



PLECS

Tutorial

Introduction to PLECS Blockset

Start learning PLECS Blockset by building two simple electrical circuits

Tutorial Version 1.0

www.plexim.com

- ▶ Request a PLECS trial license
- ▶ Check the PLECS documentation

1 Introduction

The only way to become familiar with a new program is by using it. This tutorial has you model two example circuits step-by-step and in doing so demonstrates the basic features of PLECS.

In this exercise you will learn the following:

- How to build a simple model using components from the PLECS Library browser
- How to interconnect components on the PLECS schematic
- How to parametrize the components
- How to create a subsystem
- How to build a multi-domain model with controls in PLECS
- How to build a multi-domain model with controls in Simulink
- How to use the Probe block

Before you begin Before you begin, ensure that you have installed PLECS Blockset (with Matlab R2007a or higher) and have the reference files that you can compare with your own models at each stage of the exercise.

2 Building a Simple Passive Network

The first electrical system we are going to model is a simple RLC network, as shown in Fig. 1. A capacitor is charged by a DC voltage source via an RL-branch and its voltage is monitored with a voltmeter.

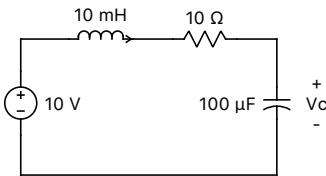


Figure 1: Simple RLC network

To access PLECS Blockset, you simply need to enter `plecslib` into the MATLAB command line. This will bring up a Simulink model that contains a generic PLECS block named “Circuit” and various component libraries. We will refer to it as the *plecslib* toolbox in this documentation. In this toolbox you find electrical components, from which you can create your own circuits, as shown in Fig. 2. Alternatively, you may access the PLECS toolbox by opening it in the Simulink library browser.



Your Task:

- 1 Enter `plecslib` into the MATLAB command line.
- 2 Open a new Simulink model. Drag and drop the PLECS “Circuit” block from the *plecslib* toolbox.
- 3 Double-click on the “Circuit” block to open the PLECS schematic window.

The yellow schematic window that appears is the modeling interface. The components required for the PLECS circuit must be copied into this window from the PLECS “Components” library. Like in Simulink, this is done by dragging them from the library and dropping them into the schematic. You can also pull up the PLECS Library Browser from the **Window** menu of the PLECS schematic window or pressing **Ctrl-L** (for Mac users, use **Cmd** instead of **Ctrl** for all shortcuts from here on).

