.
   d. On the **Output/Display** tab, change the **Visibility** of the **INPUT[1]** entry to **None**.
   e. Click **OK** to apply the changes.

3. Connect the input pin of the **SUM3** with the current pin of the ammeter.



Defining Simulation Parameters

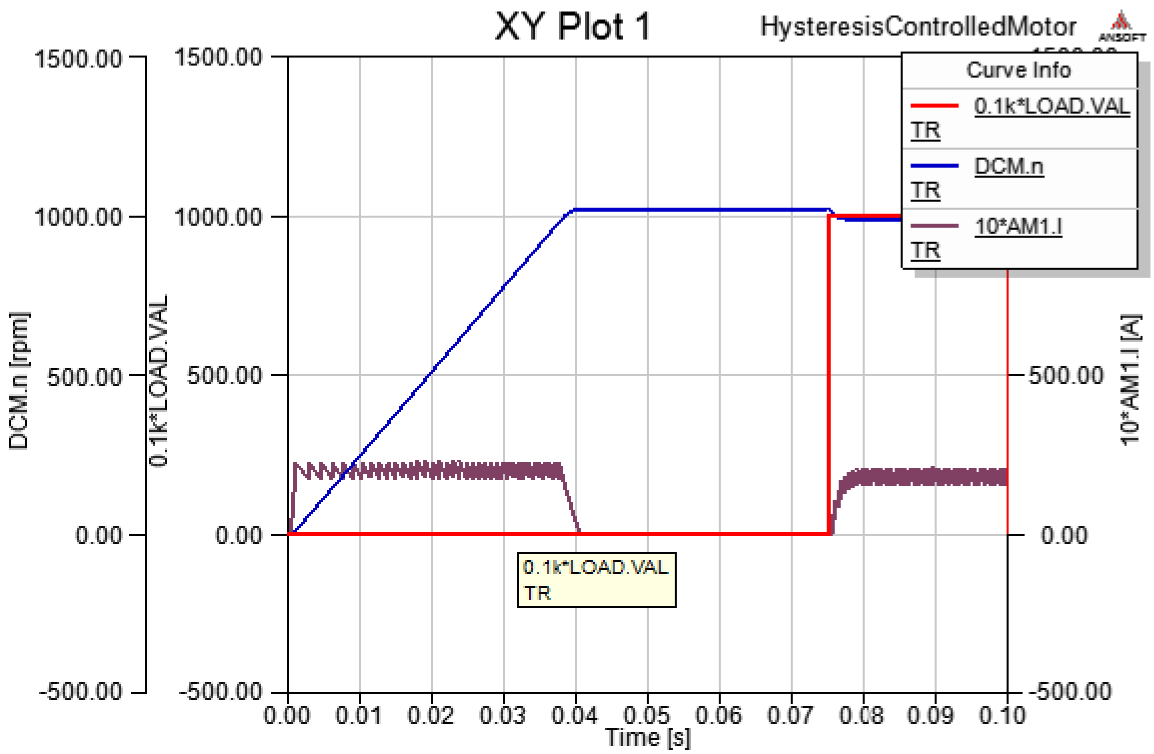
Simulation parameters control the simulation process. The values chosen for a simulation determine the success, and affect the quality, of a simulation result.

1. Choose **Simplorer Circuit>Add Solution Setup>Transient** to define the simulation parameters.
2. Change the default values for simulation **End Time** from 40ms to 0.5ms, for **Min Time Step** from 10us to 1ns, and for **Max Time Step** from 1ms to 1us. Click **OK** to apply the changes.

Analyze and Display Simulation Results (VHDL-AMS)

1. Select **Simplorer Circuit>Output Dialog** and check **DCM.n** and **AM1.I** to add them to the **Defined Outputs**. Click OK to close the **Output** dialog box
2. Start the simulation with the **Simplorer Circuit>Analyze** menu command.
3. In the **Project Manager** tree, expand the existing XY Plot and note that the **DCM.N** and **10*DCM.IA** icons show that these traces are no longer valid because the variable names to which they were linked have been removed.
4. Right-click **DCM.N**, select **Modify Report**, and set the **Y** trace to **DCM.n**.
5. Similarly modify **10*DCM.IA** to **10*AM1.I**. Click **OK** to apply the changes.

6. Double-click the XY Plot icon to display the plot which should look similar to the one below.



7 - Variants of PWM Modeling

A frequently used component in power electronic applications is a Pulse-Width Modulation controller. This device works using a constant frequency, variable impulse width, and a pulse duty factor.

Simplorer offers several ways to generate a model of this kind of device. In addition to the implementation of the internal circuit structure, behavioral models are increasingly important because they provide high simulation speed combined with sufficient accuracy.

The following example contains PWM controllers designed using several distinct Simplorer modeling methods. All methods are included in one sheet so that you can easily compare the differences between them.

