Electronic System Design: From Prototype to PCB

1. Schematic Capture
2. Good Design Practice
3. Custom Components
4. Printed Circuit Board (PCB) Layout

Electronic Systems Design - Reflection

Electronic Systems Design will be used as an example.

Self-Reflection

What do you already know about PCB design?

What software tools are you comfortable with? Do you remember Eagle?

When is it time to move from prototype to PCB?

What is good PCB design practice?

Systems Design - Solution Deployment

1. Schematic Capture
   - Structuring the Schematic
   - Minimum Circuitry - Datasheets
   - Using Net Labels
2. Good Design Practice
   - Decoupling Capacitors
   - Design for Manufacture
   - Reflection
3. Custom Components
   - Schematic and Footprint Connection
   - Making Custom Components
4. Printed Circuit Board (PCB) Layout
   - Utilising the Ratsnest
   - Routing Tracks
   - Ground / Power Planes

Schematic Capture - Structuring the Schematic

Develop the schematic of the system in modules. These may be functional blocks, or components of the system that are heavily inter-related that will be located near eachother on the PCB. A structure may involve separation of:

- Power supply with voltage regulators and decoupling capacitors
- Modules that share common communication busses

It is good practice to have information flow from left to right, and top to bottom. Having a structure such as: connections to central elements of the system are on the left, and peripheral connections are on the right, will make the schematic easy to read and troubleshoot.

Schematic Capture - Minimum Circuitry

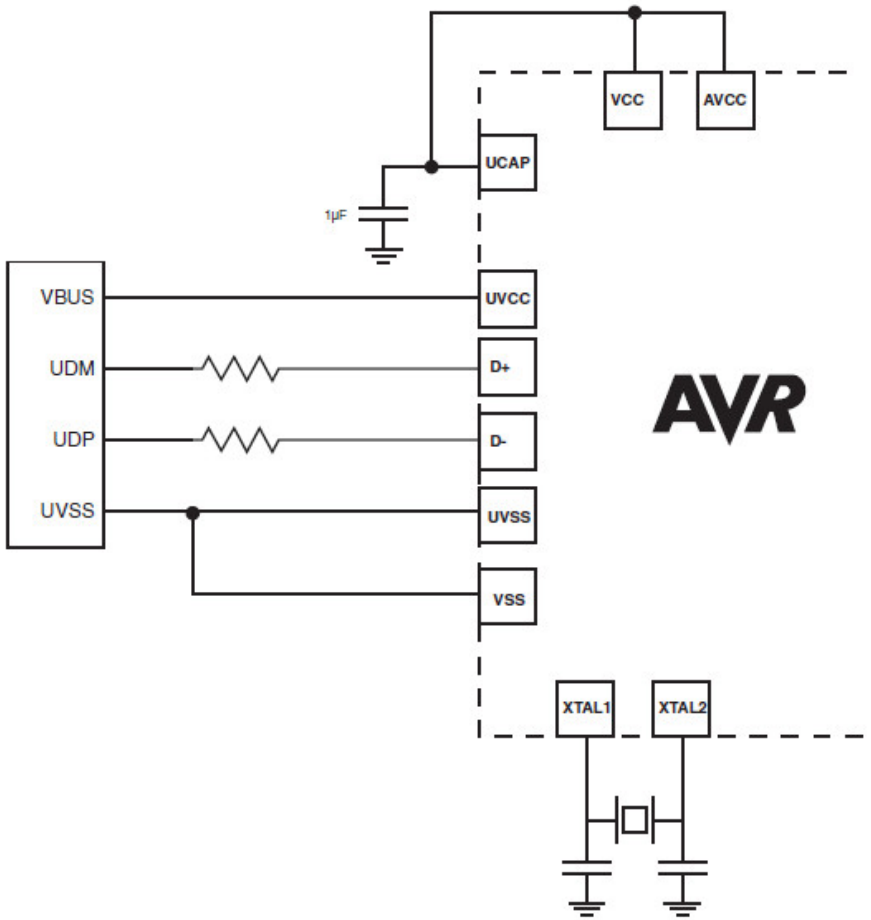
Focus first on achieving the minimum circuitry for each module before connecting them together.

Datasheets typically have a minimum connection diagram (though they are not always obvious to find)

What basic functionality is required before all features are added?

Schematic Capture - Minimum Circuitry

Figure 20-4. Typical Bus powered application with 3.3V I/O



Schematic Capture - Using Net Labels

In a Schematic, a group of wires that connect to a common node are known as a net.

