

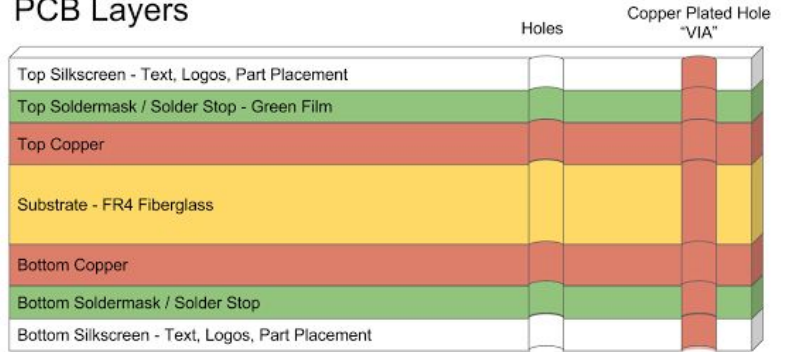
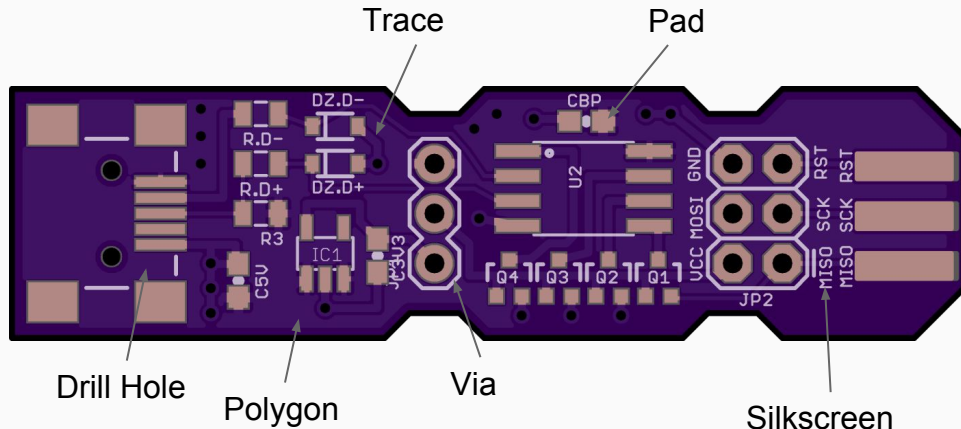


Advanced Projects

Lecture 4: PCB Design

What is a PCB?

- A printed circuit board (PCB) is a board copper traces that create a circuit, and exposed pads or through-holes for soldering on circuit components.



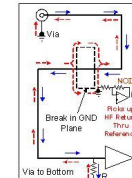
Parasitic Elements

- All traces have some resistance and inductance, and are capacitively linked to nearby traces
- These parasitic impedances can distort signals and cause circuits to fail
- To minimize resistance and inductance, keep traces short and wide
- To minimize capacitances, increase the space between traces



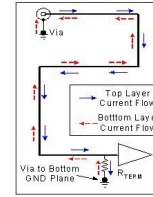
Input Signal Routing

Use direct routing and avoid loops



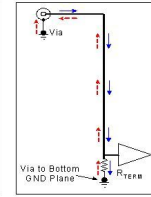
WORST

Large Current Loop +
Discontinuous GND
Plane



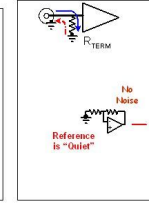
BAD

Large
Current Loop



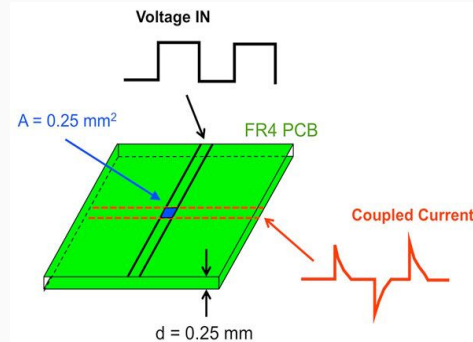
BETTER

Reduced
Current Loop

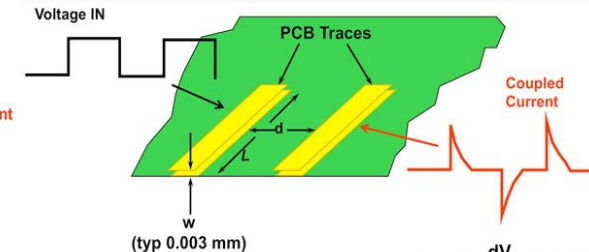


BEST

Minimum Current
Loop



Note: 10 mil = 0.25 mm.