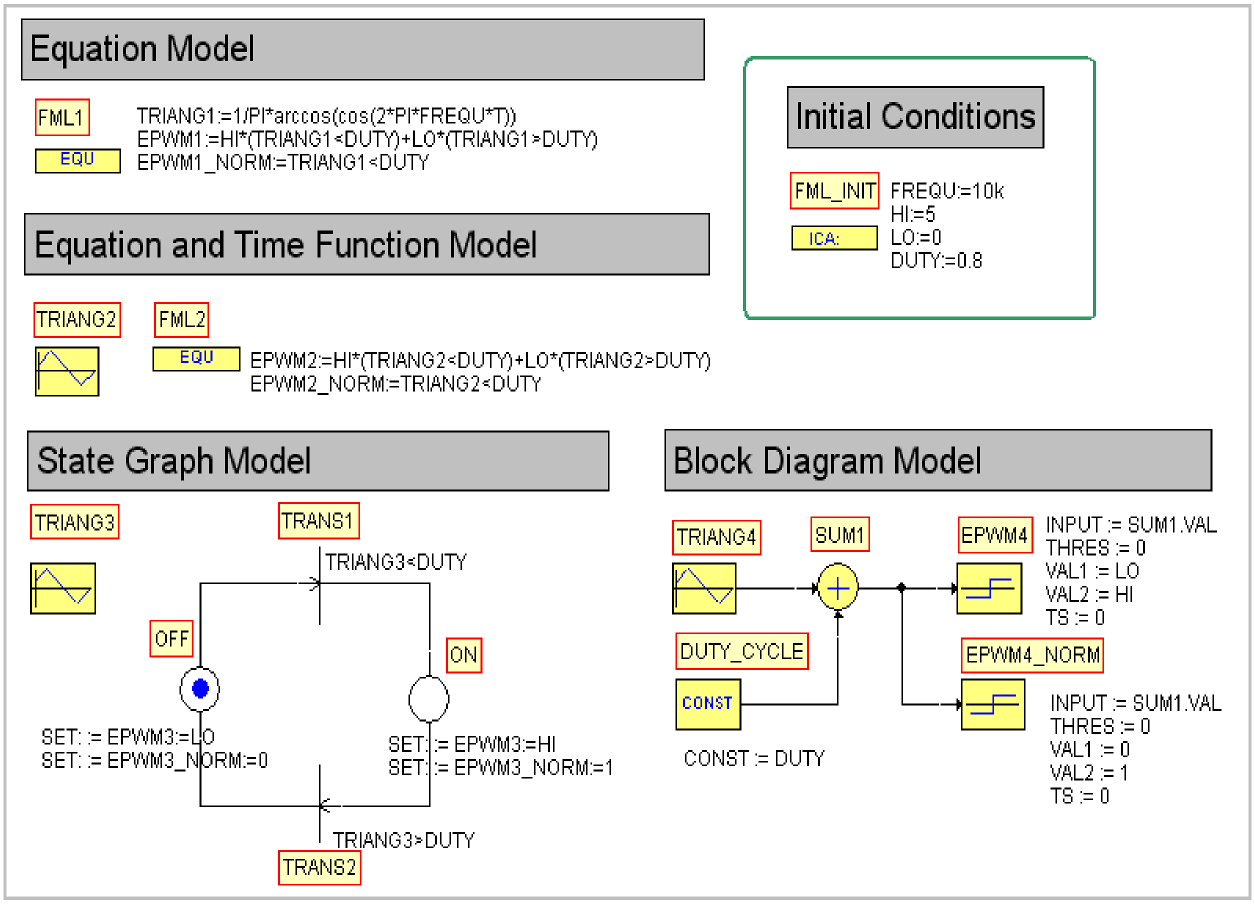
Simplorer functions you will use in this example:

- Basic functions (selecting, placing, arranging, and connecting components)
- Modeling with block components
- Using Display Elements for displaying simulation results
- Using equations
- Defining block Sequence

PWM Modeling Overview

The figure below shows the Schematic sheet of the PWM controllers with the corresponding values of components. The sheet contains the following modeling variants:

- Equation
- Equation and Time Function
- State Graph
- Block Diagram



Create a New Project for the PWM Models

Create a new Project and Schematic sheet for the Pulse-Width Modulation models that you will be designing. Refer to Creating a New Project for details.

Setting Initial Conditions

The initial values defined in this section are used by all modeling examples in this chapter. These values are set only once at the start of simulation.

1. In the Simplorer model library ***Basic Elements >*** *Tools* > *Equations* in the Component Libraries window, select the component *Initial Values* and drag it onto the sheet.
2. Double-click the **ICA** symbol to open the FML_INIT parameters dialog box. Create a new entry using the Add button . Click in the **Equation** field and enter *FREQU:=10k*.
3. Repeat step 2 to define the remaining initial values:
   - *HI:=5*
   - *LO:=0*

- *DUTY:=0.8*

4. Click **OK** to apply the changes.

PWM Modeling Using Equations

This first example describes the modeling of a PWM controller with the use of equations only. You can separate the description into the following steps:

- Triangle function
- Real value output
- Normalized value output

As you build this model, note that ***Time*** and ***PI*** are used in the expressions describing the simulation model.

- The Simplorer simulator uses intrinsic variables for internal computation. One of these intrinsic variables is the simulation time: ***Time***. You can use this read-only variable in expressions.
- The simulator also provides some natural and mathematical constants which can be used in mathematical expressions within component dialogs. One such constant is ***PI***.
- In addition, some standard mathematical functions, such as ***cos***, are used in the expressions. *Refer to the online help for information on Simplorer's standard functions*.

1. Choose the Simplorer ***Basic Elements>Tools>Equations*** library in the Component Libraries window. Select the *FML:Equation* component and drag it onto the sheet.
2. Double-click the **EQU** symbol to open the **FML** dialog box. Create a new **Equation** entry with the Add button. Click in the **Equation** field and enter: *TRIANG1:=1/PI*acos(cos(2*PI*FREQU*Time))*
3. Repeat step 2 and enter the following equations:
   - *EPWM1:=HI*(TRIANG1<DUTY)+LO*(TRIANG1>DUTY)*
   - *EPWM1_NORM:=TRIANG1<DUTY*
4. Ensure that the **Calculation Sequence** for all equations remains at the default **Before Analog Solver** setting. Click **OK** to apply the changes.

Defining Simulation Parameters

Simulation parameters control the simulation process. The values chosen for a simulation determine the success, and affect the quality, of a simulation result.

1. Choose **Simplorer Circuit>Add Solution Setup>Transient** to define the simulation parameters.
2. Change the default values for simulation **End Time** from 40ms to 0.5ms, for **Min Time Step** from 10us to 1ns, and for **Max Time Step** from 1ms to 1us. Click **OK** to apply the changes.

