

PCB Layout Notes

1.0 Introduction

The following is intended to serve as a collection of short but useful tips, tricks, shortcuts, and other advice to hasten and improve the student PCB layout experience.

2.0 PCB Layout Notes

1. Prior to routing, take plenty of time to carefully place components in proper places on your circuit boards. Careful part placement is vital to producing optimal board layouts. Place high-priority parts (oscillators, high-speed interfaces such as USB, etc.) first to ensure that high-priority routes are optimized. Group related parts together (for example, all power supply and battery monitoring parts should generally be grouped closely together). In general, the microcontroller (and/or FPGA, CPLD, etc.) should be centrally located for convenient routing to all peripherals
2. While the autorouter feature of layout tools such as Eagle can be used to route traces and/or boards quickly, it often does so at great expense to electrical performance and area efficiency. Better results can nearly always be obtained through hand-routing, and hand routing is very strongly recommended for ECE477.
3. PCB layout and routing of circuit board traces are iterative processes. It is common to finish an initial draft of a layout and then create a new layout from the previous draft, improving on flaws and suboptimal aspects of the draft in the new board. It may take 2, 3, or even more iterations to produce a high-quality PCB layout that performs well.
4. Where specified, you should always follow manufacturer layout guidelines. These guidelines, if they exist, are typically discussed near the end of the datasheets for a given parts. Certain parts, such as oscillators, switching power supplies, and wireless transceivers, are very likely to have layout guidelines. ALWAYS leave the area beneath and immediately around your oscillators free of other traces (don't want to inject noise into the clock circuitry of ICs)
5. Route high-priority traces first. High-priority traces include any oscillator or clocking circuits, power supply/regulation circuits, and high-speed interfaces (such as USB).
6. Octagonal mode (trace angles constrained to 135-degrees) is strongly recommended for PCB routing. This will produce clean, professional results and generally minimize routing issues.