Rather than connecting all the wires in a net, they can be joined by labels. This makes the schematic much more tidy, and easy to interpret.

Use a logical naming convention to ensure that all of the required nets are connected.(Depending on software they can be case sensitive)

Printed Circuit Boards - Good Design Practice

PCB - Good Design Practice

- How can design for contingency/future proofing be incorporated?
- What electrical considerations should be made?
- What else needs to be considered at the schematic stage, and how can it be revised?
- What is good design practice that you can't find in a datasheet?
- How do you make the circuit easy to troubleshoot? (eg. LED power light)

Printed Circuit Boards - Decoupling Capacitors

Decoupling capacitors are placed between power and ground, near the power supply pins of a chip.

They prevent electrical noise causing the voltage to drop, which would cause chips to 'brown out'

Decoupling Capacitors

Consider the following rules of thumb:

- Decoupling capacitors should be connected as close to the power pins as possible.
- Have 10 μF near a μC power junction, and a (0.1μF + 10nF) combo at any other power pin.

http://electronics.stackexchange.com/questions/15135/decoupling-caps-pcb-layout/15143#15143

https://www.youtube.com/watch?v=G7ULnQ9i7H0

Printed Circuit Boards - Design for Manufacture

PCB - Design for Manufacture

- Are all of the resistor / capacitor variants required, or can a smaller selection be used?
- Are the pads accessible for soldering / assembly?
- Can peripherals be connected easily once the PCB is mounted in its destination?
- How will the board be assembled?
- Which manufacturing technology will be used? (Eg. Milled or etched PCB, Solder resist, pick-and-place, or hand assembly)

Printed Circuit Boards - Reflection on Design Methods

Consider, how will you develop your concepts into a design for:

- Contingency / Future-proofing
- Modularity / Scalability
- Serviceability / Repair
- Manufacture (As discussed on previous slide)
- Troubleshooting (Eg. Testpads)
- Best electronic practice (Eg. Star Configuration)

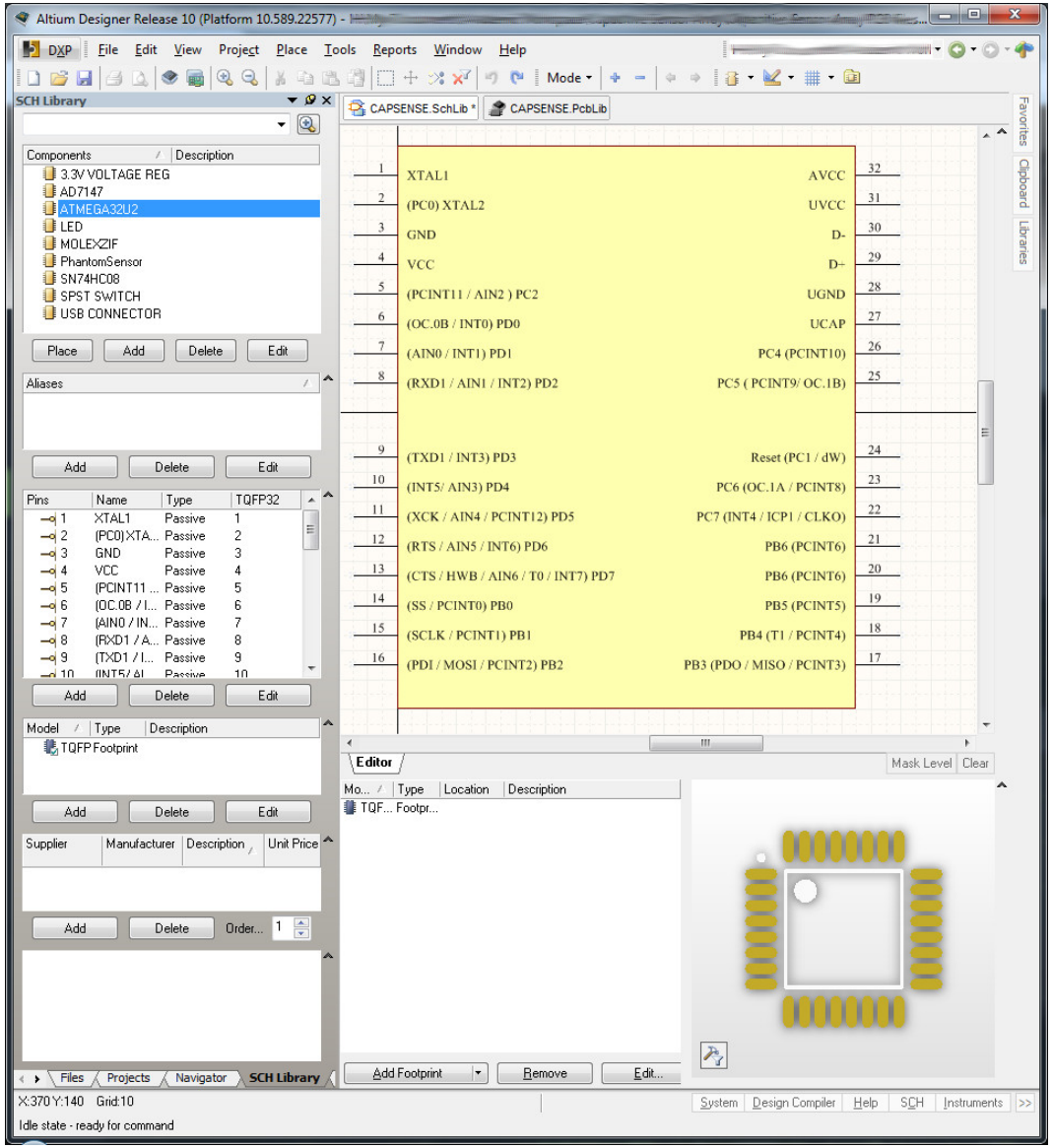
Printed Circuit Boards - Custom Components