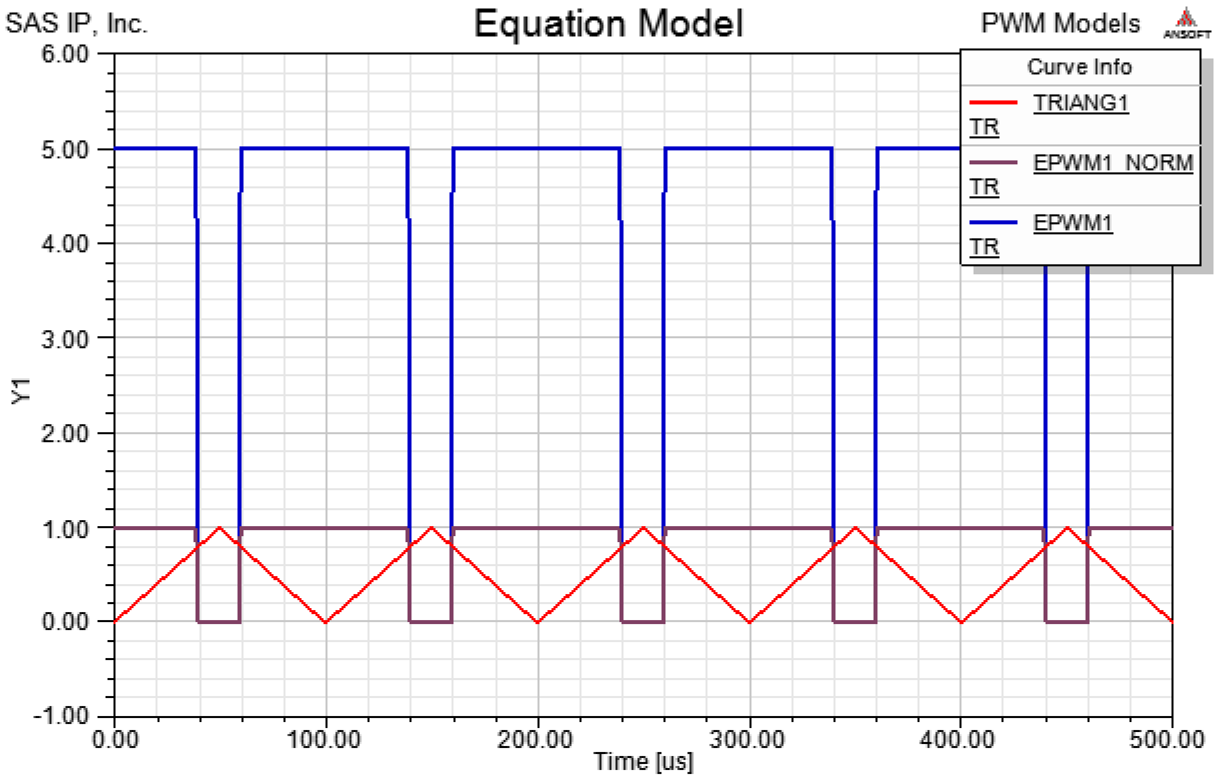
Displaying Simulation Results with Reports

During a simulation several types of data are generated. These data can be displayed and printed by means of Simplorer reports, or can be saved in files. Simplorer reports can be either stand-alone, or can be placed directly on the sheet like components.

1. To add an on-sheet plot, choose **Draw>Report>Rectangular Plot**. A hollow rectangular box appears with the cursor at its center. Move the rectangle and click in the schematic window to place the plot center. The cursor moves to the lower right corner of the rectangle. Drag the cursor to resize the rectangle as needed, then click again to finish placing the plot.
2. Choose **Simplorer Circuit>Output Dialog** to open the **Output** dialog box. Define outputs for ***TRIANG1***, ***EPWM1***, and ***EPWM1_NORM*** by checking the output boxes of these quantities. Click **OK** to apply the changes.
3. Double-click the on-sheet report to open the **Report** dialog box. Ensure that the **Solution** is ***TR***, the **Domain** and **Primary Sweep** are ***Time***, and that the **X** channel is set to **Default** via the checkbox.
4. Click ***All*** in the **Category** list. In the **Quantity** list **Ctrl**+click ***TRIANG1***, ***EPWM1***, and ***EPWM1_NORM***. Click **Add Trace** to add the traces to the plot, then **Close** to exit the dialog box.
5. Name the plot ***Equation Models*** by right-clicking its current name under Results in the Project Manager and selecting **Rename**. Press **Enter** to apply the change.
6. Save the project by choosing **File>Save As** and entering a name.
7. Start the simulation with the **Simplorer Circuit>Analyze** menu command. The simulation model is compiled and calculated.

Simulation Results

The plot displays the simulation results for the triangle function (**TRIANG1**), actual PWM signal (**EPWM1**) and normalized PWM signal (**PWM1_NORM**). The value of **DUTY** determines the pulse width.



PWM Modeling with Equations and Time Function

The second example models a PWM controller using both *equations* and a *time function*. The same initial value assignments set in the first example are also used in this example. It is unnecessary to connect the components directly because references will be created using component and parameter names.

1. In the ***Basic Elements>Tools>Time Functions*** library folder, select the *TRIANG:Triangular Wave* component and drag it onto the sheet. Select the folder ***Equations*** and drag an *FML:Equation* component onto the sheet.
2. Right-click the *TRIANG* symbol to open the **Parameters** dialog box. On the **Parameters** tab, change the **Name** to *TRIANG2*. Set the Amplitude (AMPL) to *0.5*, Frequency (FREQ) to *FREQU*, and Offset (OFF) to *0.5*.
3. Select Value (VAL) and clear the **Show Pin** box. Click **OK** to apply the changes.

Note	You can leave the Value (VAL) output pin visible; however the pin is not used in this example.