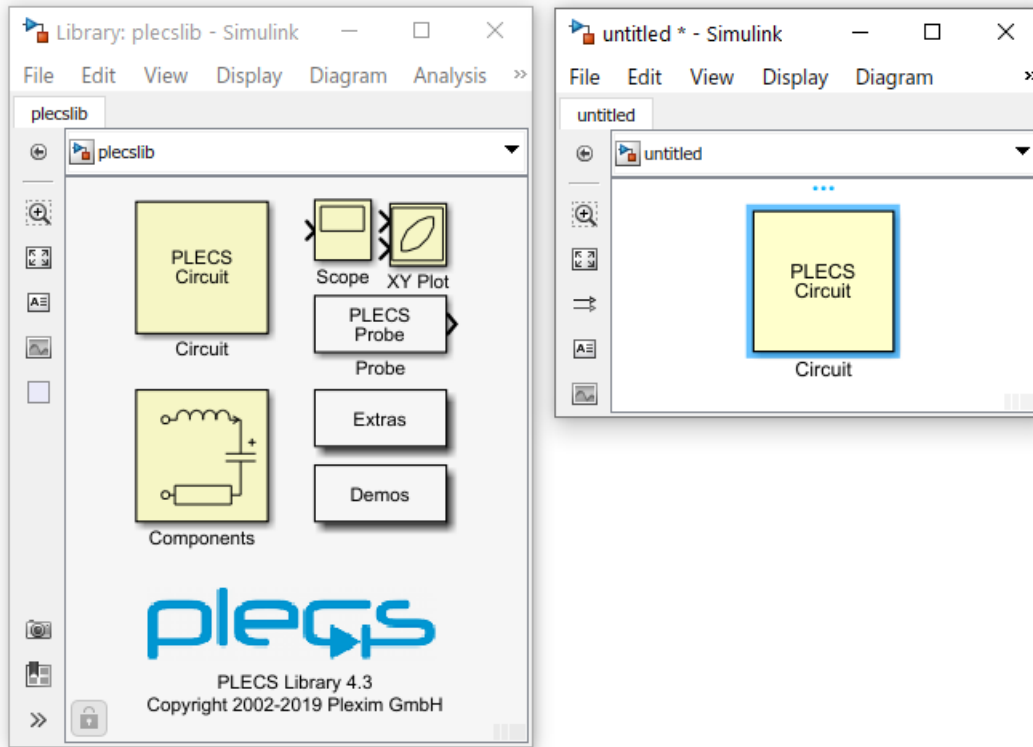


Figure 2: *plecslib* toolbox and PLECS circuit in Simulink

Components

The components required for building the circuit must be dragged and dropped into the model window from the PLECS Library Browser. If you want to duplicate an existing component already present in the model window, hold down the **Ctrl** key, drag and drop the component to a new location or use the right mouse button (of course you can also select the component and **Copy** and **Paste** as usual).



Note: Simulink components cannot be placed inside the PLECS schematic window and components from the PLECS library cannot be placed at the Simulink level, since both programs do not share the same Graphical User Interface. A second version of the PLECS Scope, XY Plot and Probe blocks are provided, as shown in Fig. 2, specifically for use within the Simulink environment.

The electrical components that you need to build the RLC network can be found in the Electrical domain library, under the sub-libraries Sources, Meters and Passive Components. The Scope component is located in the System library. Instead of browsing for the components by clicking on the arrows next to the domain names or using the arrow keys on your keyboard once something is selected, you can also search for them by typing the first letters of the component you need in the search bar. For example, typing **sc** displays the scope or typing **res** displays all the available resistors etc.

Signals

In addition to the electrical connections (black wires) that are used to connect electrical components, PLECS also makes use of unidirectional signals. The signals are in green and have an arrowhead to indicate their direction. These signals carry non-electrical information like measurement values or triggering pulses for switches. Signals can be used in calculations or displayed in a scope. Electrical connections cannot be fed into a scope directly, you always have to use a voltmeter or an ammeter to convert the electrical quantities into a signal first.



Your Task:

- 1 Drag and drop the Voltage Source DC, Inductor, Resistor, Capacitor, Voltmeter and Scope components from the library onto the schematic window, as shown in Fig. 3.
- 2 You can move the required components with the left mouse button. To rotate selected components press **Ctrl-R**, to flip them horizontally press **Ctrl-F**, or to flip them vertically press **Ctrl-I**. All these functions can also be accessed from the **Format** menu.



After you have copied all components to the schematic window, your model should be the same as the reference model `rlc_network_1.mdl`.

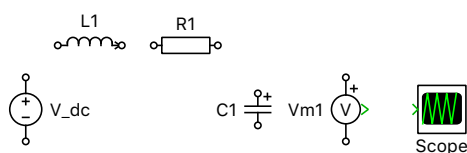


Figure 3: Components of the RLC network

Connections

Each component in the current schematic has black circles or green arrowheads to represent terminals. If we bring the mouse pointer close to such a terminal the pointer shape changes from an arrow to a cross. We now can drag a connection to another component by holding the left mouse button down. When we approach another terminal or an existing connection of the same connection type, the pointer shape changes into a double cross. As soon as we release the mouse button a new connection will be created.

For creating a branch connection from an existing connection, place the mouse cursor over it, hold **Ctrl**, then click and drag your cursor in the desired direction where you want the branch to start.



Your Task: Interconnect the components to complete the RLC circuit. Then connect the output terminal of the voltmeter to the input terminal of the scope.



At this stage, your model should be the same as the reference model `rlc_network_2.mdl`, as shown in Fig. 4.

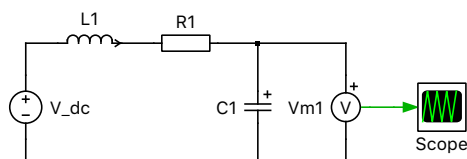


Figure 4: RLC network with scope

Component Parameters

Each component is identified by a unique name, which is chosen automatically. If desired, this name can be changed by double-clicking on the component in the schematic. This is intended for documentation purposes only and does not affect the simulation. The parameters, on the other hand, are of greater importance. The parameters determine, for example, the inductance of an inductor, the capacitance of a capacitor, or the voltage of a DC voltage source. Double-clicking on the component icon opens a dialog window in which you can set these parameters, as shown in Fig. 5.