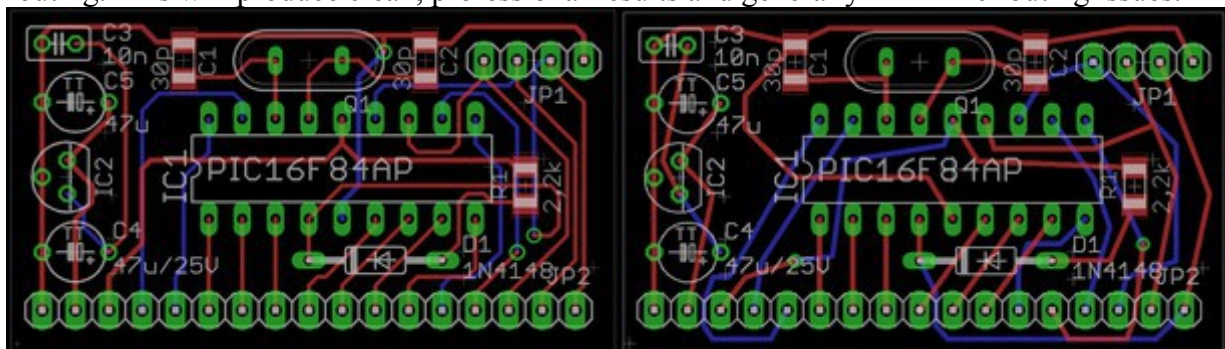
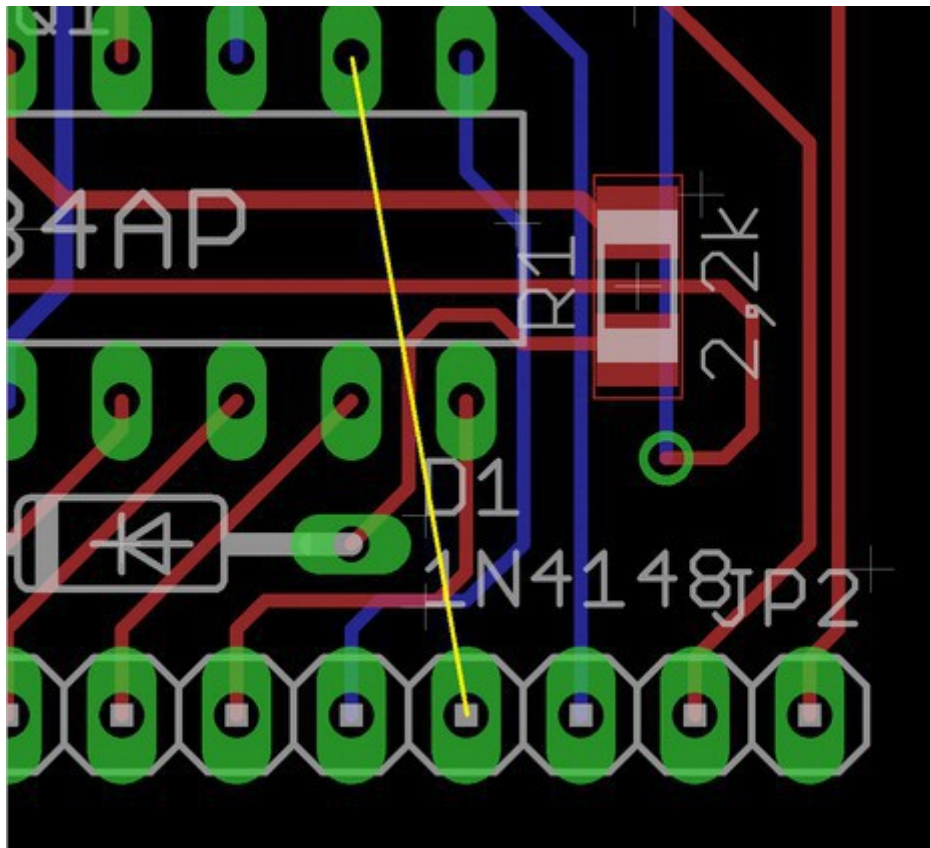
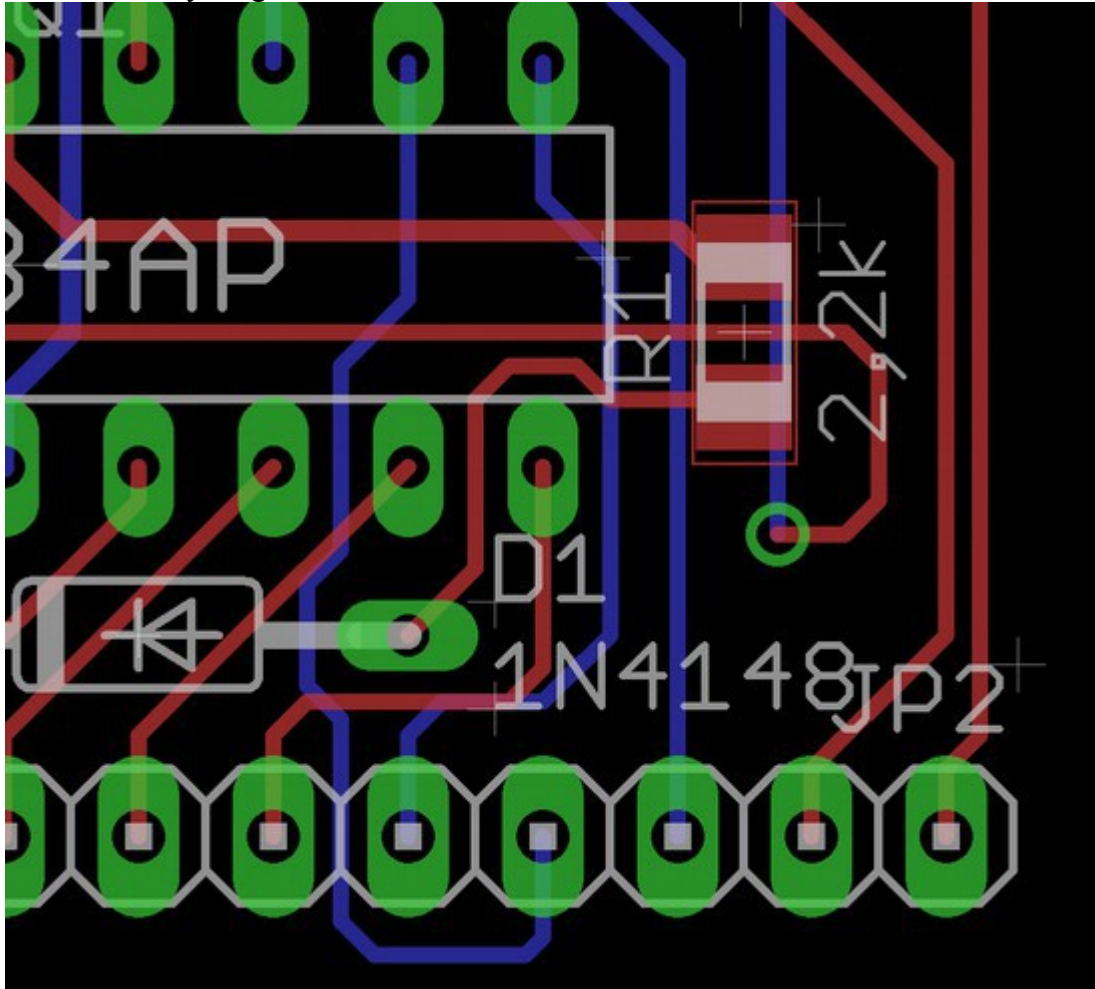