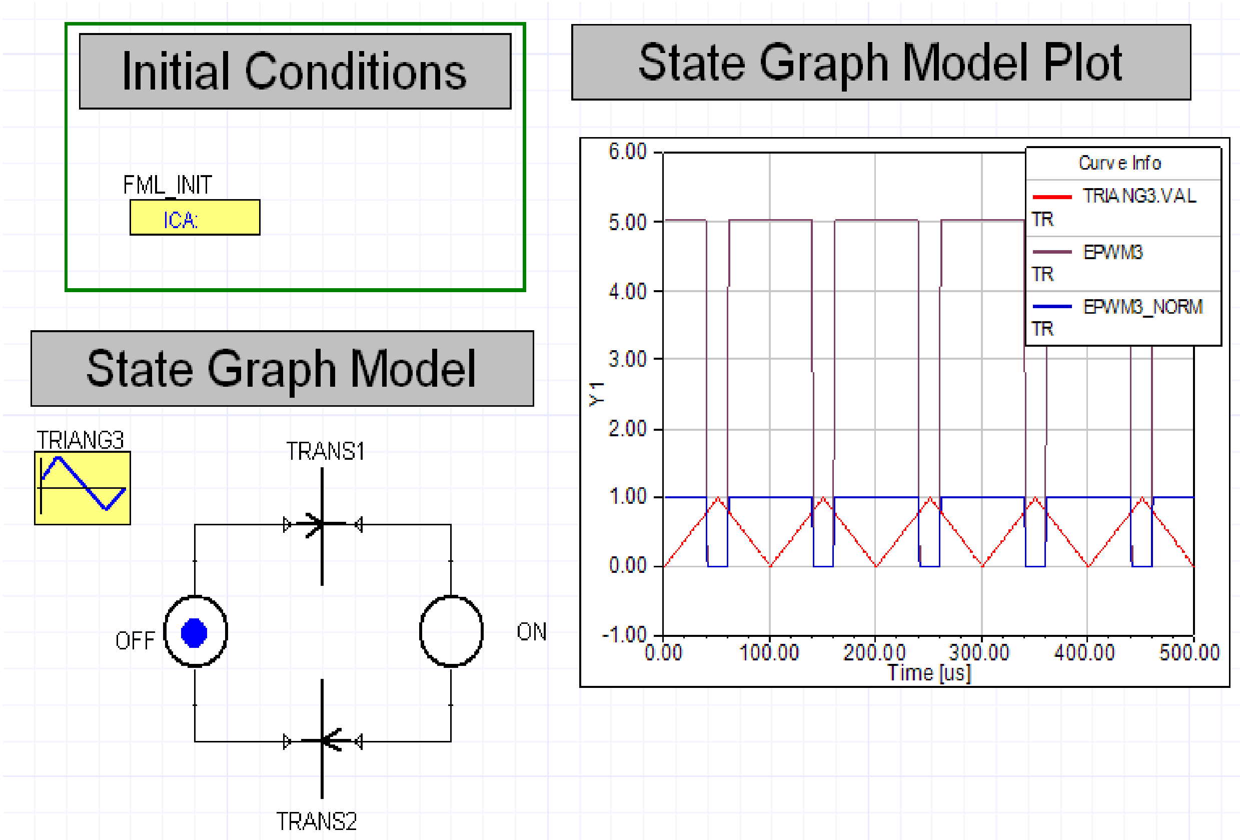
4. Double-click the EQU symbol to open the FML **Parameters** dialog box. Create a new **Equation** entry with the Add button. In the **Equation** field enter: *EPWM2:=HI*(TRIANG2<DUTY)+LO*(TRIANG2>DUTY)*.
5. Repeat the above step to add the following equation: *EPWM2_NORM:=TRIANG2<DUTY*

   Click **OK** to apply the changes.
6. Add an on-sheet Rectangular Plot as in the first example. However, for this plot, define outputs and add traces for the ***TRIANG2.VAL***, ***EPWM2***, and ***EPWM2_NORM*** signals.
7. Name the plot ***Equation and Time Function Model***; and save the project using **File>Save As**.
8. Start the simulation with the **Simplorer Circuit>Analyze** menu command. The simulation model is compiled and calculated.

   The resulting plot should be identical to the one plotted in the first example.

PWM Modeling with State Graph Components

This example models the PWM controller using state graph components. The same initial value assignments set in the first two examples are also used in this example. Simplorer's state graph simulator, which is based on the Petri Net theory, allows you to model event-driven, discontinuous processes. This modeling method divides a system into significant states and events, then transitions from one state to another.



Place and Arrange the Components on the Sheet

1. In the **Basic Elements** library, open the **States** folder and drag a **State_11** component onto the sheet. Change its name to ***OFF*** and rotate it to match the figure above. Drag a second **State_11** instance onto the sheet and label it ***ON***. Next, drag two **TRANS** components onto the sheet and name them ***TRANS1*** and ***TRANS2***. Make sure to rotate them so that their arrows match the figure.
2. Drag a **TRIANG** component from the **Basic Elements>Tools>Time Functions** folder onto the sheet. Change the component name to ***TRIANG3***.
3. Connect the components as shown in the figure.

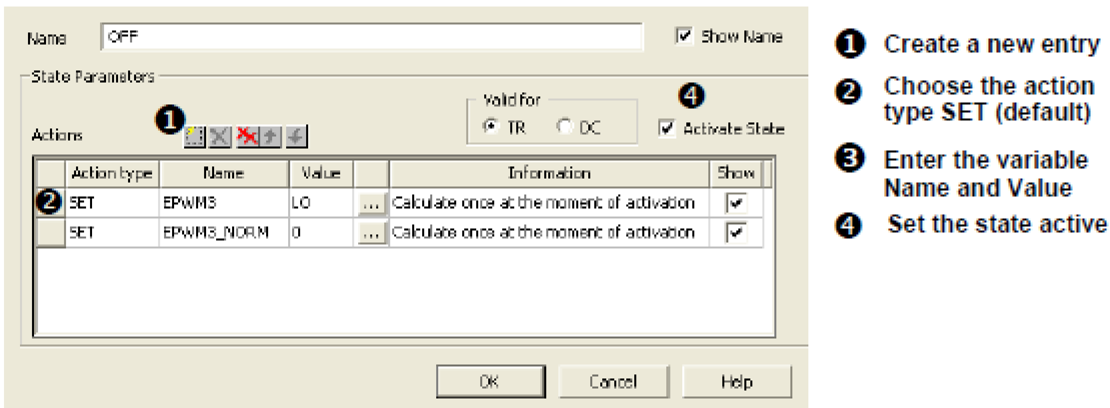
Define Component Properties

A process sequence can be considered as a sequence of states. The current state is called *active*. Switching an activity from the current state to its successor state is called an *event*. An event occurs only if:

- All previous states are active.
- All following states are inactive.
- The transfer condition — in the form of a logical expression — is true.

At the beginning of the simulation, one state ***must*** be defined as *active*.

1. Double-click the ***OFF*** state symbol to open its **Parameters** dialog box. Create a new **SET** Action type entry with the **Add** button. Enter *EPWM3* in the **Name** field and *LO* in the **Value** field. This entry means that the variable *EPWM3* is set to the value *LO* if the state is *active*. Create another **SET** entry and enter *EPWM3_NORM* in the **Name** field and *0* in the **Value** field. Check the **Activate State** box to set the state *active* at the beginning of simulation. Click **OK** to apply the changes. A blue circle in the symbol indicates the state is *active*.