$$I = C \frac{dV}{dt} \text{ (amps)}$$

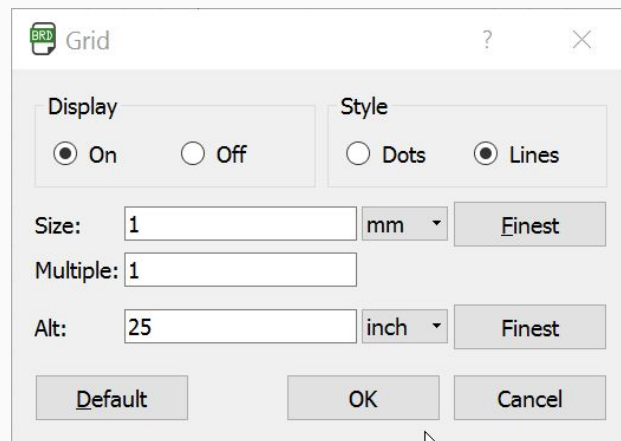
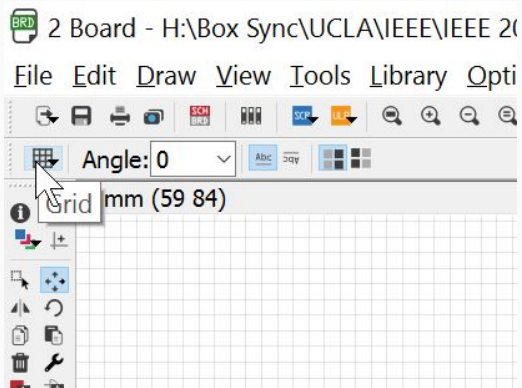
Creating a Layout File

Until now, you have only worked with the schematic capture tool in EAGLE. Now, you will begin working with the layout editor.

- Just as EAGLE saves schematics in .sch files, it saves board/layout designs in a **.brd file**.
- In virtually all cases, you should **design the schematic before you design the board**.
- Once you are ready to begin creating a board, use the **board** command to open up a layout editor (or click the “Generate/switch to board” button)
- When editing the schematic/board, keep both files open: Eagle automatically keeps the two files in sync, but only if they are both open at the same time.

Setting up your Environment

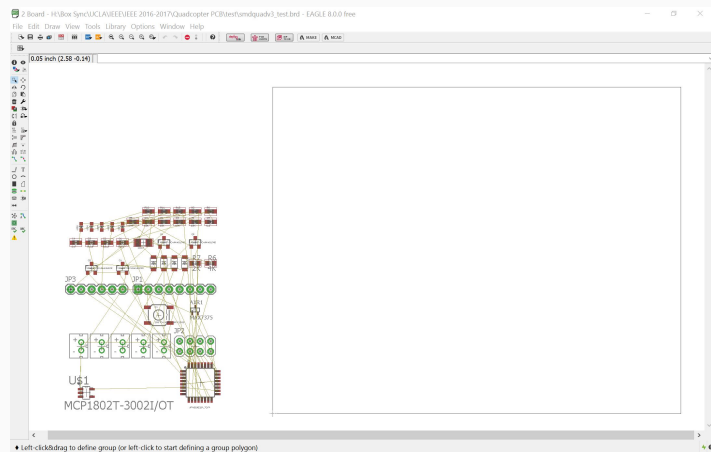
Use the **grid** menu to specify the units your design will use. Use **mm** for creating the dimensions of your board, but use **mil** for drawing traces.



