Designing a PCB

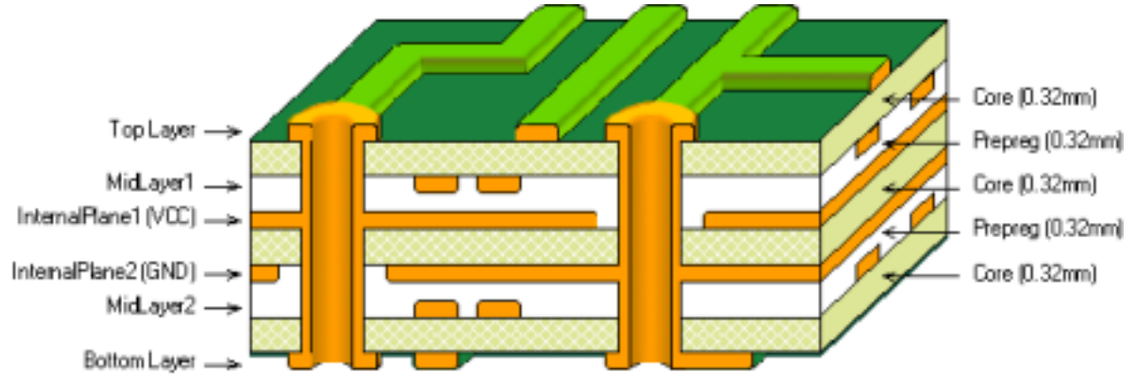
EE155/255 - Fall 2015

Ned Danyliw

Why a PCB?

- Permanent, repeatable (hopefully) circuit
- Lower parasitics
  - Necessary for high speed designs
- Smaller form factor

Cross-section of a PCB



Designing a PCB

- Design the schematic
- Select the components
- Place components on PCB
- Route signals and power planes
- Send design to manufacturer
- Assemble

Software

- Free software
    - Eagle
        - Probably the most popular
        - Free
            - Huge library of pre-made components
            - Free version - 2 layer and limited PCB size
    - FreePCB
    - KiCAD
- Paid software
    - Altium
        - Prohibitively expensive for personal projects
        - Much easier to route large designs

Schematic Design

- Flexibility is good. You don’t need to populate every component.
- Make sure components are properly bypassed
    - Typically a .1µF capacitor on their supplies
- Clearly label parts
- Named nets are good
- Organize schematic logically

Component Selection

- Build bill of materials with components supplier
  - Preferably a single supplier
    - Digikey, Mouser, Jameco, etc.
- Be aware of package size/code
- Always check the datasheet
- Order typically 3x

Board Layout Pt. 1

- Most important step in designing a PCB
- Intelligent placement makes routing easy
- Typically can place circuit in “blocks”
- Keep components close together (shorter traces typically are better)
- Keep power dissipation in mind (spacing/trace widths)
- Clearance for connectors
- Separate “noisy” and “quiet” signals
- Add test points

Board Layout Pt. 2

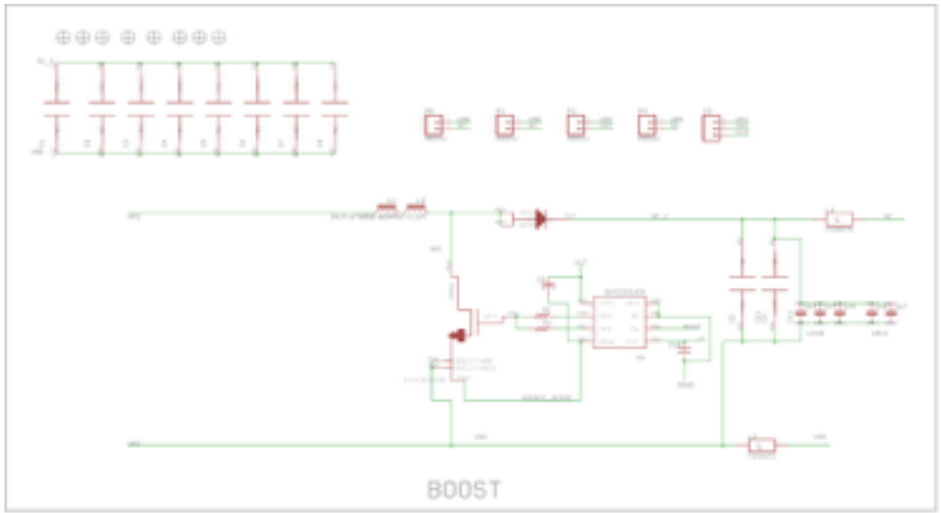
- Route the critical signals (critical power loop, high speed digital, analog signals, etc.)
- Don’t autoroute*!!!!!!
- Often power signals should use polygons
- In general traces should be as wide as they possibly can
- Run DRC (design rule check) often!
- With multiple layers, map what each layer (generally) is for