Components consist of a schematic symbol, and a PCB footprint. They connect the conceptual "net" to a physical location (a solder pad).

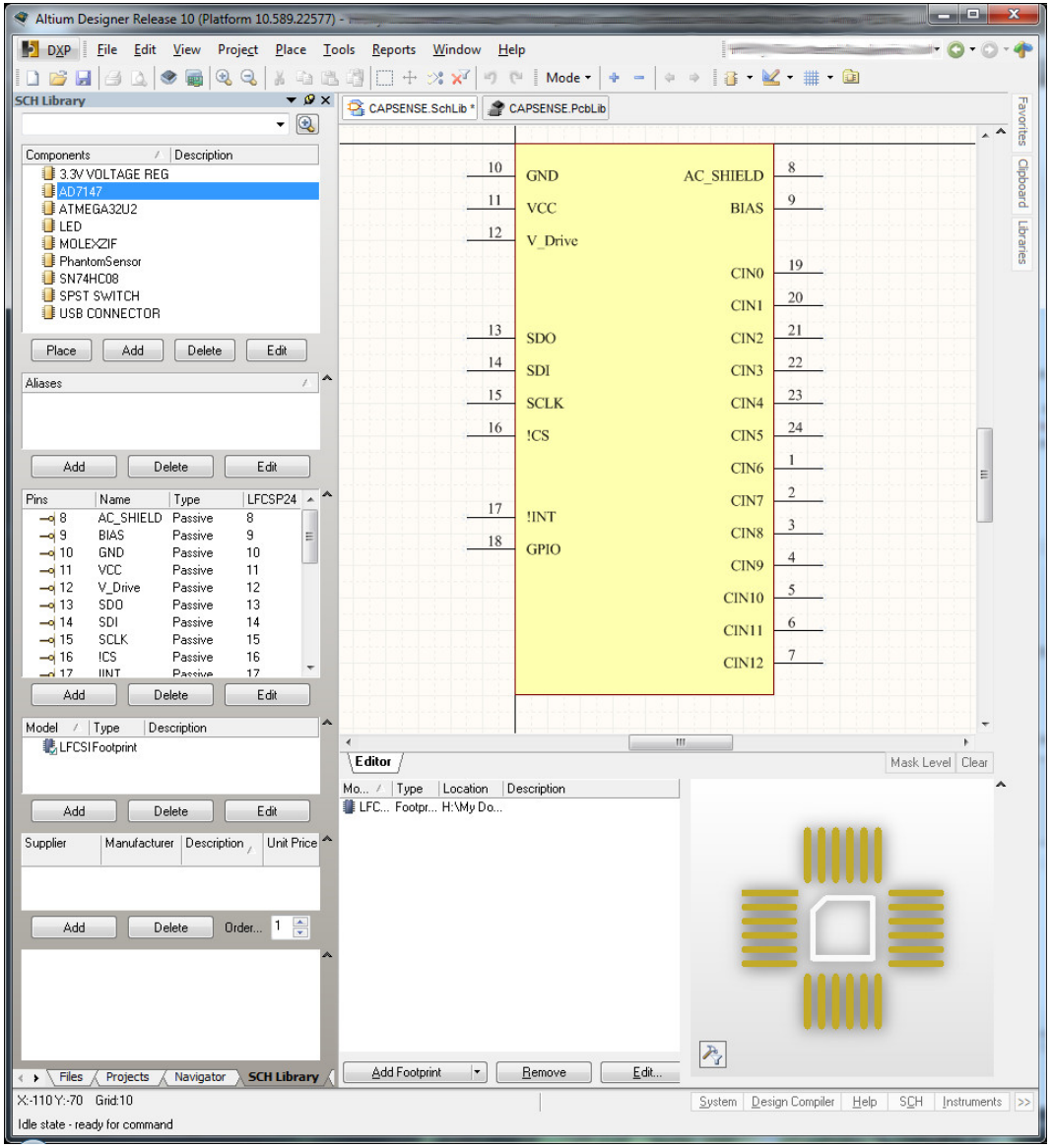
The symbol and footprint are typically designed seperately because a single component may be available in multiple different footprint formats. Similarly, standardised footprints may be common across different components.