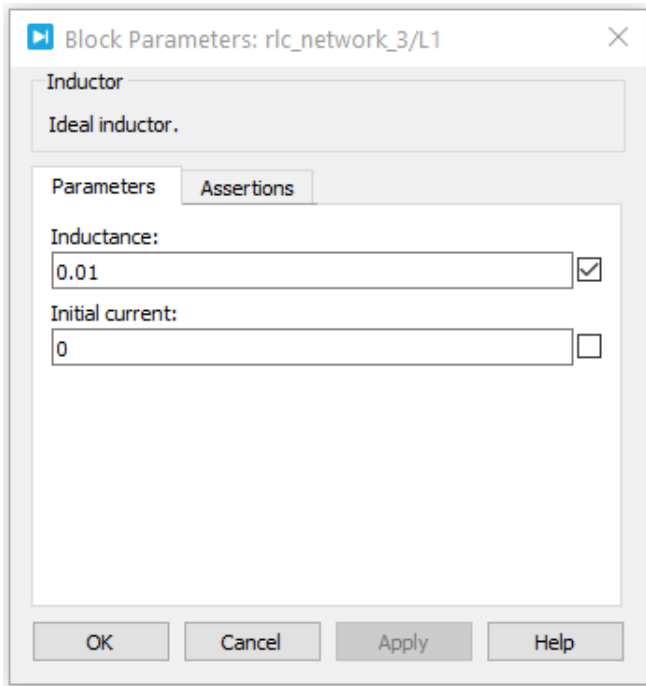


Figure 5: Parameter dialog window of an inductor

Notice that there is a checkbox to the right of each parameter field. If you check this checkbox, that specific parameter value will be displayed at the schematic level. This allows you to view and edit component parameters directly from the schematic. To avoid unnecessary text in the schematic, it is preferable to only display the most important parameters of a component.

Units

Like Simulink, PLECS does not understand units since letters can be used as custom variables. Therefore, the user must make sure that the variables are scaled correctly. For power electronics, it is recommended to use the SI quantities. For example, use $10\text{e-}3$ or 0.001 , instead of 10 mH for an inductance value. For power systems, it may be more appropriate to work with “per unit” quantities.



Your Task:

- 1** Double-click on each component of the RLC network to display its parameter dialog window. Then set the source **Voltage** to 10, **Inductance** to 0.01, **Resistance** to 10 and the **Capacitance** to $100\text{e-}6$.
- 2** Check the checkboxes next to these parameters to display them on the schematic.
- 3** The inductor and capacitor have additional parameters such as **Initial current** and **Initial voltage**, respectively. Leave both these values as their defaults of zero.



At this stage, your model should be the same as the reference model `rlc_network_3.mdl`, as shown in Fig. 6.

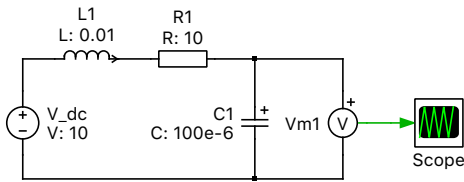


Figure 6: RLC network with parameters

Simulation

PLECS Blockset uses Simulink's solvers directly. In many ways a PLECS Circuit block behaves just like a regular Simulink subsystem and is treated natively by the Simulink solvers. Therefore simulation parameters such as the simulation time, solver type, solver options etc. must be set in the **Simulink Parameters** dialog window of the **Simulation** menu. This dialog can also be opened using the keyboard shortcut **Ctrl-E**.



Your Task:

- 1** Open the Simulink **Model Configuration Parameters** dialog window from the **Simulation** menu and set the simulation **Stop time** to 0.1. Leave the other parameters as their default values.
- 2** Next, start the simulation by selecting **Start** from the **Simulation** menu or by pressing **Ctrl-T**.



Your simulation results obtained should be the same as those shown in Fig. 7.