*can use autoroute tools for some signals but you need to know what you are doing

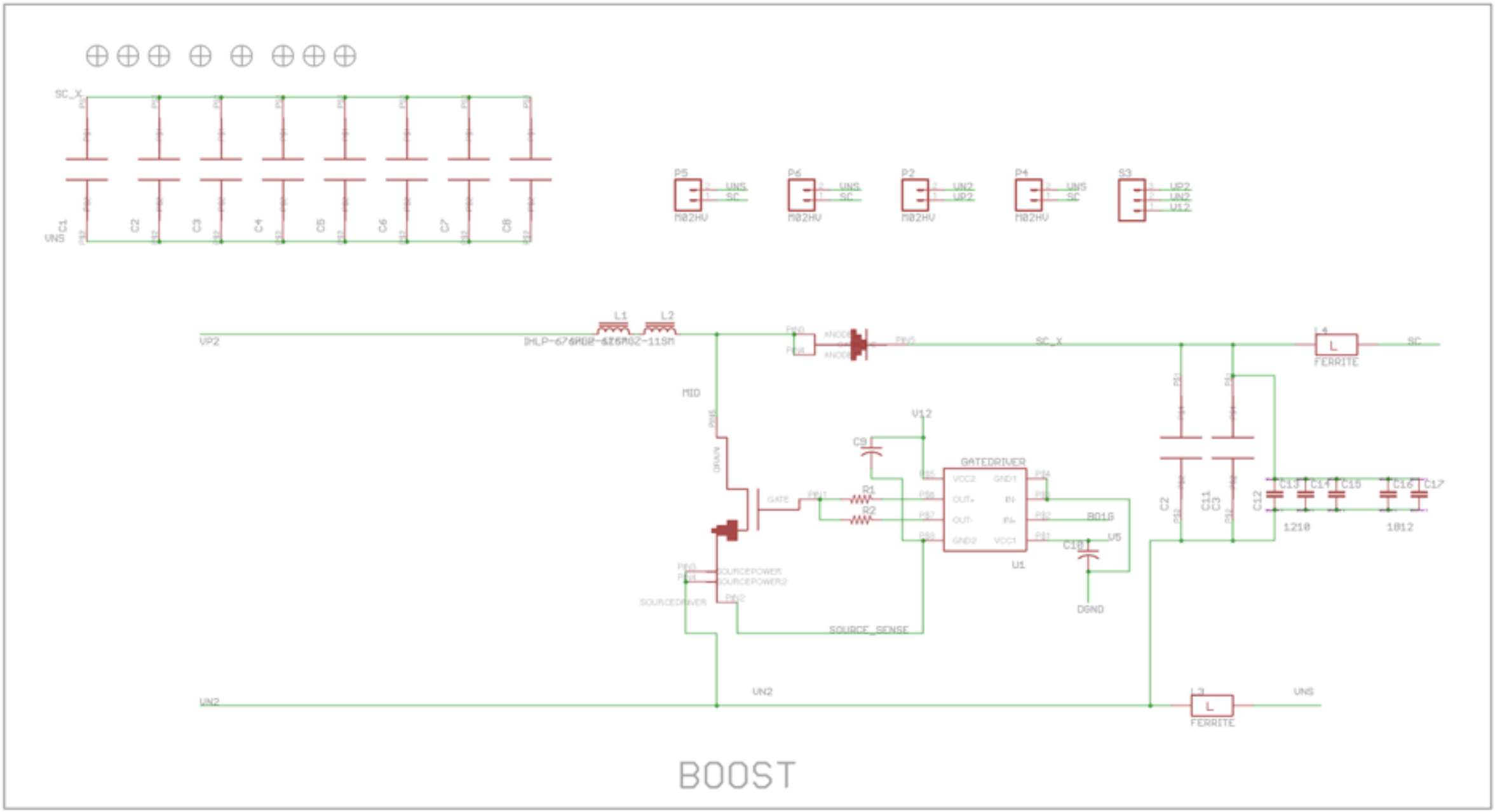
Design Example

- Microcontroller board for littlebox
- STM32 microcontroller
- Instrumentation amplifiers for sensors
- Boost converter
- Interface with 3 other boards