The symbol (schematic representation) and footprint (PCB layout representation) are then combined together to create a component that is then stored in a library.

Printed Circuit Boards - Custom Components



Printed Circuit Boards - Custom Components



Printed Circuit Boards - Layout

In order to achieve a logical design, and an efficient deployment, a structured approach is required for PCB Layout

PCB Layout - Structured Approach

- Begin with the most central element of the system (typically the microcontroller)
- Consider the direction of comms busses
- Place the major components of peripheral modules in the direction they are desired to be located
- Consider which components should be near eachother
- Use the Ratsnest feature to decipher the best orientation

PCB Layout - Utilising the Ratsnest

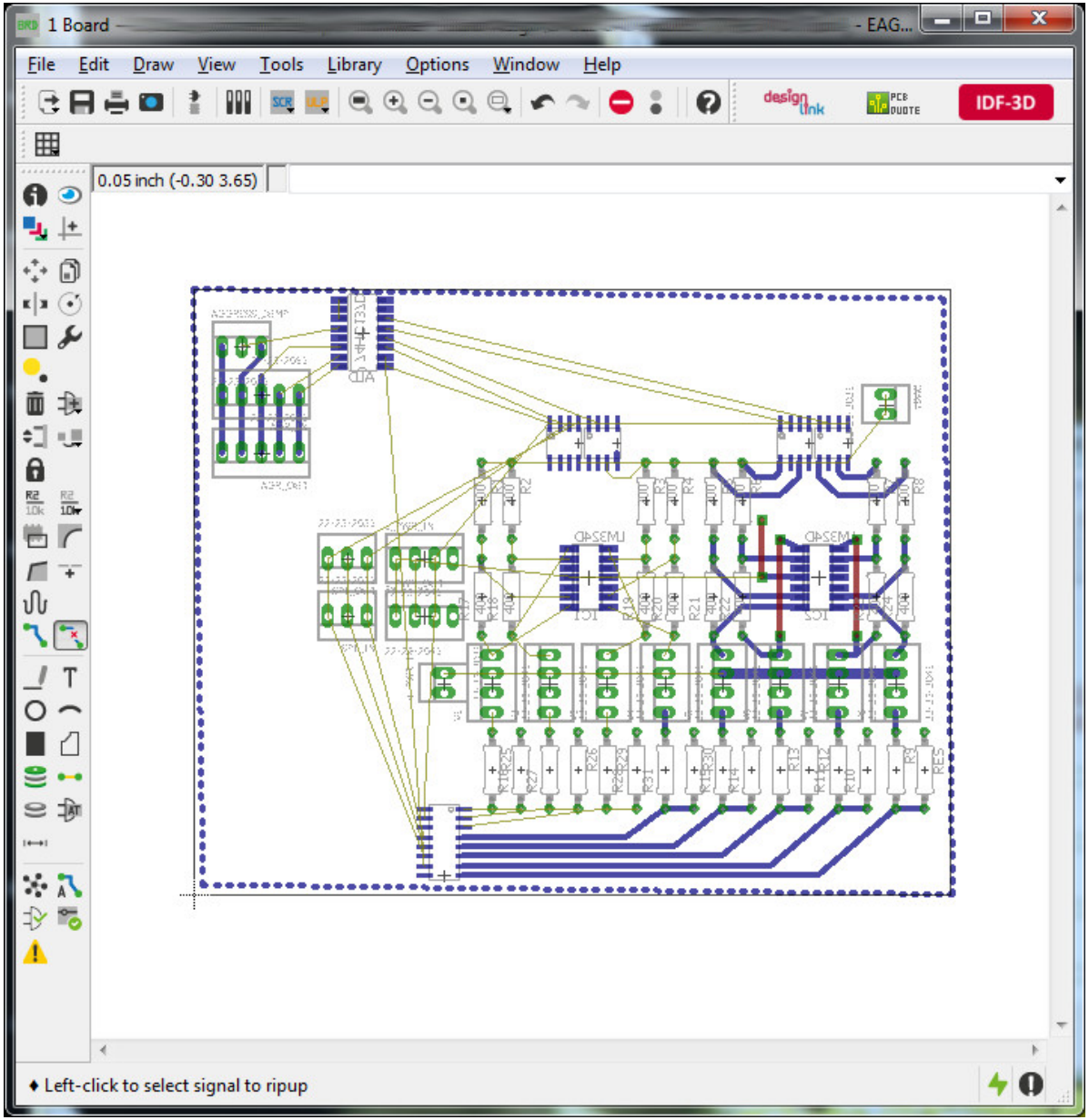
Ratsnests are a useful tool for orientating components, and for refining the PCB schematic.