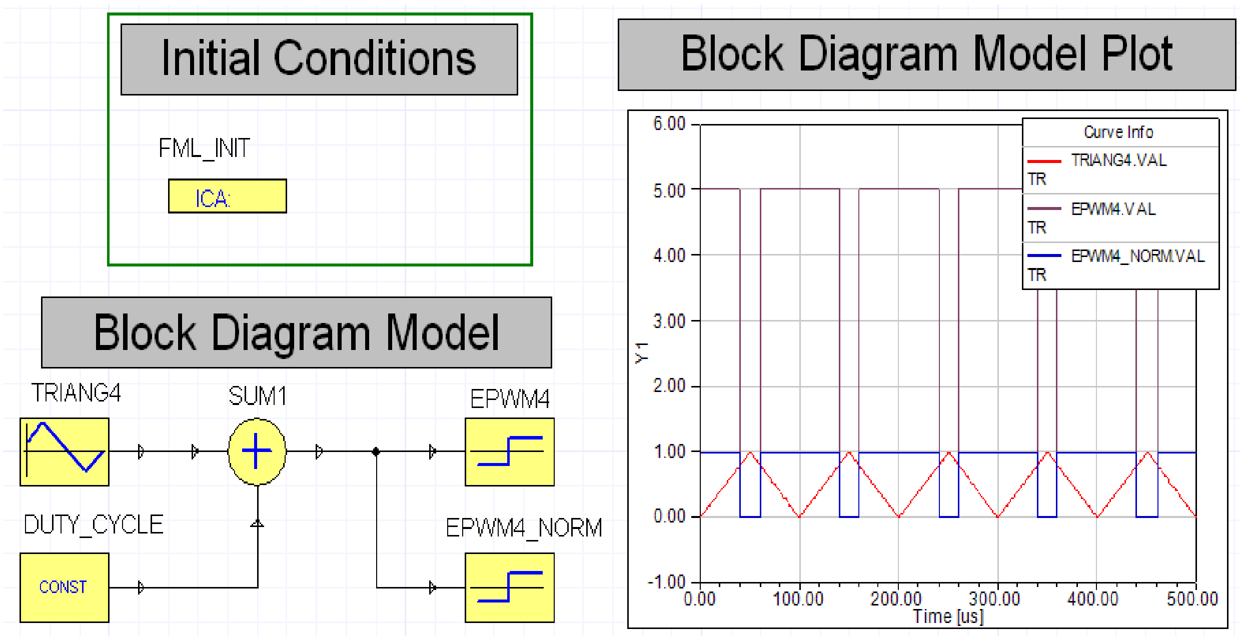
2. Similarly define the parameters of the state *ON* component in its **Parameters** dialog box**.** Create a new **SET** entry and enter *EPWM3* in the **Name** field and *HI* in the **Value** field. Create another **SET** entry and enter *EPWM3_NORM* in the **Name** field and *1* in the **Value** field. Do not check **Activate State**. Click **OK** to apply the changes.
3. Double-click the **TRANS1** symbol and enter *TRIANG3<DUTY* in the **Condition for transition** field. This entry means that the condition becomes true if the value of the triangular function, **TRIANG3**, is lower than the *DUTY* value as defined in the initial value condition. Click **OK** to apply the changes. Similarly, for **TRANS2,** enter *TRIANG3>DUTY* in the **Condition for transition** field. In this case, the condition becomes true if the value of the triangular function is lower than the *DUTY* value. Click **OK** to apply the changes.
4. Double-click the **TRIANG3** symbol to open its **Parameters** dialog box. On the **Parameters** tab, set the Amplitude (**AMPL**) to *0.5*, Frequency (**FREQ**) to *FREQU*, and Offset (**OFF**) to *0.5*. On the Output/Display tab uncheck **Show Pin** for Value (**VAL**) as the pin is not used in this model. Click **OK** to apply the changes.
5. Add an on-sheet Rectangular Plot as in the previous examples. However, for this plot, define outputs and add traces for the ***TRIANG3.VAL***, ***EPWM3***, and ***EPWM3_NORM*** signals.

6. Name the plot ***State Graph Model***; and save the project using **File>Save As**.
7. Start the simulation with the **Simplorer Circuit>Analyze** menu command.

   The resulting plot should be identical to the one plotted in the first example.

PWM Modeling with Block Diagram Components

This example models the PWM controller using block diagram components. The PWM is a comparator with two input signals: One input is the triangular wave and the other is the modulation signal. The comparator compares the signals and provides an output signal proportional to the modulation signal.



Placing and Arranging the Components on the Sheet

1. In the **Basic Elements>Blocks>Sources Blocks** folder, select the **CONST** (Constant Value) component and drag the component onto the sheet.
2. In the **Basic Elements>Blocks>Signal Processing Blocks** folder, locate and place one **SUM** (Summation) and two **COMP** (Comparator) components onto the sheet.
3. In the **Basic Elements>Tools>Time Functions** folder, locate and drag a **TRIANG** (Triangular Wave) component onto the sheet.
4. Arrange and connect the components as shown in the figure above.
5. Double-click the TRIANG symbol to open its **Parameters** dialog box and change the **Name** to *TRIANG4*. On the **Parameters** tab, set the Amplitude to *0.5*, Frequency to *FREQU*, and Offset to *0.5*. Click **OK** to apply the changes.

6. Double-click the **CONST** block symbol to open the **Parameters** dialog box. Change the **Name** to *DUTY_CYCLE* and **Value** to *DUTY*. Click **OK** to apply the changes.
7. Double-click the **SUM** symbol to open its **Parameters** dialog box. Click in the **Sign** column of the triangular wave **Input Signal*TRIANG4.VAL*** and select the minus sign (**–**) from the list. Click **OK** to apply the changes.
8. Double-click the uppermost comparator symbol to open its **Parameters** dialog box. Change its **Name** to *EPWM4*. On the **Parameters** tab, set Val1 to *LO*. and Val2 to *HI*. On the Output/Display tab, clear the **Show Pin** box for the output **VAL**. Click **OK** to apply the changes.
9. For the remaining comparator, change its Name to *EPWM4_NORM***,** Val1 to *0*. and Val2 to *1*. Clear the **Show Pin** box for the output **VAL**. Click **OK** to apply the changes.
10. Add an on-sheet Rectangular Plot as in the previous examples. However, for this plot, define outputs and add traces for the ***TRIANG4.VAL***, ***EPWM4.VAL***, and ***EPWM4_NORM.VAL*** signals.
11. Name the plot ***Block Diagram Model***; and save the project using **File>Save As**.
12. Start the simulation with the **Simplorer Circuit>Analyze** menu command.

    The resulting plot should be identical to the one plotted in the first three examples.

8 - Using Legacy Schematics

To use a legacy Simplorer project, you must first translate it using Simplorer Release 16.2 or earlier and then migrate it into Simplorer R17.

In this chapter you will complete the following tasks:

- Translate a legacy Simplorer 7 schematic file using Release 16.2 or earlier.
- Migrate the translated *.asmp file into Simplorer (ANSYS Electronics Desktop 2017.2) as an *.aedt file.
- Set up and run an analysis on the translated model.
- Create reports.