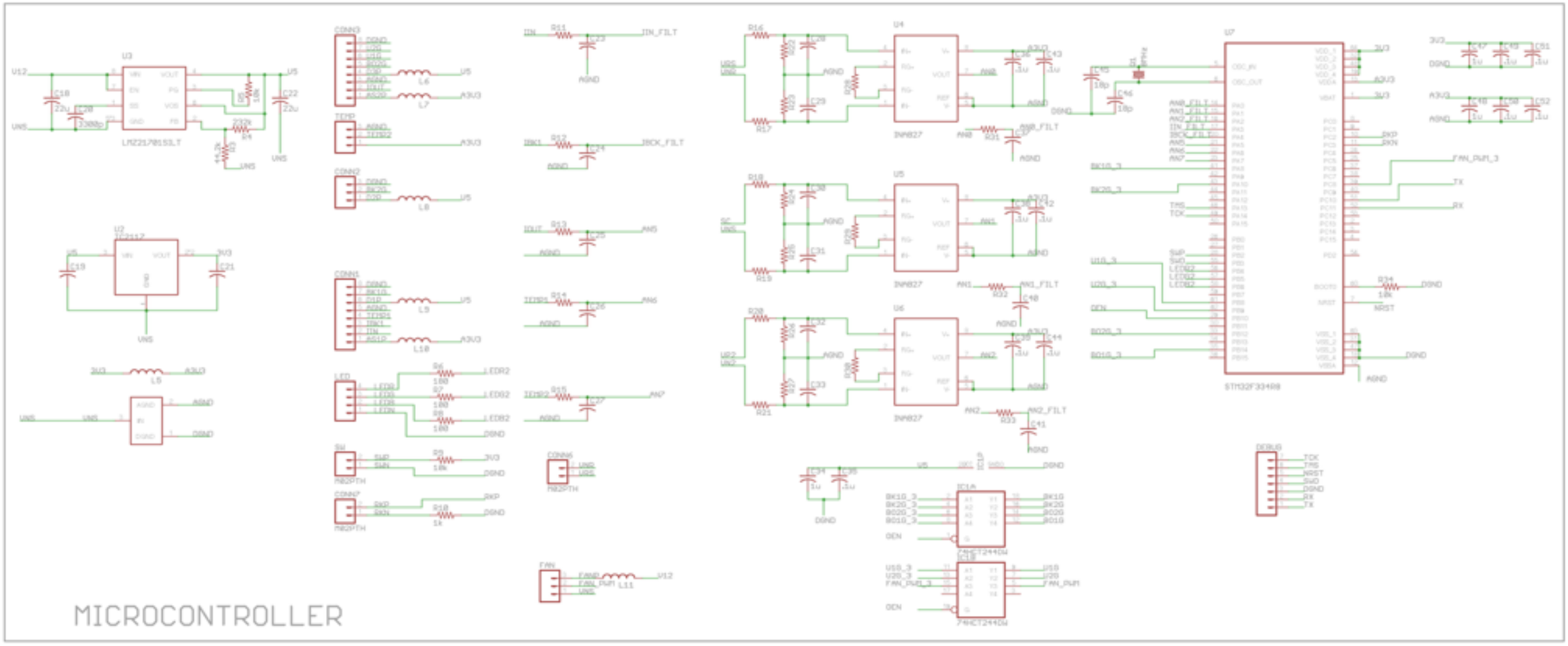
Schematic



Boost Schematic



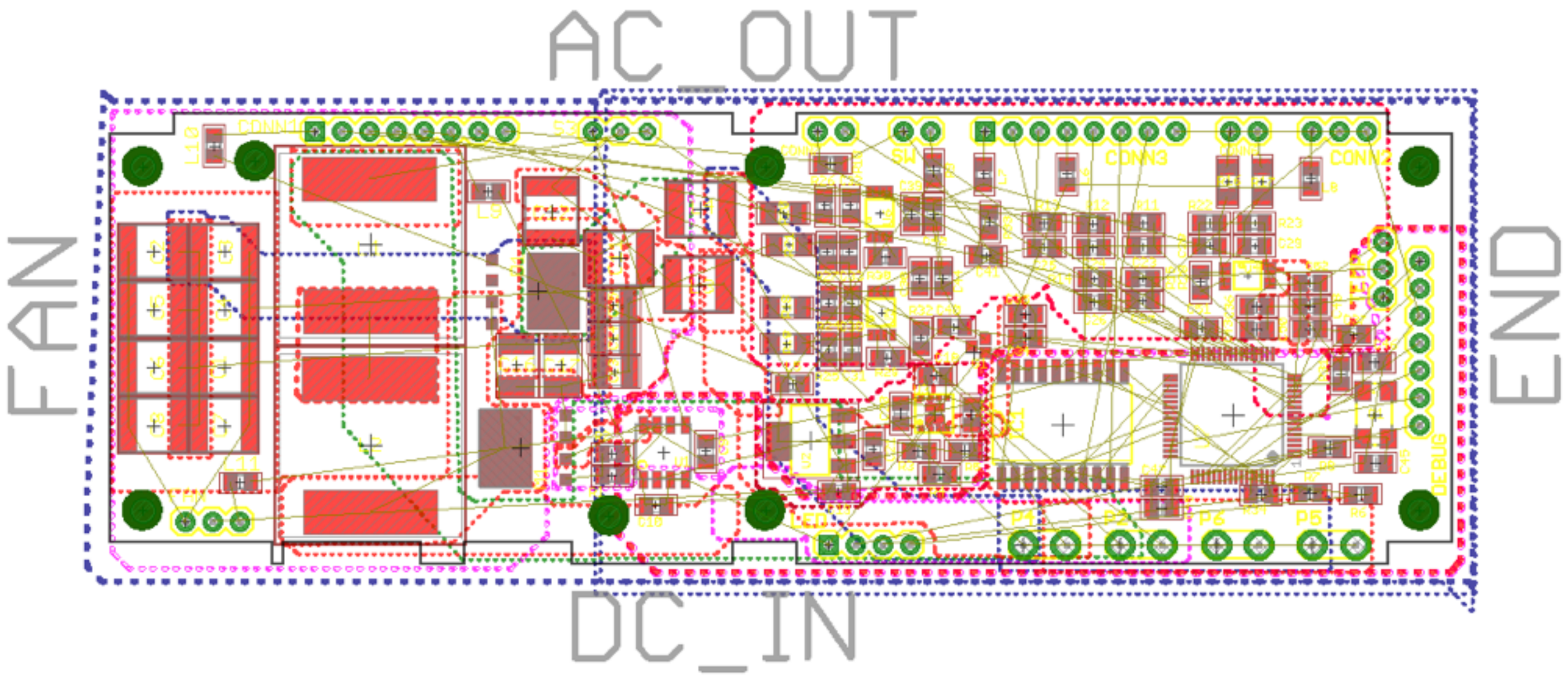
µC Schematic