Non-critical routing can be optimised by altering which communication busses are used. Peripherals may be remapped to make this easier. Repeat this Ratsnest/revision process multiple times.

Untangle as much as possible before routing anything.

With the main components in place check that the schematic and PCB are correct and synchronised

PCB Layout - Utilising the Ratsnest



PCB Layout - Routing Tracks

Begin with critical components such as communication busses and signals.

Ignore ground connections as these can be achieved though a ground plane (discussed later)

Consider the amount of current that needs to flow through the track (Eg. Use a suitable width)

Save decoupling capacitors and pull up resistors for last - leave room for them, but they are usually the easiest to place

PCB Layout - Ground / Power Planes

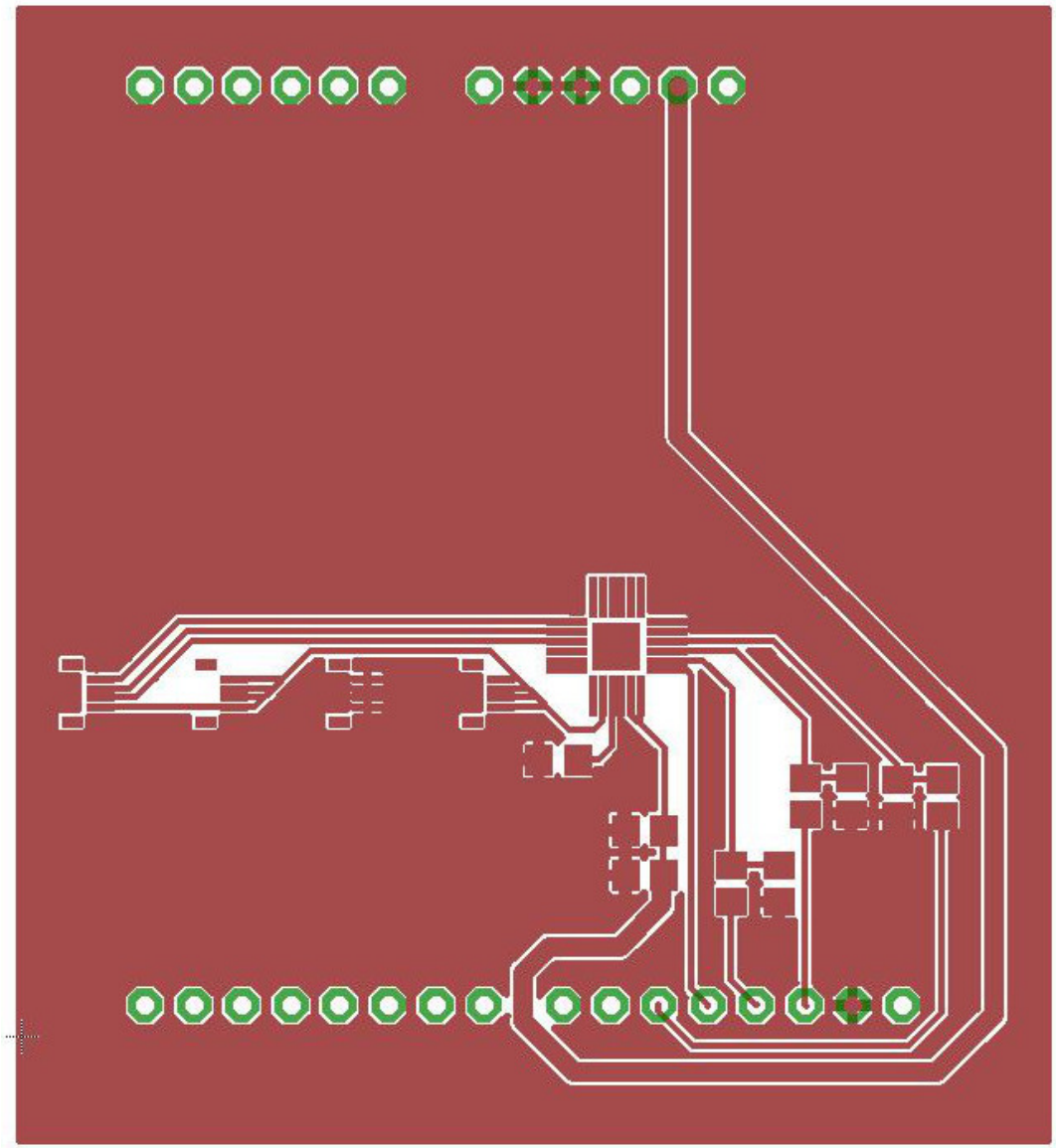
The remaining copper left on your PCB (excluding traces) can be used as a ground or power source.

