






















Eagle Schematic Reference

SparkFun Electronics Summer Semester

SCHEMATIC SHORTCUTS:

KEY	ICON	COMMAND	DESCRIPTION
F3		Delete	Deletes any object from the schematic (Warning: Deleting a trace or a component will cause the corresponding trace/component on the PCB to be deleted.)
F4		Name	Allow the renaming of components or 'Nets' (Signal)
F5		Value	Used to set the value of a component
F6		Smash	Allows the component name and value to be repositioned independent of the component
F7		Move	Moves a component (Hint: Hold 'Control' while moving to move a selected group)
Alt + F7		Group	Use this to select a group of objects
F8		Copy	Copies a component or trace (Warning: Won't copy a group!)
F9		Net	Use this to create a 'wire' connecting different signals
Alt + F9		Label	Puts a text label with a nets name on the schematic
F10		Set Small Grid	Changes the grid to 0.05"
Alt + F10		Set Metric Grid	Changes the grid to 1mm
		Add	Adds a part to the schematic
		ERC	Run the Electrical Rule Check
		Grid	Opens the grid setting window
Alt + F1		Info	Show the attributes of a component
		Junction	Adds a junction node to the intersection of two signals
		Layer	Show the Layer options for the schematic
		Cut	Copies a group of parts in the schematic
		Paste	Pastes a copied group in the schematic (Not used with Copy)
F1		Show	Shows the all of the connected signals in a specific net
		Create Board	Creates a PCB design file and adds all of the parts from the schematic to the board.

Eagle Schematic Reference

SparkFun Electronics Summer Semester

SCHEMATIC TOOLBAR

Info		Show
Layers		Mark
Move		Copy
Mirror		Rotate
Group		Change
Cut		Paste
Delete		Add
Swap		Replace
Gateswap		
Name		Value
Smash		Miter
Bend		Activate
Line		Text
Circle		Arc
Rectangle		Polygon
Bus		Net
Junction		Label
Attribute		
ERC		Show Errors

SCHEMATIC TIPS AND TRICKS

Always use a frame in a schematic, and keep all the parts within the frame. If more space is needed, add a new page and frame to the design.

Change shortcut assignments by going to the Assign... window in the Options menu.

Copy, Cut and Paste don't work like you'd think they would. Be careful to understand exactly what you're doing before using these tools.

The normal commands (Ctrl+C, Ctrl+X and Ctrl+V) don't work in Eagle.

If a trace is copied and pasted, go back and double check it's connections using the 'show' command to ensure it's been connected in the intended spot.

The 'Cut' command is used to copy a group of components and nets. It won't remove the group that is 'cut.'

Name your nets. This will make routing the board easier.

Add text labels to sections of the board to make it easier to understand from a third persons view.

Always remember to run the ERC before moving onto the PCB layout.

To move a group of components first select the group, then with the 'Move' tool selected hold the 'Ctrl' key and right-click near the center of the group. Keep holding the 'Ctrl' key and move the group to the desired location; click again to finish the move.

Press and hold the middle mouse button and drag the mouse around to pan around the window.