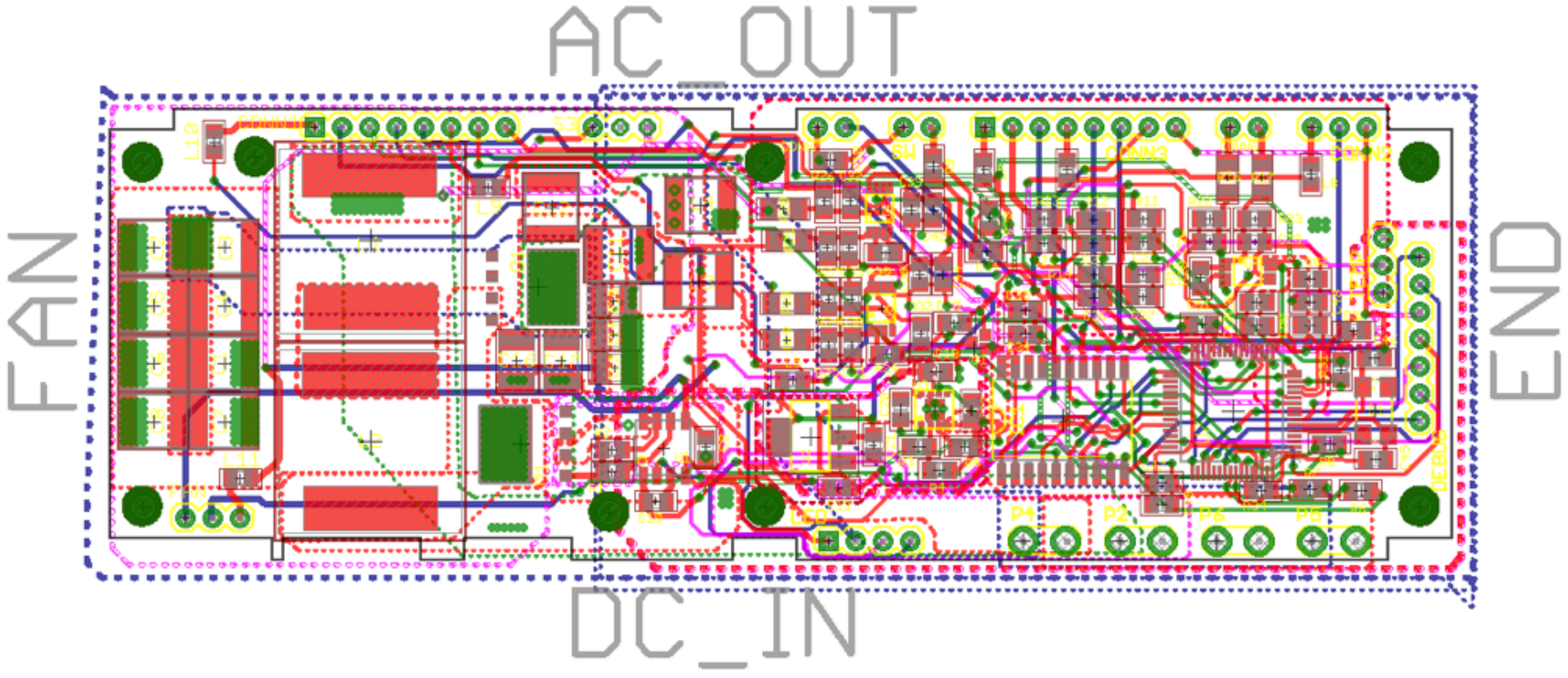
Planning

- Mechanical considerations for component placement
  - Account for connectors and required positions
  - Mounting holes
    - Heat sink placement
- Low inductance switching loop
- High current traces
- Noisy/quiet power planes