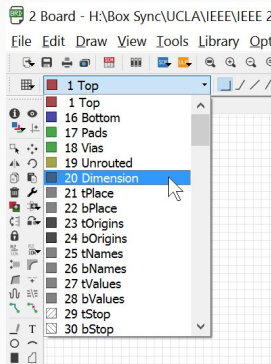
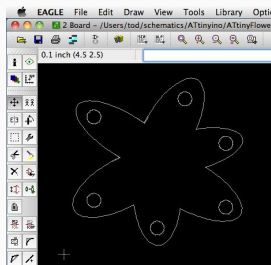
Turning **display on** will overlay a grid to assist in your layout.

The Board Window

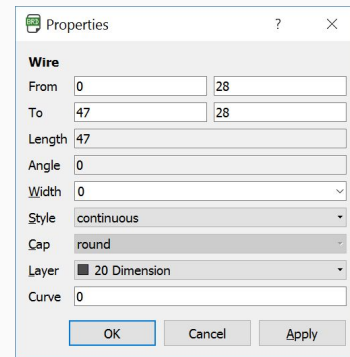
To start off, all of your schematic components will be located off the board. The perimeter of your board is defined by the thin grey outline.



Defining Board Dimensions



To modify your board size, edit the
Properties of the board outline.



Alternatively, define your own custom borders by
using the **Wire** command and drawing wires on
the **Dimension** layer.

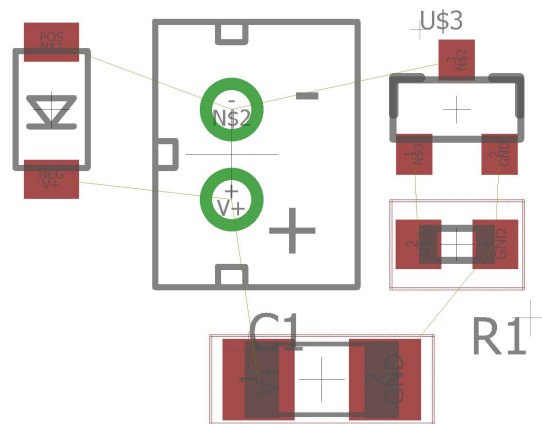
Arranging Parts

Place parts with the **move** and **rotate** commands

Airwires show connections between parts as defined in your schematic.

Arrangement should be logical and compact. Place parts to **minimize trace lengths**.

Use **ratsnest** to reevaluate airwires.

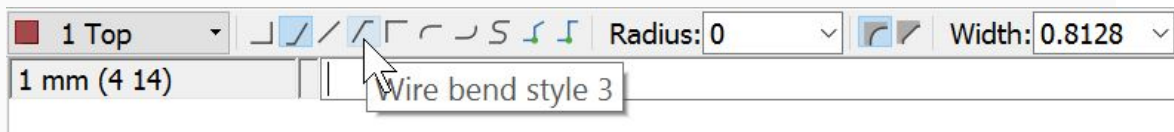
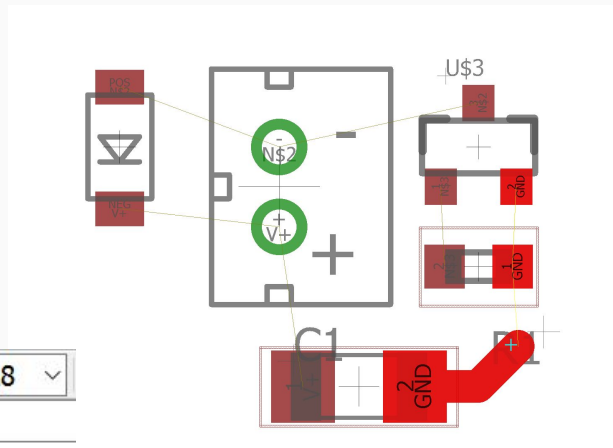


Routing the Traces

Use the **route** command to begin routing connections between components.

The **ripup** command deletes paths.