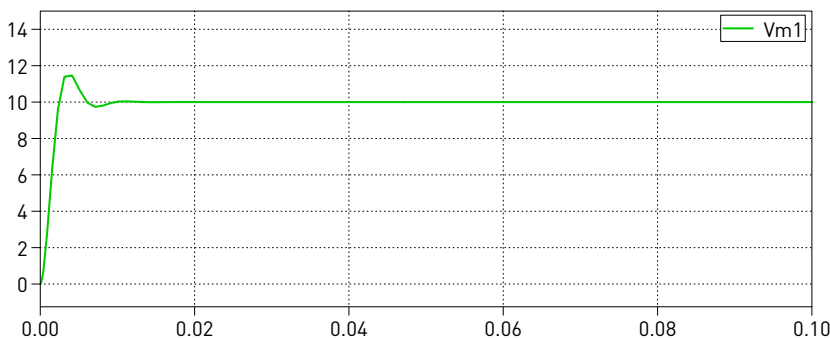


Figure 7: Simulation results of the RLC network capacitor voltage

Adding Control Blocks

To enhance the static electrical model, we can add some dynamic behavior. Let's observe how the capacitor in our example charges and discharges if we apply a pulsed voltage.



Your Task:

- 1 In the schematic, delete the Voltage Source DC component and replace it with a Voltage Source (Controlled) block. This block accepts any signal from the Control domain as an input.
- 2 Drag and drop a Pulse Generator component to the left of the voltage source. Set the **High-state output** to 10 and **Frequency** to 25. Leave all the other parameters as their default values.
- 3 Connect the output terminal of the Pulse Generator block to the input terminal of the controlled voltage source.
- 4 Now drag and drop a Signal Multiplexer component in between the Voltmeter and Scope blocks and set the number of inputs to 2.
- 5 Using the Signal Multiplexer, connect the outputs of the Pulse Generator and the Voltmeter to the Scope.
- 6 Start the simulation.



At this stage, your model should be the same as the reference model `rlc_network_final_plecs.mdl`, as shown in Fig. 8. Your simulation results obtained should be the same as those shown in Fig. 9.

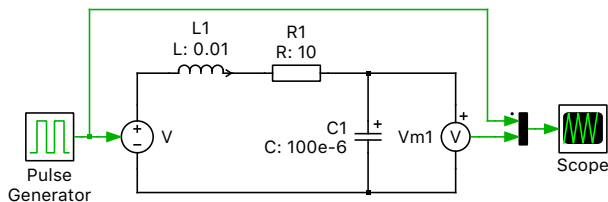


Figure 8: RLC network with a pulsed voltage source in PLECS

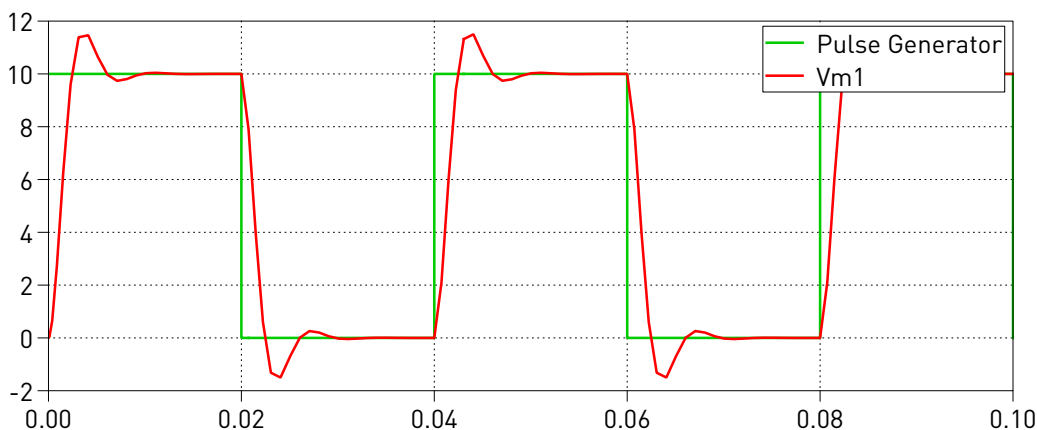


Figure 9: Simulation results of the RLC network with dynamic stimulation

Connecting PLECS to Simulink

Up to now the PLECS circuit lacks a connection with the Simulink environment. You will notice this from the fact that the PLECS Circuit block in Simulink does not have inputs or outputs. In order to

add inputs and outputs we need to use port blocks from the PLECS System library, in this case to access in Simulink the voltage measured by the voltmeter inside the PLECS subsystem and receive from Simulink a square wave signal to modulate the voltage source also in the PLECS subsystem. Signals in PLECS correspond to the connections between Simulink blocks and the port blocks provide unidirectional information interchange between components in PLECS and Simulink. To demonstrate this, let's recreate the control blocks, shown in Fig. 8, at the Simulink level, to simulate the electric circuit we already made in PLECS.