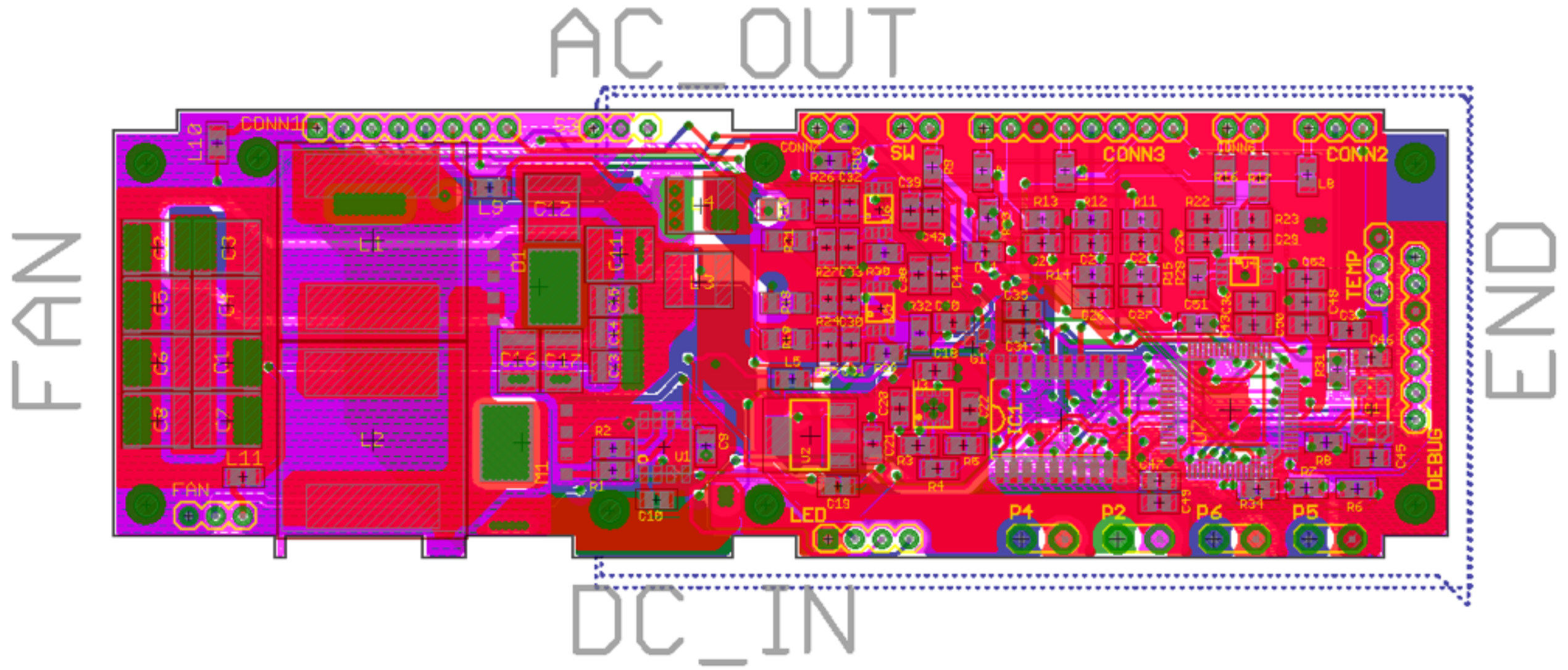
Component Placement



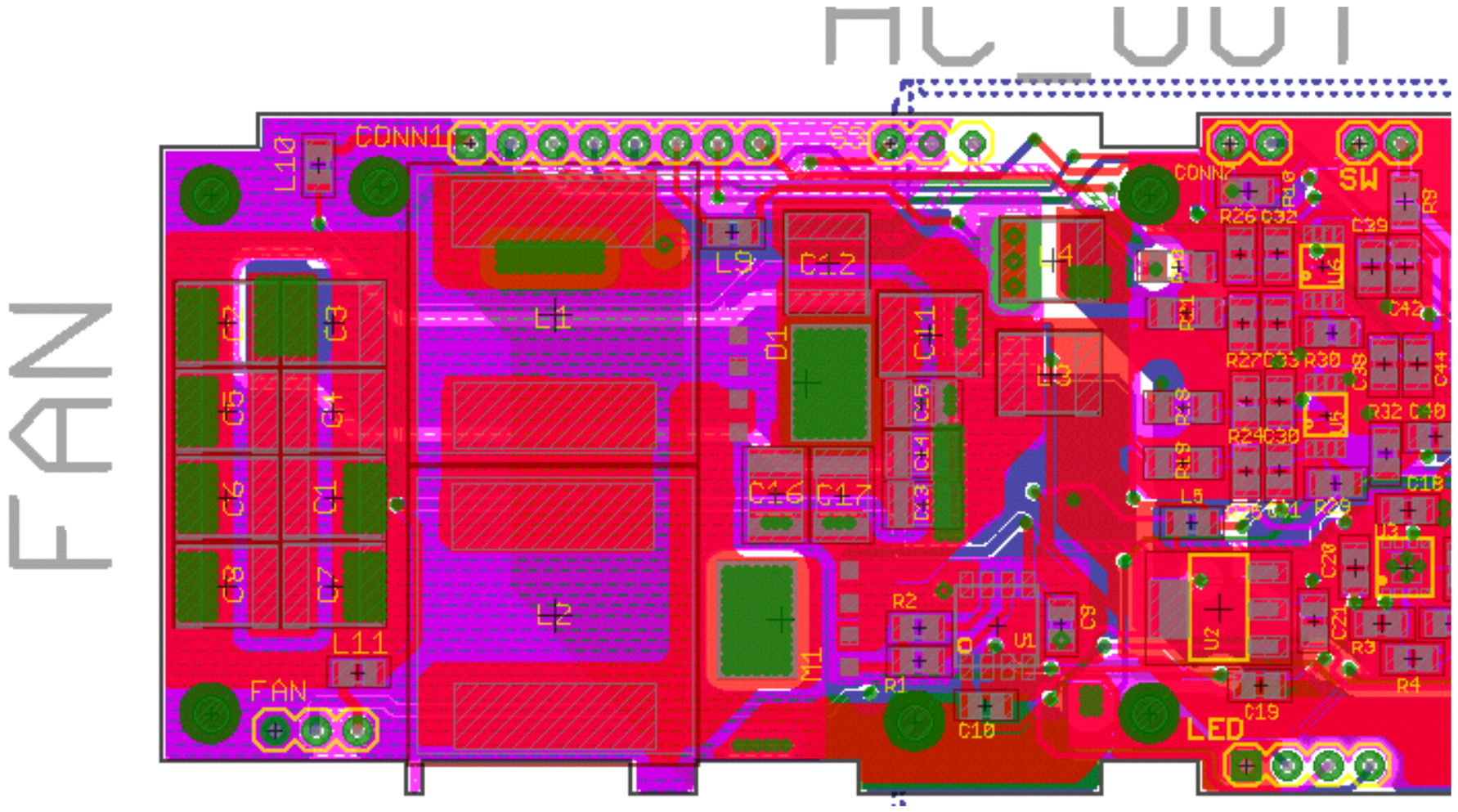
Route Signals



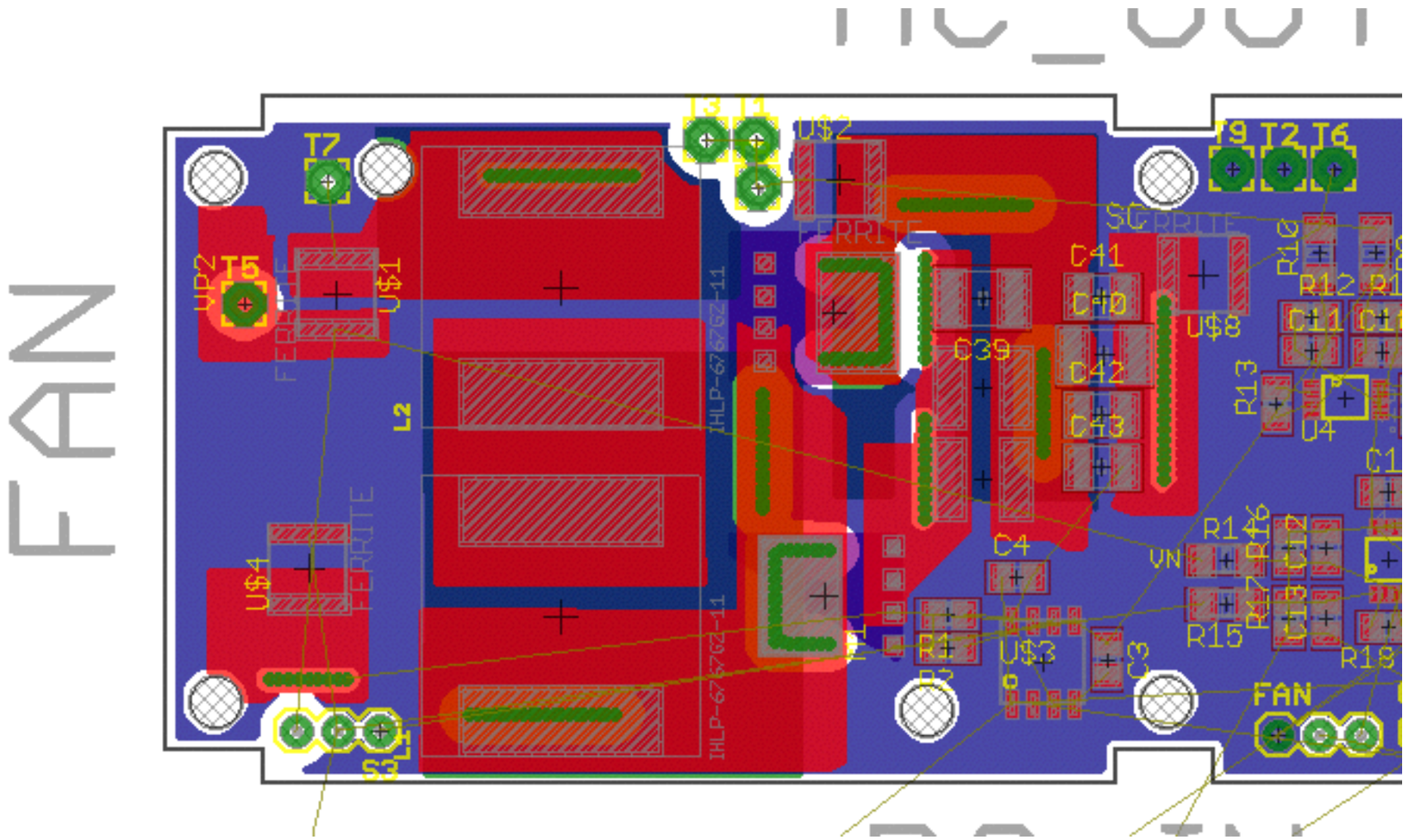
