Making Better PCBs

A start-to-finish guide for getting started designing quality boards

Meta: About this talk

- Kinda wordy! (Sorry)
- Breadth over depth
- Laid out for later reference.

The General PCB Design Process

- What does it need to do?
- Initial Part Selection
- Draw schematic
- Lay out pcb
- Redo everything
- Prep for fab

The Process: Design Decisions

- Inputs?
- Output?
- Sensor Data
- Microcontroller? Analog?
- One-off or product?
- Where will the board be used?
- What parts simplify all your work?

Your first board:

- Will probably be wrong.
  - Make it cheap
  - Make it useful
- What you’ll mess up
  - Part spacing + orientation
  - Footprints
  - Via/header size
  - Bad part choices
  - Assembly

THAT’S OK!

Board one: How to learn from it

- Build 2 copies
    - 1 to fix at all costs
    - 1 to verify corrections or failures
    - If possible, save 1 as a “control” layout.
- Make notes on spacing/sizes
- Avoid adding microcontrollers
- Make notes of other difficulties

Picking your design tool

- Cost to you
- Features + Upgrade path
- Ease of Use + Learning Curve
- Community + Documentation
- Existing parts
- Cost to others
- Every tool sucks

Picking Your Design Tools

Clearly biased suggestions:

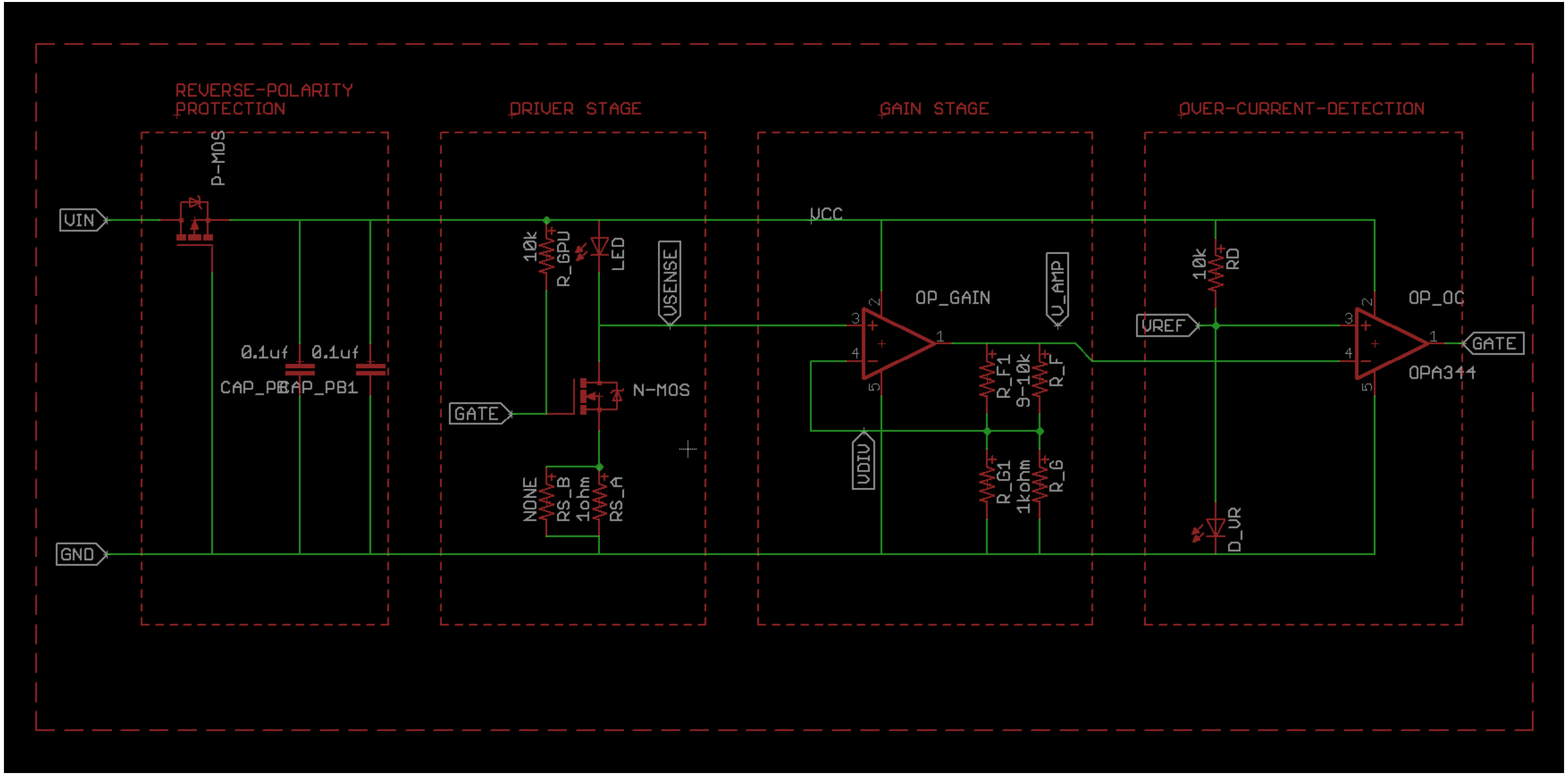
- KiCAD
  - Free, open source
  - Great community
  - No design restrictions
  - !! Quirks, bugs, version-specific issues
  - Rapidly improving

Picking Your Design Tool

- Eagle
    - Easy to get started with
    - Great community
    - Well developed community libraries
    - de-facto standard in the Arduino community
    - !! VERY stagnant development.
    - !! Lots of restrictions on free versions

If in doubt, try KiCAD first.