Your Task:

- 1 Open the `rlc_network_final_plecs.mdl` model, and add one Signal Inport block and one Signal Outport block from the System library into the PLECS schematic window.
- 2 Delete the Pulse Generator block and replace it with the Signal Inport component.
- 3 Delete the Scope block and replace it with the Signal Outport component.

You should now see an input and an output terminal on the PLECS Circuit block at the Simulink level.



Your Task:

- 1 Next, drag and drop the Pulse Generator component from the Simulink library and the Scope component from the *plecslib* toolbox to the Simulink schematic window and connect these components to the PLECS circuit.
- 2 Set the required parameters for the Pulse Generator as given earlier and start the simulation.



At this stage, your model should be the same as the reference model `rlc_network_final_simulink.mdl`, as shown in Fig. 10. Your simulation results obtained should be the same as those shown in Fig. 9.

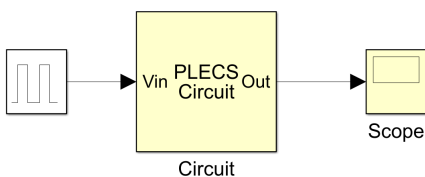


Figure 10: RLC network with a control block and PLECS Scope in Simulink



Note: The signal generated by the pulse generator is discrete, i.e. its value changes abruptly. Normally, the PLECS Scope would determine the signal type automatically and display vertical slopes. In this case, however, the discrete signal coming from the pulse generator is multiplexed with a continuous signal before reaching the Scope, so you may see trapezoidal curves. In order to avoid this, the signal type must be set manually to **discrete** in the Data window of the Scope, as shown in Fig. 11.

3 Buck Converter

Next, in this section we will introduce the concept of ideal switches. We will build the buck converter shown in Fig. 12 along with a closed-loop controller.



Data		
Name	Cursor 1	Cursor 2
Time	0.0343148	0.066315
Plot 1		
Vin 	0	0
Vm1 	0.00556783	0.111828

Figure 11: Data window of the PLECS Scope

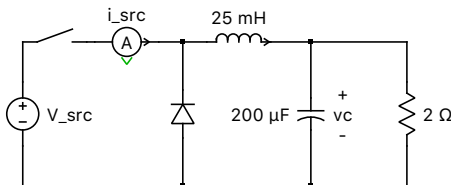


Figure 12: Schematic of the buck converter

Switches

For the buck converter shown in Fig. 12, the transistor can be modeled using a controllable switch, or an ideal IGBT or MOSFET component, bearing in mind that it may conduct current in only one direction. We also need a free-wheeling diode. The diode is a switch that closes as the voltage across it becomes positive, and opens as the current through it becomes negative.

The diode can be found in the Electrical/Power Semiconductors library and the switch in the Electrical/Switches library. All components in these two libraries are based on ideal switches that have zero on-resistance (you can optionally add some) and infinite off-resistance. They open and close instantaneously. For more information, right-click on any component in the library and select **Help**.

The switch is controlled by an external signal. It will close upon a non-zero input and open when the signal goes back to zero.



Your Task:

- 1 Create a new Simulink model, then drag and drop a PLECS circuit block onto it.
- 2 Drag and drop the Switch and Diode components from the PLECS library onto the PLECS schematic window.
- 3 Build the rest of the electrical part of the buck converter, as shown in Fig. 13.



Note: To hide a component's name on the schematic, right-click on the component, then from the **Format** menu, uncheck **Show name** or use the keyboard shortcut **Ctrl-Shift-N**.



At this stage, your model should be the same as the reference model buck_converter_1.mdl.

Subsystems

We can separate certain parts of the complete model from one another, such as the electrical part from the control part in this case. This has no effect on the simulation result but makes the whole system

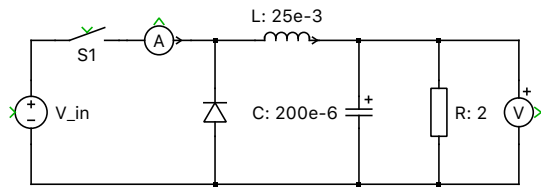


Figure 13: Electrical part of the buck converter

more structured. The use of subsystems can provide a clear presentation of the model and make debugging easier.