Add Polygons



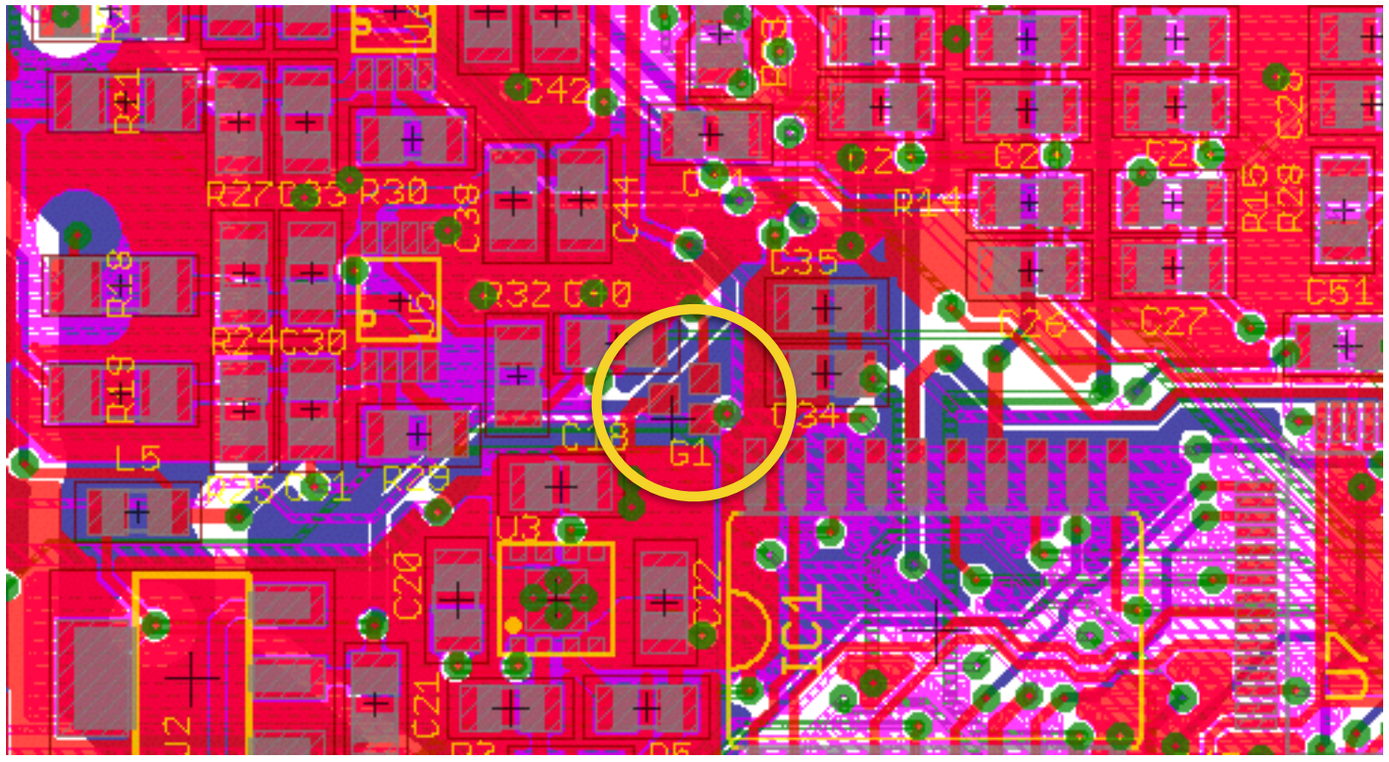
Critical Loop



Many Iterations



Combining Grounds



General Hints

- Design in stages
- Frequently review layout and check for errors
- Keep notes
- Leave testpoints
- If possible leave pads for circuit adjustments