Translating a Legacy Schematic using Release 16.2 or Earlier

If you want to use your legacy version 7 schematics in Release 18.0 or later projects, you must first use Simplorer Release 16.2 or earlier to translate and save the project to make it compatible.

In the following example, you will import a Three-Phase Rectifier .ssh file created in Simplorer version 7. The legacy file's component parameters, simulation setup parameters, and outputs are all translated into a Simplorer Release 16.2 project, which you can then bring into Simplorer Release 18 for analysis and reports.

Importing the Legacy Schematic into Simplorer Release 16 or earlier

To import a legacy .ssh file:

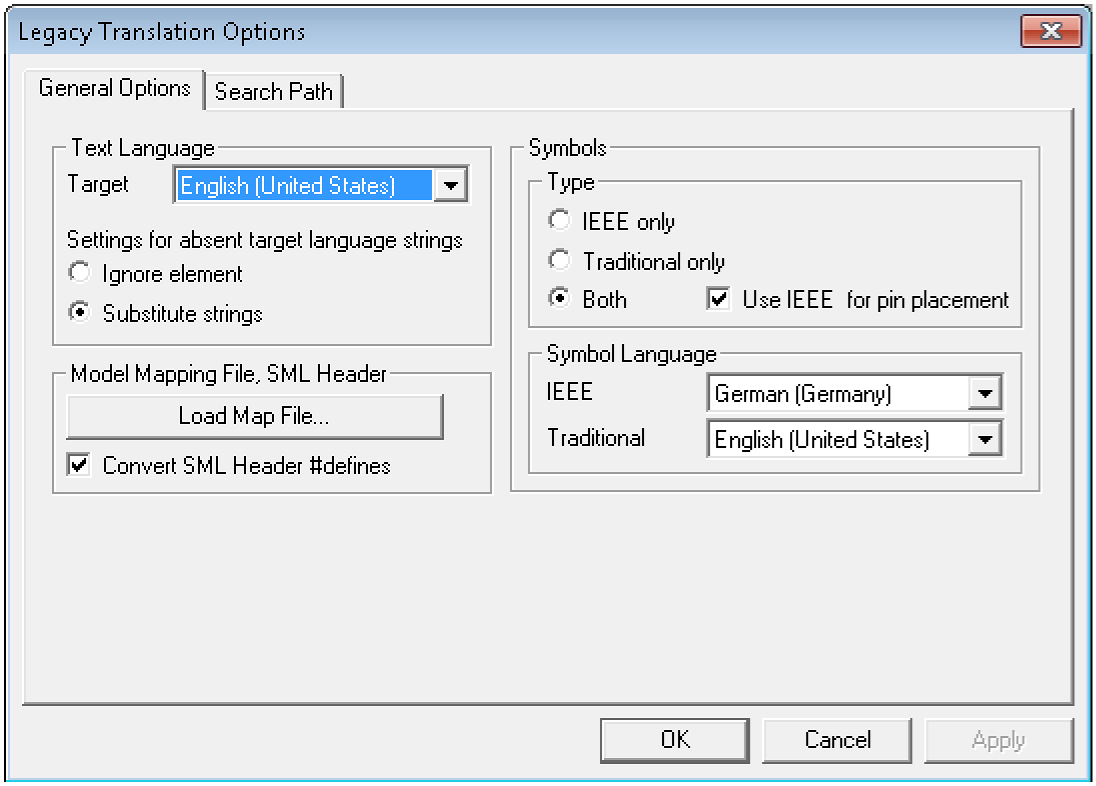
1. Click **File>Open**.

   A standard Windows **Open** dialog box appears.

2. Select **Legacy Simplorer Schematic (*.ssh)** in the **Files of type** drop-down list to filter the window contents.
3. Locate the desired .ssh file (**LegacyRectifier.ssh** for this example) and click **OK** to begin the import process.

Hint	You can also drag the file directly onto the open Simplorer window.

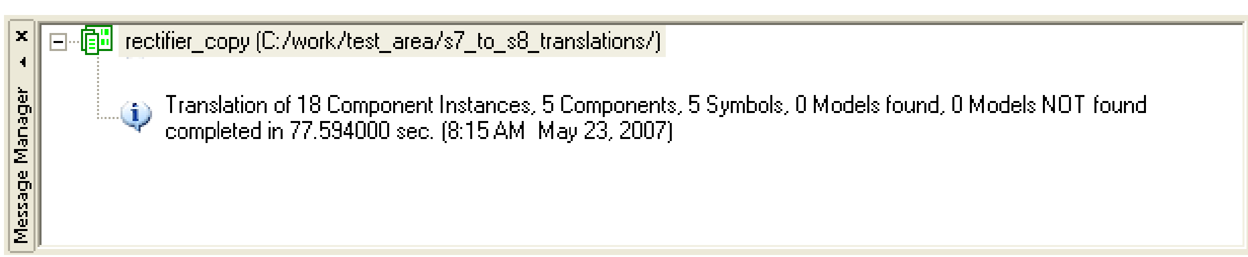
   A Legacy Translation Options dialog box appears.



- The **Text Language** panel allows you to choose the **Target** language into which the project's text strings will be translated. If the target language has no equivalent text for a given element, you can tell the translation tool either to **Ignore element**s for which there are no target language strings; or to use **Substitute strings**.
- The **Symbol** panel allows you to select whether the symbols in the translated schematic will be **Normal** size or five times (**5x**) normal size. You can also select the symbol **Type** and associated **Symbol Language** available for display on the translated schematic.

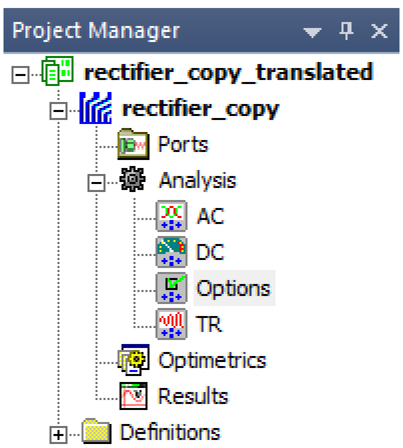
4. For this example, we will use the default values, so leave the settings unchanged and click OK to begin translation.

   The **Message Manager** window informs you of the progress of the translation.



