WEBSITE: sparkfun.com

6175 LONGBOW DRIVE, SUITE 200 ZIP CODE: 80301
BOULDER. COLORADO **USA**

[303]

284.0979 [GENERAL]
443.0048

Eagle Layout Reference
SparkFun Electronics Summer Semester

PCB LAYOUT SETTINGS:

0.05" Grid turned ON

0.02" Round Vias

0.01" Trace width

15% Ratio for Text

PCB SHORTCUTS:

| KEY	ICON	COMMAND	DESCRIPTION
F1		Show	Highlights the selected object and all similar.
Alt + F1		Info	Shows the attributes of an object.
Alt + F2		Zoom to ft	Redraws the window to ft all of the objects in the layout.
F3		Delete	Deletes an object from the PCB
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		Ratsnest	Finds all the 'airwires' in the design (unconnected signals!)
F9		Route	Use this to create a 'wire' connecting different signals
Alt + F9		Ripup	Deletes a trace
F10		Set Small Grid	Changes the grid to 0.05"
Alt + F10		Set Metric Grid	Changes the grid to 1mm
F11		Change Layers 1	Shows the 'Routing' layers
Alt + F11		Change Layers 2	Shows the layers to create an assembly sheet
F12		Turn on polygons	When the ratsnest is processed, polygons will be included (often takes much longer, but required before the design can be finished)

Eagle Layout Reference
SparkFun Electronics Summer Semester

Alt +F12		Turn of polygons	Polygons not included in ratsnest. Faster, but won't connect signals to the signal planes on the board (false airwires will appear).
		Copy	Copies a component (Like text or polygons).
		Cut	Copies a group of components.
		Paste	Pastes a group of components that have been cut.
		Generate Gerbers	Generates the design f les for the PCB.
		DRC	Runs the Design Rule Check.
	T	Text	Adds text to the PCB
		Via	Adds a Via to the board (can be used to transfer a route from the top layer to the bottom layer).
Alt +Backspace		Undo	

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

SparkFun Electronics Summer Semester Educational Material

TIPS AND TRICKS

Try placing all components on a 0.1" grid

For signals use at least a .008" (8mil) trace width

For power traces use at least .01" (10mil) trace width, but if there's room to make it bigger then do so.

LABEL, LABEL, LABEL. Label switches (and their states), LEDs, buttons, and headers. Use the silkscreen to designate the purpose of these items.

If a label interferes with a via, either move the via or make sure that the via will be tented.

Put a Date Code on the back of the board. This will help when you've got multiple versions of the same board.

Create Ground Pours on the top and bottom layer

Change the Isolation of the Ground Pour to 12 mil. This prevents manufacturing errors

Group components on the PCB according to how they are grouped in the schematic.

Keep at least 8mil between traces

Only use 45 degree angles on the traces. Never put a 90 degree angle on a trace.

Traces on the same layer (same color) must not intersect unless they are the same signal!

Route traces from the middle of the pad.

While routing a trace, press the middle mouse button to insert a via and transfer the route from one layer to the other.

Use the 'Ratsnest' to f nd out if there are airwires(unconnected signals).

If you can't f nd the airwires, open the 'layers' window and click 'none' to turn off all of the layers. Then f nd the 'Unrouted' layer in the window and turn that layer on. This will show you where the airwires are. After you f nd them, press F11 to turn all of the layers back on.

When a board is f rst created there are 16 layers; typically only 2 layers are used on a board. To get rid of the extra layers (a very annoying bug in Eagle):

1. Open the DRC

2. Go to the 'Layers' tab

3. In the 'Setup' text box erase the parenthesis around 1*16.

4. Press 'Check'.

5. Open the DRC again.

6. Put the parenthesis back around 1*16 in the Setup box of the layers tab.

7. Press check again

Hold the 'Alt' key while routing a trace to place the route on a f ne grid. (This also works while moving components, but the components should stay on the major grid when possible).

Use the DRC to check the board for design errors.

ALWAYS ALWAYS ALWAYS print a 1:1 copy of the layout and make sure that the parts f t the way they should.