Fig. 1. Octagonal Trace Routing (right) vs. Arbitrary Routing (left)

7. Whenever possible, 90 degree angles should be avoided. It is acceptable to enter traces, pins, pads, and vias at 90 degree angles, however. AVOID acute angles (angles < 90)
8. Vias are necessary for routing signals between the top and bottom layers of a circuit board. Like 90 degree angles, vias are sometimes needed but should be minimized wherever possible
9. Vias and other holes in a circuit board contain a ring of metal around them known as an annular ring (for vias) or pad (for through-hole parts). These rings exist on both the top and bottom of the board; be careful not to short traces together through the copper in pads and annular rings.
10. Eagle CAD requires both the schematic (.sch) and board layout (.brd) files open simultaneously in order to perform forward and back annotation. Always have both files open; do not work one while the other as closed, as this can mess up Eagle's synchronization and lead to redesigns.
11. Always choose appropriate trace widths and via drill sizes based on the amount of current flowing through a given trace or via. A useful reference for current carrying capacity can be found here: <http://circuitcalculator.com/wordpress/2006/01/31/pcb-trace-width-calculator/> (note: for ECE477 PCBs, 1oz copper should be specified for the "Thickness" field)
12. When routing signals, it is possible to route some (10-16 mils) traces between the pads of headers, surface mount parts, and some large (metric size 1206 or greater) SMD passive components. This can assist in routing traces efficiently while minimizing overall trace lengths.
13. Signals can be sometimes routed optimally by routing around one another prior to routing the signals to the destination. Consider the circuit below:



Traces on the top and bottom sides of the board prevent a minimal, direct route to from the source pad to the destination. However, by routing through the bottom of the bottom pad and bypassing the bottom-side copper trace, a successful route can be made without vias and without excessively long traces:



14. Routing is an exercise in applied geometry, and practice is helpful for discovering patterns and sequences that lead to efficient routes. The mobile flash game “Flow” may be a useful exercise for people seeking to improve their spatial abilities and produce more efficient routes. At the time of this writing, flow can be found here: <http://www.moh97.us/>
15. Eagle contains many keyboard shortcuts; below are a few that can substantially expedite the layout process:
 - Left click (while routing airwire) – Toggle between various routing styles (left to right)
 - Shift + Left click (while routing airwire) – Toggle between various routing styles (right to left)
 - Depress mouse scroll wheel/center click (while routing tool selected) – toggle between various copper layers. If this is done mid-route, it will automatically drop a via between layers of the shape specified by the shape modifier of the routing tool
 - Ctrl + Right click (while routing tool is selected) – This combination allows a route to start from an arbitrary point along a trace, as opposed to the given ends of an airwire