Known as a "pour" or a "plane" (Eg. Ground pour, or Ground plane)

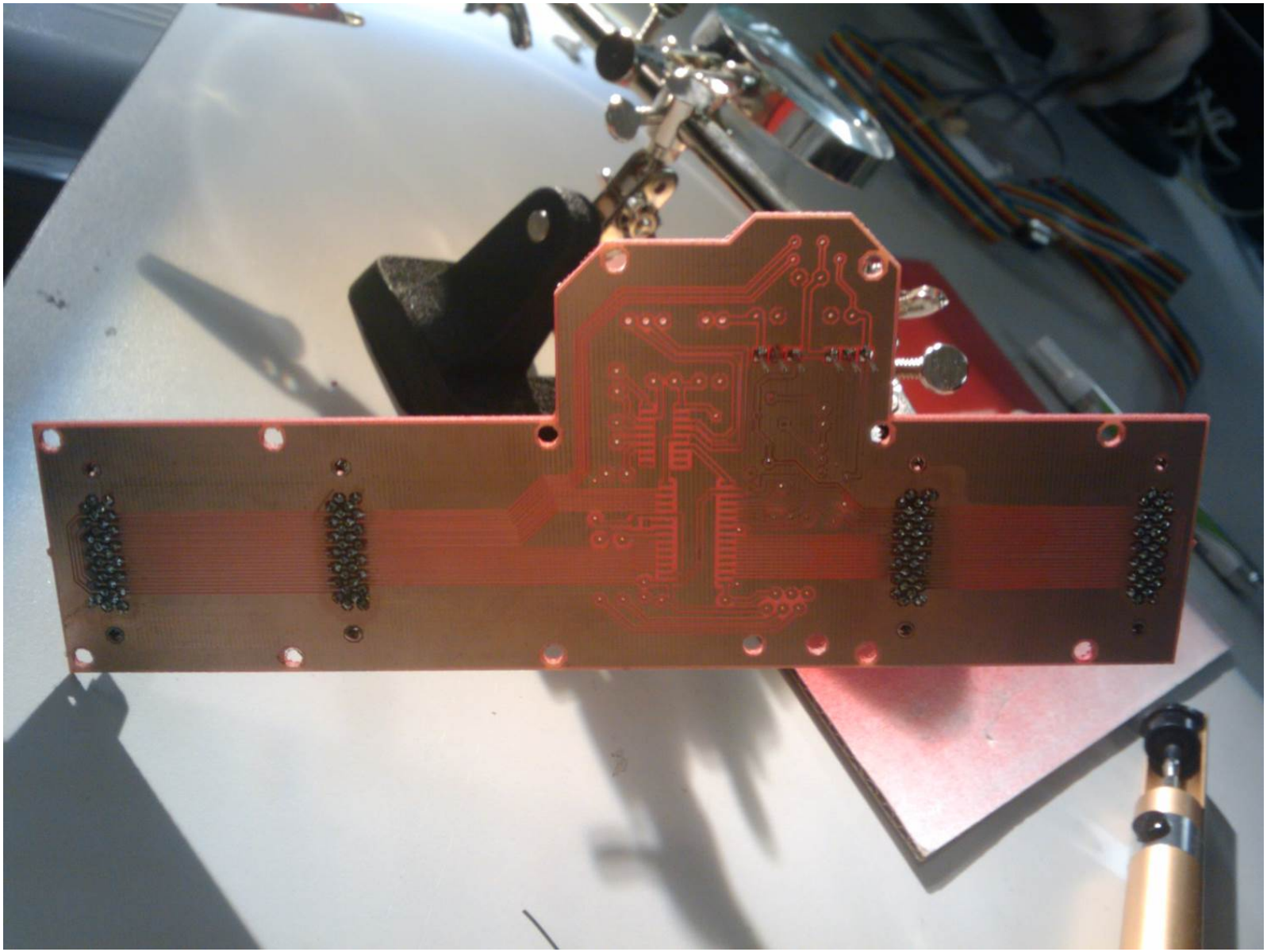
Ground / Power planes are useful to help with routing and contribute to the stability of the voltage reference.