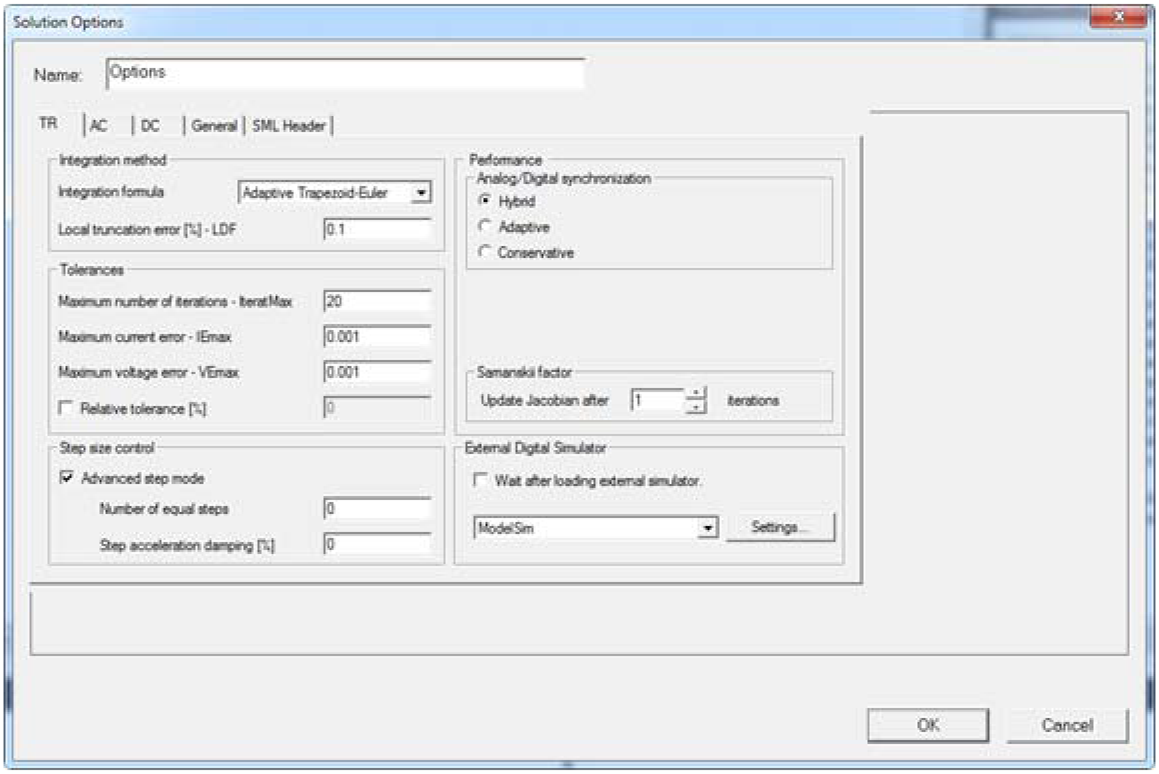
Note	
	• Virtually all of the project's pre-processing data is translated. • Optimetrics setup information, including design variables, is not translated. However, the nominal model is translated. • Solution data is not translated. Therefore, you must solve legacy Simplorer projects again after translation. Existing solution data can be imported either by right-clicking **Analysis** in the project tree and selecting **Import SDB File** from the context menu, or by selecting **Simplorer Circuit> Import SDB File** on the main menu bar.

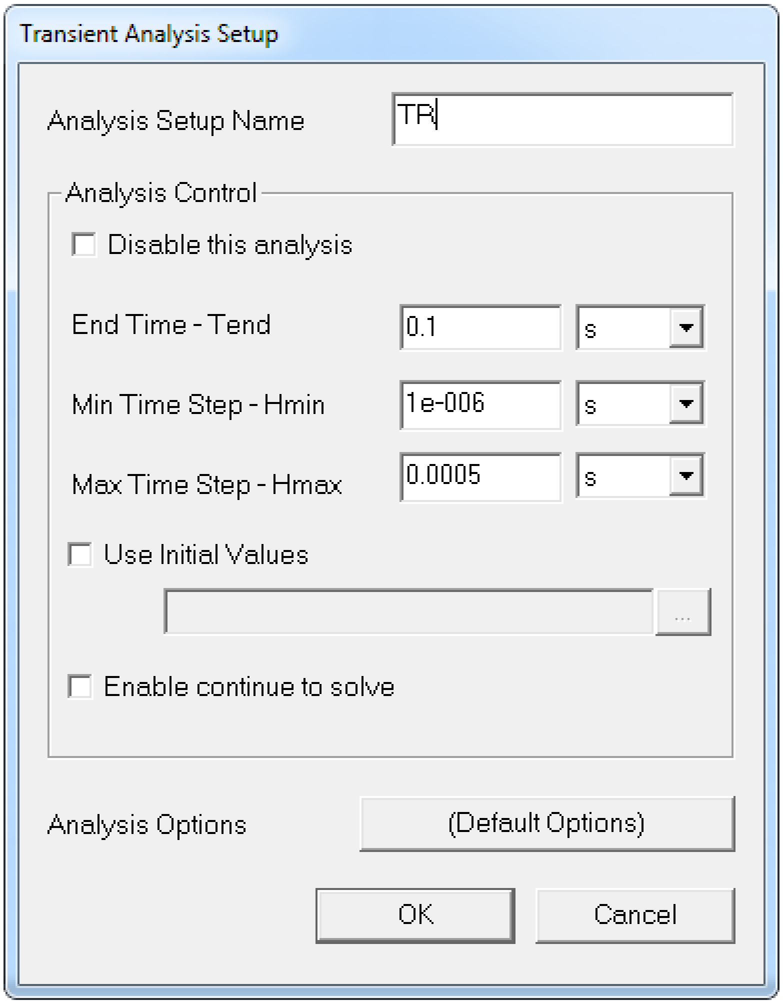
5. When translation is complete, choose **File>Save As** and save the translated project with a name of your choosing before proceeding.
6. In the Project Manager, browse the project tree.



