

PLECS Shortcuts Cheat Sheet:

Modeling, Simulating, and the Scope



Modeling

To **copy** components already placed in the window hold down [Ctrl] while dragging them to a new location.

To **rotate** selected components clockwise press [Ctrl-R], to **flip them horizontally** press [Ctrl-F], and to **flip them vertically** press [Ctrl-I]. (Also use [Shift] for counter-clockwise direction)

For **drawing a branch** from an existing connection place the mouse pointer where it is desired that the branch will start. With the right mouse button or with the left mouse button while holding down the [Ctrl] key, create a branch to a new destination by dragging the crosshairs.

To **connect a component into** an existing connection, simply drop a component on the connection so that the input and output ports are aligned.

To **connect a component to** a second component, simply click on the first component and while holding down the [Alt] key, click on the second component.

To **remove selected components from wired connections**, press [Shift] while dragging them to a desired destination.

To **move a terminal of a subsystem** to another free slot of the border, use the left mouse button while holding down the [Shift] key and drag the port to the desired slot.

To **probe a component**, drag the component directly onto the Probe block in the model with the left mouse button and release once an arrow appears.

To **zoom in or out** of your schematic, scroll up or down, while holding down the [Ctrl] key. Alternatively, press [Ctrl and +] or [Ctrl and =].

To **zoom to fit** in your schematic, press the [Ctrl and *].

To **annotate** a circuit, simply double-click on a blank part of the model window and type text in the box that appears.

The **symbol format for resistors and capacitors** can be changed between DIN or ANSI style from the PLECS Preferences window [Ctrl-,].

A **grid** setting in the PLECS Preferences window [Ctrl-,] can be used to display a grid in the background of the schematic windows to facilitate placing components and their connections.

Simulating

The **solver and simulation settings** are accessed from the corresponding menu entry in the “Simulation” menu or press [Ctrl-E]. (This will open the Simulink Configuration Parameters window for PLECS Blockset users)

To **start** the simulation, press [Ctrl-T] or select “Start” from the “Simulation” menu. Use the same command to **stop** a simulation that is running.

Using the PLECS Scope

To **add a plot** to the Scope, right-click on an existing plot and choose “Insert plot above” or “Insert plot below”, or access the Scope parameters from the File menu or icon along the top of the Scope.

To **pan and view waveforms** that are out of the Scope window, click and drag either of the X or Y axes of the plot with the hand symbol that appears.

Two zoom modes exist: Constrained Zoom and Free Zoom. The zoom mode is selected using the toolbar button. To **temporarily switch zoom modes** press the [Ctrl] key.

To **move cursors in between time steps** hold the [Shift] key while dragging to the right or left. The data values will update in the fields based on the cursor location.

To **change the signal display type**, click on the signal type icon next to the signal name in the data view window. Signals can be of the continuous, discrete or impulse type.

The **Scope background color and appearance of the curves** can be configured from the Scope Colors tab in the PLECS Preferences window [Ctrl-,].

*Mac users should use [Cmd] rather than [Ctrl] for all of the above.