Ground plane saves manufacturing cost and time (Copper does not have to be mahcined or etched away, which cost time and money resources)

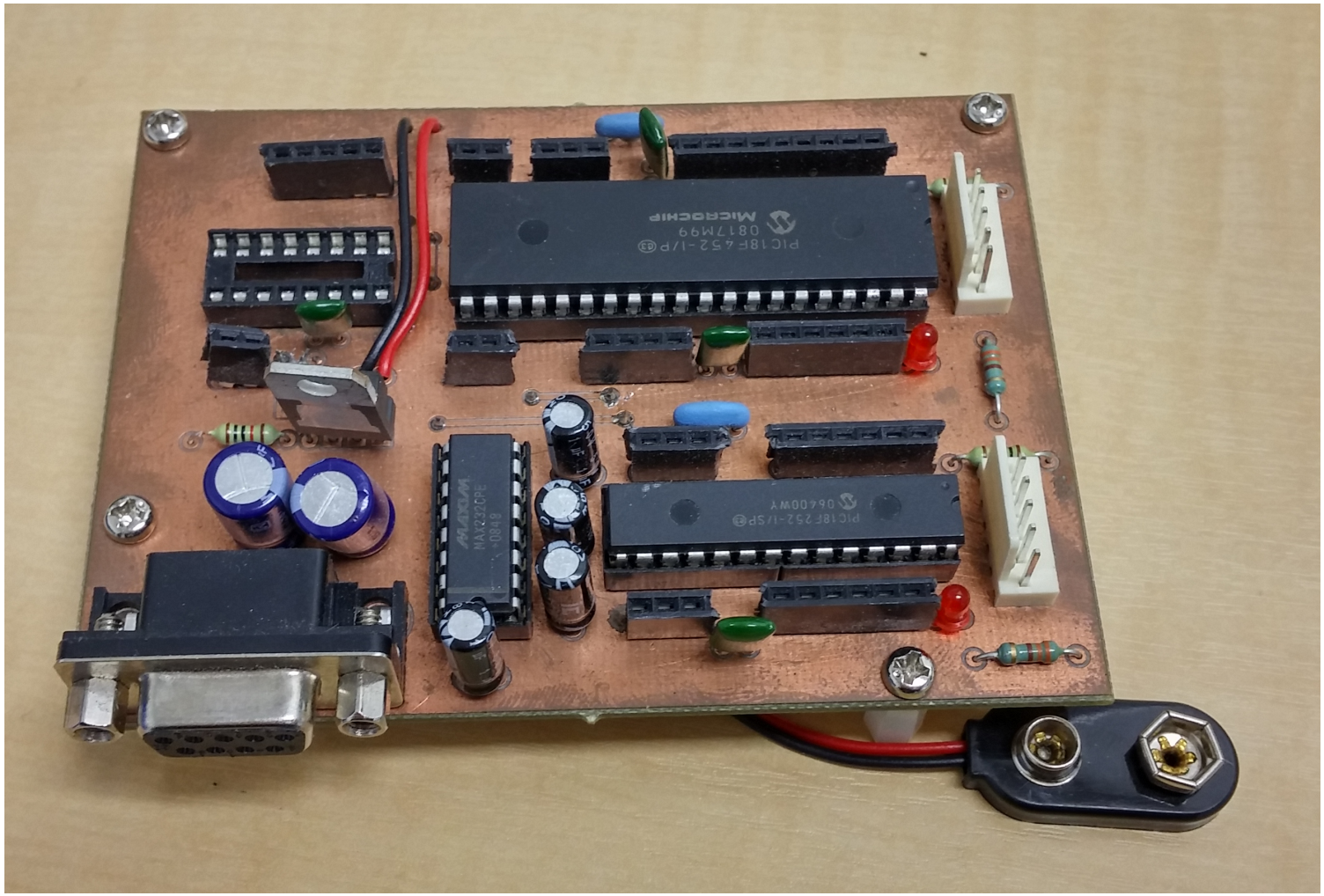