Your Task:

- 1 After completing the circuit shown in Fig. 13, select all the components (either by clicking an empty space in the upper left corner of the schematic and dragging a frame to the lower right corner, or by using the keyboard shortcut **Ctrl-A**).
- 2 Create a new subsystem by clicking on **Create Subsystem**, either from the **Edit** menu, or by right-clicking on one of the selected components, or using the keyboard shortcut **Ctrl-G**. The electrical components should now be in a new subsystem called “Sub”.
- 3 Optionally, you can rename the subsystem to, e.g. “Circuit”.



Note: The size of the subsystem icon can be changed by dragging one of the selected corners. You can also move the name label to another position by clicking and dragging it to the borders or the corners of the icon.

Signal Ports

Link in Simulink, to connect a PLECS subsystem to the outer schematic we need to place ports into it.



Your Task:

- 1 Drag two Signal Inputs and two Signal Outputs into the subsystem schematic.
- 2 Connect the Signal Inputs to the voltage source and the switch, respectively.
- 3 Connect the Signal Outputs to the voltmeter and the ammeter, respectively.
- 4 Optionally, rename the signal ports.



Note: You will notice that a new terminal appears in the subsystem icon for each port that you drag into the subsystem schematic. To move these terminals along the icon’s frame, drag them with the mouse while holding down the **Shift** key.



At this stage, your model should be the same as the reference model `buck_converter_2.mdl`.

Controls in PLECS

For the buck converter, we will implement a hysteresis type control that keeps the capacitor voltage roughly in a $\pm 0.2\text{V}$ band around 6V . This involves using a Constant block as a reference set point and feeding back the measured load voltage. An error signal is then generated as the difference of the set point and measured voltage, and this is then fed to a Relay block whose output is now the gating signal to the switch. The control scheme here sets the gate signal to the switch high if the load voltage falls below 5.8V , and the gate signal to low if the voltage increases above 6.2V . We are simply regulating the current flow through the inductor and therefore the charge state of the capacitor.

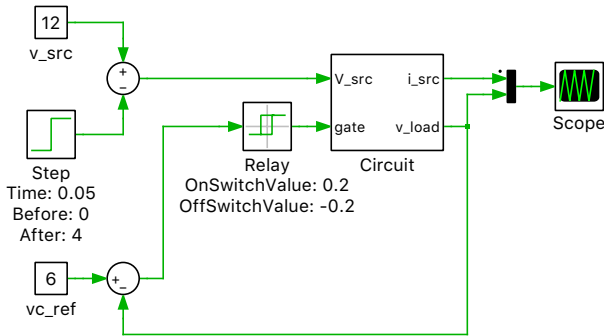


Figure 14: Buck converter with hysteresis control in PLECS



Your Task:

- 1 Drag and drop a Relay, Sum, and Constant block from the Control domain library onto the schematic window.
- 2 Build the controller shown in Fig. 14.
- 3 To make things a bit more interesting, apply a step change, from 12V down to 8V , to the input voltage during the simulation.
- 4 Set the simulation **Stop time** as 0.1s and start the simulation.



At this stage, your model should be the same as the reference model `buck_converter_3_plecs.mdl`. Your simulation results obtained should be the same as those shown in Fig. 15.

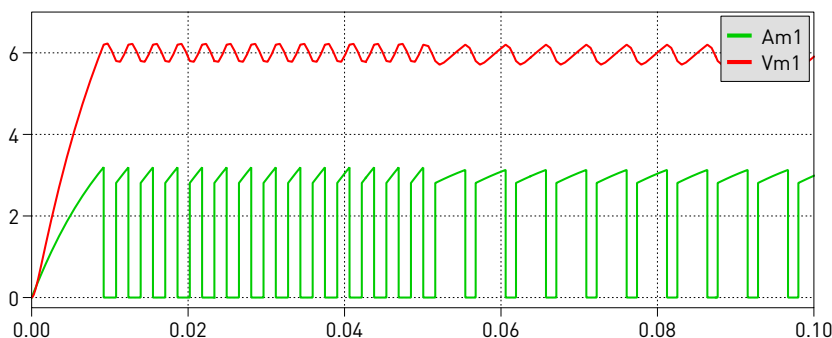


Figure 15: Simulation results of buck converter with hysteresis control

Controls in Simulink

Alternatively, the controller can also be built at the Simulink level.