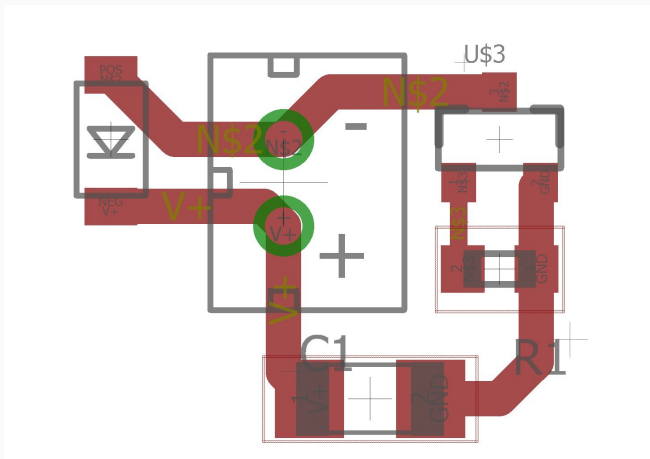
Ensure that you use **45° angles** in traces.



Routing Tips

When defining paths for traces, click to define start, end, and intermediary points.

Trace width depends on the function of the trace. Traces carrying digital signals should be at least **12 mil**. Traces carrying analog signals should be at least **20 mil** to preserve signal integrity. Traces with large currents (e.g. power, ground, and motor traces) should be as thick as possible (**30+ mil**).

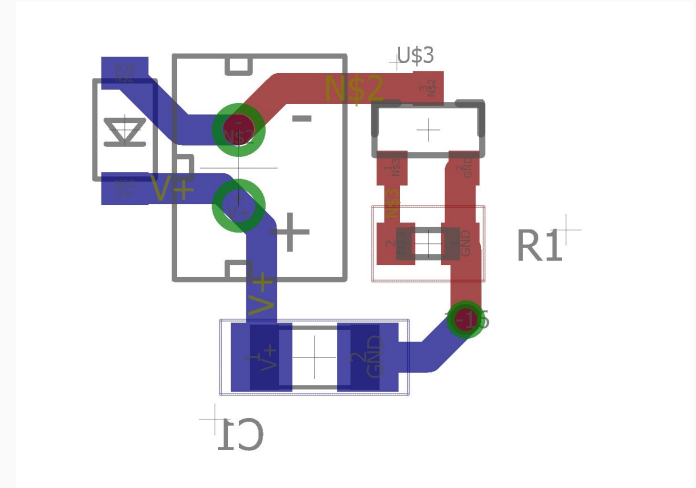


Multi-Layer Routing

Use the **mirror** command to flip a component over to the other side.

To make a trace that connects both sides, begin drawing a trace and then use the toolbar to switch layers. Then choose where you want to place your via (connection between two sides of the board), and begin tracing on the new side.

Red indicates top placement. Blue indicates bottom placement. Green indicates copper-plated through-holes.

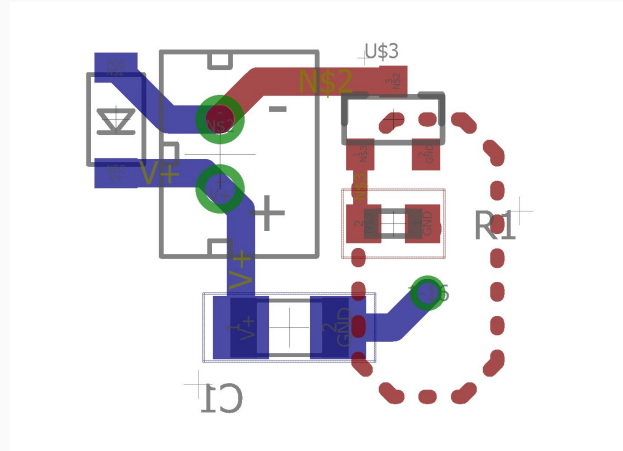


Polygons

If multiple components are connected on the same net, consider using a **polygon, or plane**, to connect them.

Often used in power/ground planes or to create a **low-impedance path**.

Defined by borders rather than a path (in contrast to traces).

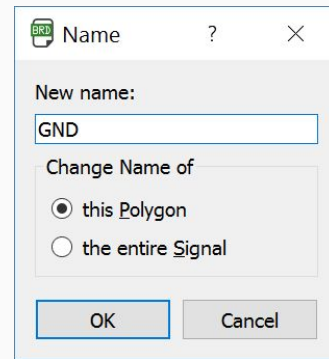


Creating a Polygon

Use the **polygon** command on the top or bottom layer to create a plane.