PCB Layout - Ground / Power Planes



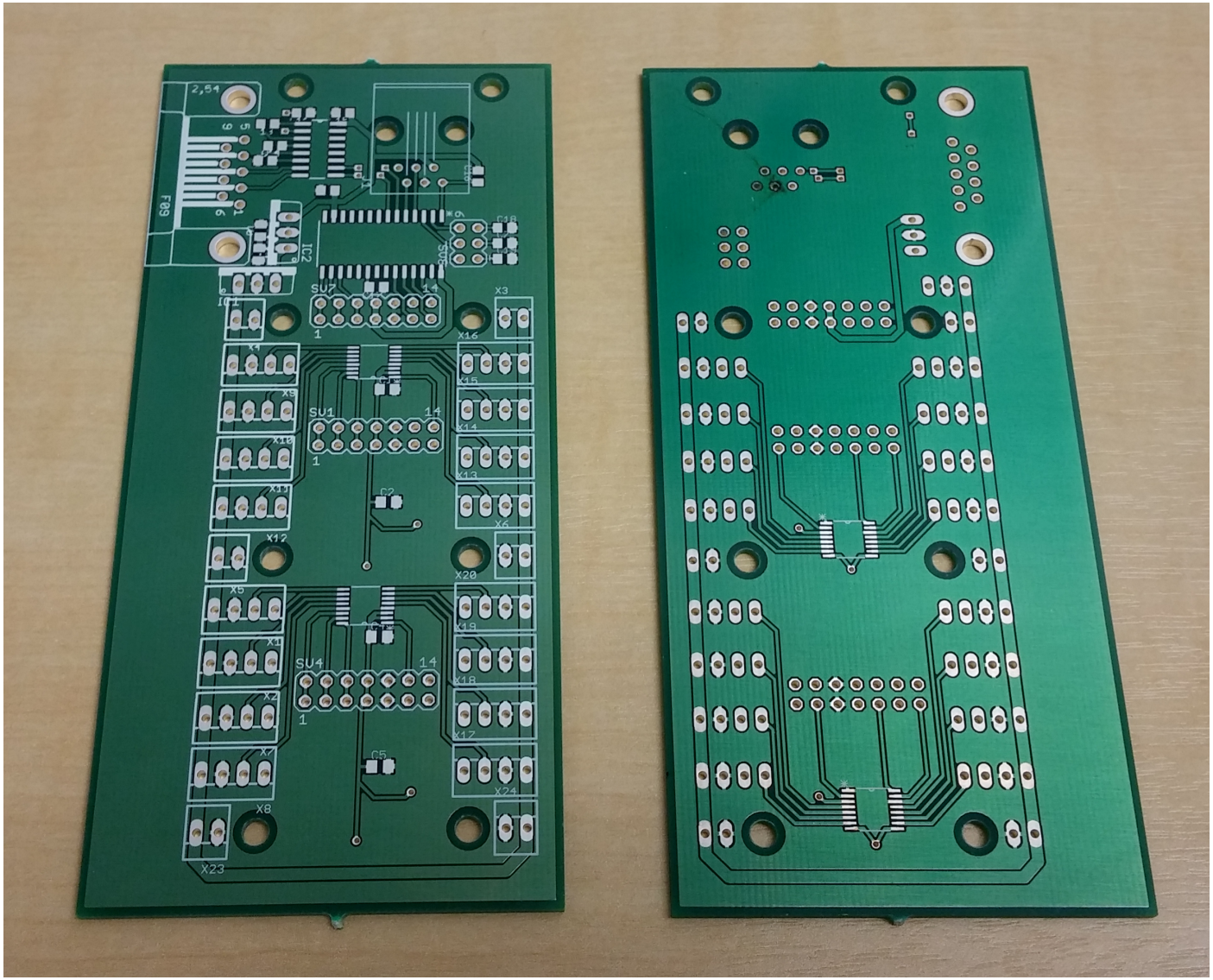
PCB Layout - Reflection



PCB Layout - Reflection



PCB Layout - Reflection



PCB Layout - Reflection