Your Task: Build the hysteresis type controller at the Simulink level and use the PLECS Scope intended for use at the Simulink level, as shown in Fig. 16.



At this stage, your model should be the same as the reference model `buck_converter_3_simulink.mdl`. Your simulation results obtained should be the same as those shown in Fig. 15.

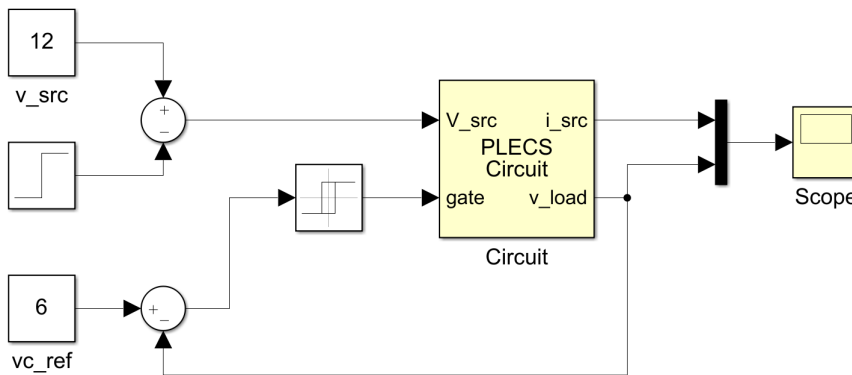


Figure 16: Buck converter with hysteresis control in Simulink

Probe

Instead of using an Ammeter or Voltmeter component to measure the signals, the signals can also be measured using a Probe block. This component can help to keep a model organized by reducing unnecessary connections, and gives you access to quantities that are difficult to measure otherwise.

There are two versions of Probe block available. One is for use within the PLECS schematic and is located in the System category of the PLECS library browser and the second version is specifically for the Simulink level (although it still only monitors quantities from PLECS components inside the Circuit block) and is located in the *plecslib* toolbox window.

To associate a component with the Probe block in PLECS, drag the component of interest onto it in the schematic and release your mouse once an arrow appears. Don't worry - if you are doing this correctly, as soon as you release the mouse, the component that you were dragging will return back to its original place. You can also double-click on the Probe block and drag the component of interest into the Probe Editor window that opens and release your mouse once an arrow appears.

To associate a PLECS component with the Probe block at the Simulink level, double-click the Probe block and drag the component of interest into the Probe Editor window that opens and release your mouse once an arrow appears.



Your Task:

- 1 Drag and drop the appropriate Probe block either into the PLECS schematic or Simulink schematic.
- 2 Associate the Voltage Source and Resistor components from the "Circuit" subsystem and select the **Source current** and **Resistor voltage** component signals, respectively.

- 3** Add a second plot to the scope window (the one at the same level as your Probe block) by right-clicking on the existing plot and selecting **Insert plot above**.
- 4** Connect the output terminal of the Probe to the newly created input terminal of the Scope block.
- 5** Start the simulation. You will notice that both the upper and lower plots are identical.



At this stage, your model should be the same as the reference model
`buck_converter_final_plecs.mdl` or `buck_converter_final_simulink.mdl`.

4 Demo Models

Now that you've built your first models in PLECS it may be worthwhile to take a look at the demo models that come with PLECS. Open the demo model browser by selecting **Demo Models** from the **View** menu within PLECS Blockset or from the button located in the *plecslib* toolbox window.

Revision History:

Tutorial Version 1.0 First release

How to Contact Plexim:

☎	+41 44 533 51 00	Phone
	+41 44 533 51 01	Fax
✉	Plexim GmbH Technoparkstrasse 1 8005 Zurich Switzerland	Mail
@	info@plexim.com	Email
	http://www.plexim.com	Web

PLECS Tutorial

© 2002–2021 by Plexim GmbH

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

PLECS is a registered trademark of Plexim GmbH. MATLAB, Simulink and Simulink Coder are registered trademarks of The MathWorks, Inc. Other product or brand names are trademarks or registered trademarks of their respective holders.