Click to define borders for your polygon. Double-click to finish.

To specify the signal the polygon should connect to, use the **name** command to name the polygon.

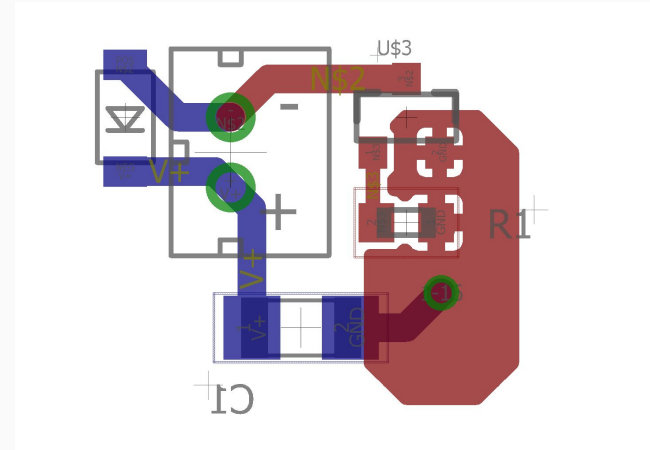


Creating a Polygon (cont.)

View your filled polygon using the **ratsnest** command.

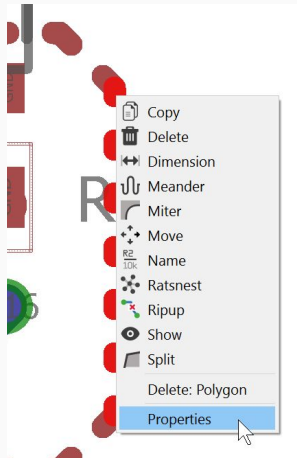
Notice the cross hatching between pads and planes. These **thermals** make it easier to heat up the pads when soldering. Changing the width property of a polygon will change the thickness of the thermals. For polygons with high current, increase the width, or turn off thermals for that polygon in properties.

To return to viewing just the polygon borders, use “**ripup @;**”.



Polygon Rank

If polygons have overlapping borders, use the **rank** attribute to specify which takes precedence.



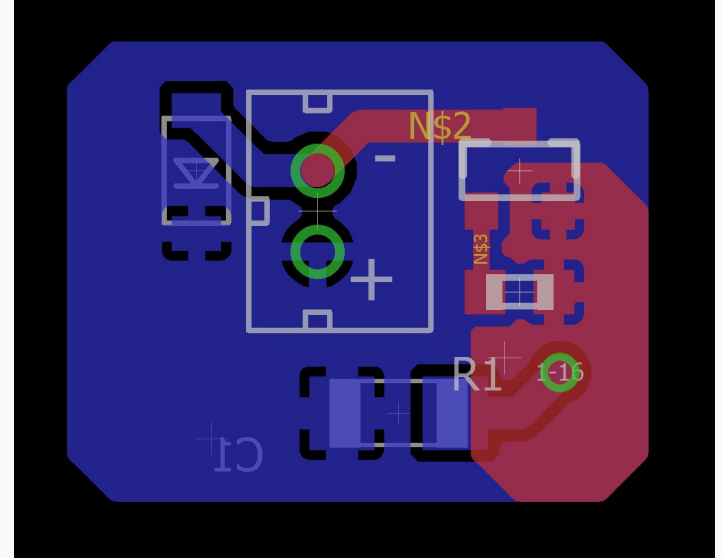
Polygon	
Polygon Pour	<input type="text" value="solid"/>
Spacing	<input type="text" value="1.27"/>
Isolate	<input type="text" value="0.3048"/>
Rank	<input type="text" value="1"/>
<input type="checkbox"/> Orphans	
<input checked="" type="checkbox"/> Thermals	