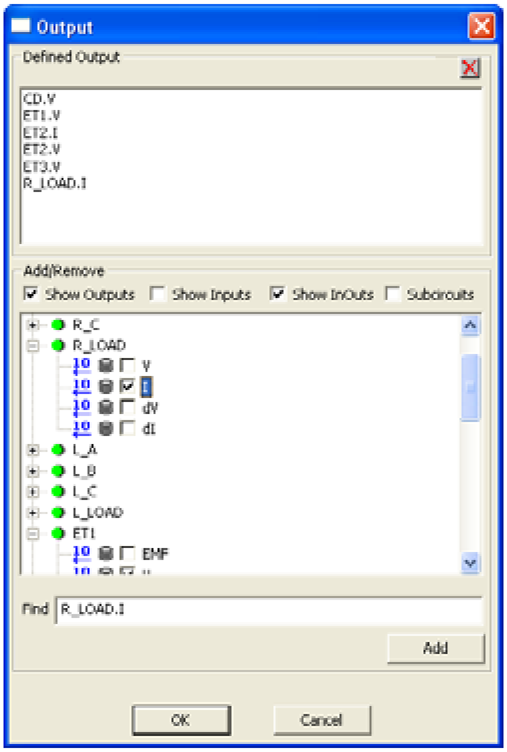
7. Double-click the **Options** and **TR** icons to open the **Solution Options** and **Transient Analysis Setup** dialog boxes, and confirm that the legacy settings have been transferred cor-

rectly.


Solution Options
Name:
Options
TR
AC
DC
General
SML Header
Integration method
Integration formula
Adaptive Trapezoid-Euler
Local truncation error [%] - LDF
0.1
Tolerances
Maximum number of iterations - IteratMax
20
Maximum current error - IEmax
0.001
Maximum voltage error - VEmax
0.001
Relative tolerance [%]
0
Step size control
Advanced step mode
Number of equal steps
0
Step acceleration damping [%]
0
Performance
Analog/Digital synchronization
Hybrid
Adaptive
Conservative
Samanskii factor
Update Jacobian after
1
iterations
External Digital Simulator
Wait after loading external simulator.
ModelSim
Settings...
OK
Cancel



8. Select **Simplorer Circuit>Output Dialog** to open the **Outputs** dialog box and observe that the defined outputs have been translated.

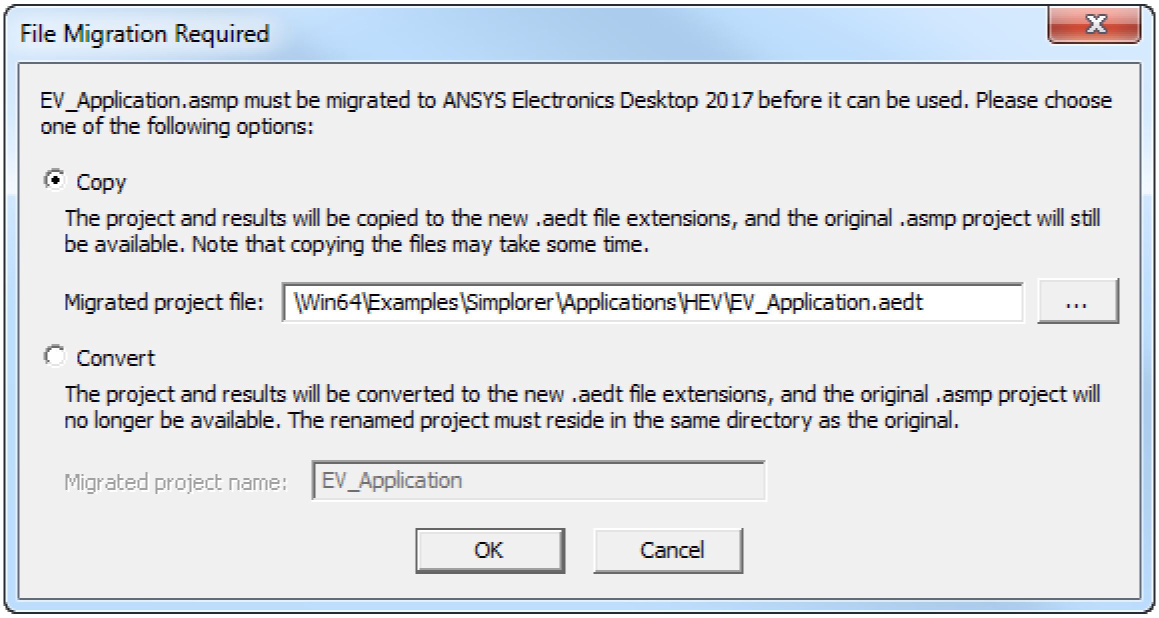


Now that you have confirmed that the necessary conditions for finding solutions for the translated circuit have been correctly set up, the next step is to open the translated schematic in Simplorer.

Opening the Translated Schematic in Simplorer 2017.1 or Later

To open the translated schematic:

1. Click **File>Open**.
2. Navigate to the location of the *.asmp project file, and open the file. The **File Migration Required** dialog is displayed:



Select either **Copy** or **Convert**.

- Select **Copy** if you want to create the new *.aedt file and save the original *.asmp file.
- Select Convert if you want to create the new *.aedt file without saving the original *.asmp file.

3. Click **OK**.

The next step is to start an analysis

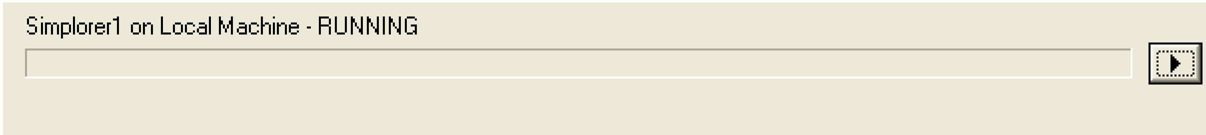
Starting an Analysis

To start an analysis:

1. Start the analysis (simulation) by selecting the **Simplorer Circuit>Analyze** menu command.

   You can also right-click on **Analysis** in the **Project Manager** window and click **Analyze** on the short-cut menu, or press **F12** on your keyboard to start the analysis.

   The simulation model is compiled and outputs are calculated. During the simulation run, the name of the model is visible in the simulation toolbar in the **Progress** window, and a button to stop the simulation is available.



At the end of the simulation, the simulator program remains open so that the simulation can be run again if any of the model parameters are changed.

2. After the simulation, the output quantities can be displayed in plots on the sheet or in report windows. Output data are also saved in a .sdb file (Simplorer Database).

Plotting Simulation Results

Refer to Plotting Rectifier Model Simulation Results for information on plotting simulation results. Detailed information on *Generating and Modifying Reports* is also available in the online help.

Index

S

T

U

V