Final Board

An additional polygon on the bottom layer for V+

Trace lengths are minimized

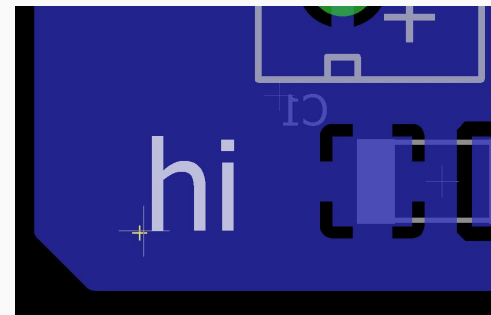
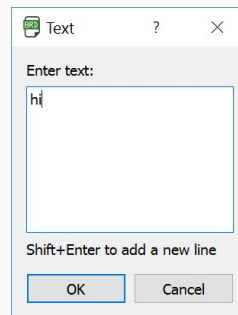
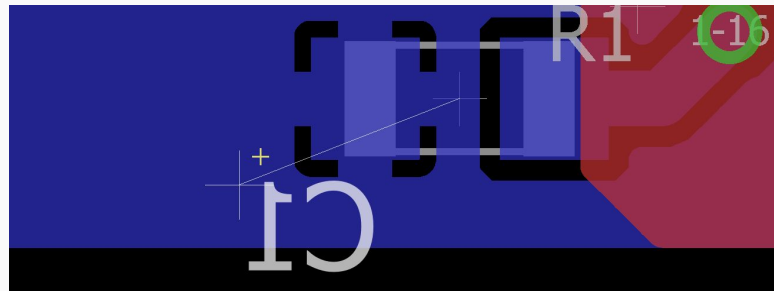
More effective use of available space



Silkscreen

Most components will have name/value labels associated with them. If you want to move these labels from their default positions, use the **smash** command to separate the label, and then **move** to reposition them.

Use the **text** command to add custom text to your PCB. Text should be placed on the silkscreen **tPlace** and **bPlace** layers (21 and 22).



Decoupling Capacitor Placement

- Recall: ICs require decoupling capacitors
 - Filters out power supply noise
 - Stores charge to handle current spikes
- Decoupling capacitors should be as close to the IC as possible
 - Parasitic inductance affects the speed at which charge can be delivered from the capacitor
- When using multiple capacitors, the smallest capacitances should be closest to the pins
 - Small capacitors work better at high frequencies

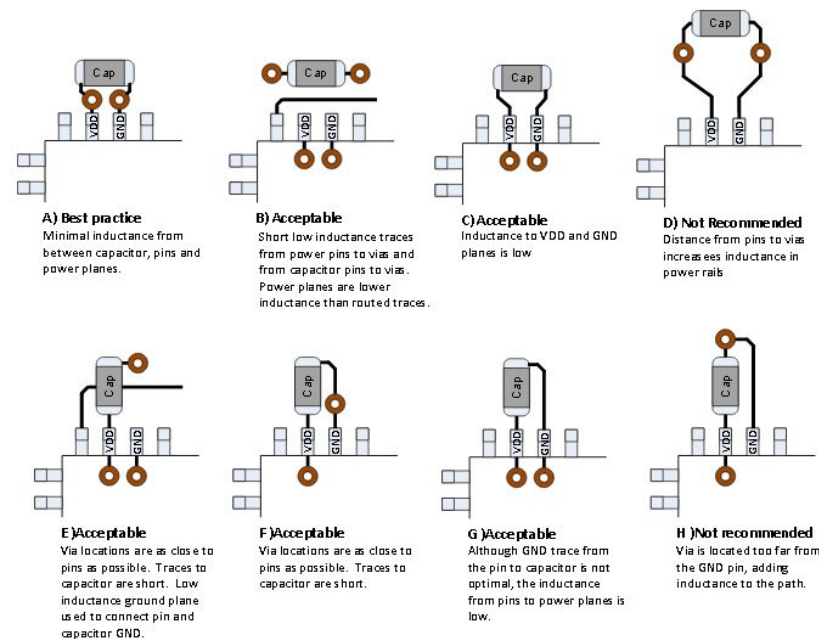


Figure 18. QFP PCB Routing Options

Ground Isolation

- Motors introduce noise into the power and ground planes, causing other subcircuits to malfunction
 - Digital ICs also introduce some noise, which can disturb analog circuits
- Solution: create three separate ground planes (motor, digital, and analog), which combine at a common reference point
 - Digital and analog planes should combine near the voltage regulator
 - Motor ground should only connect to the other grounds near the battery terminal
 - Isolation also applies to power planes