Schematic layout!



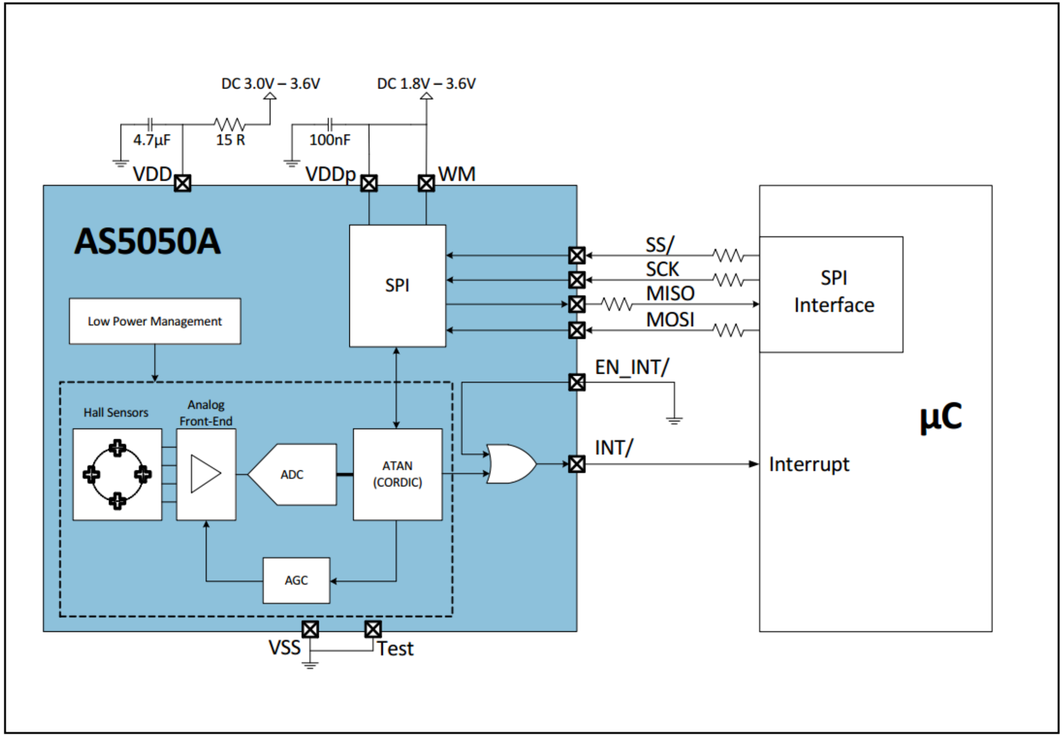
Glossary: Schematic

- Component
  - A part, such as a resistor or integrated circuit (IC)
- Net
  - A name for a particular signal
  - Can connect many wires, ICs, or board layers.
- Schematic
  - The symbolic connections between parts.
  - Connects Net, not "traces".

Schematic

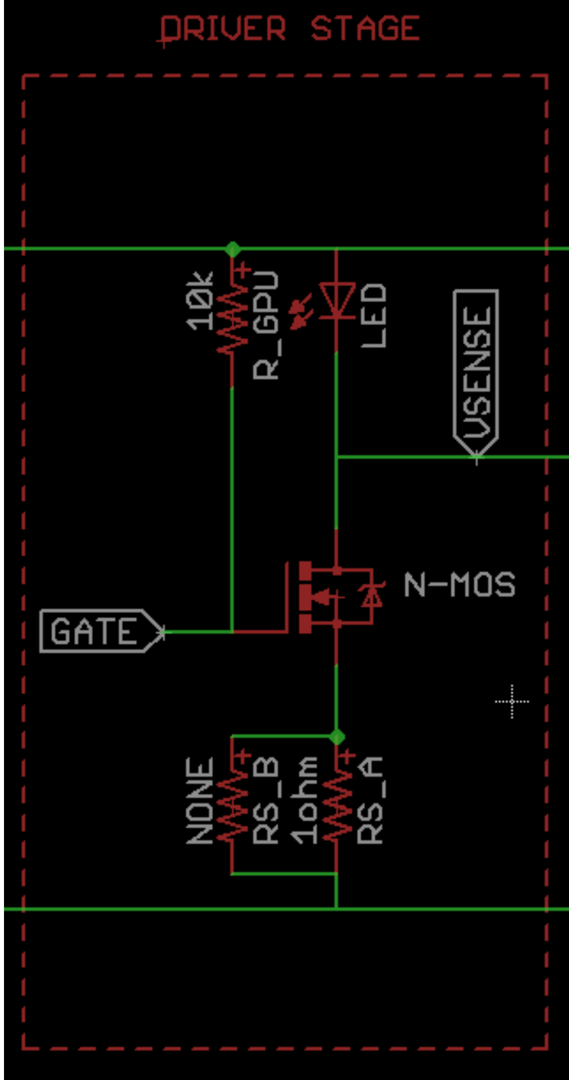
- Datasheets:
  A dummies guide
- Provide suggested components
- Details connection interfaces and options

Figure 10:
Typical Application Using SPI 4-Wire Mode and INT/ Output



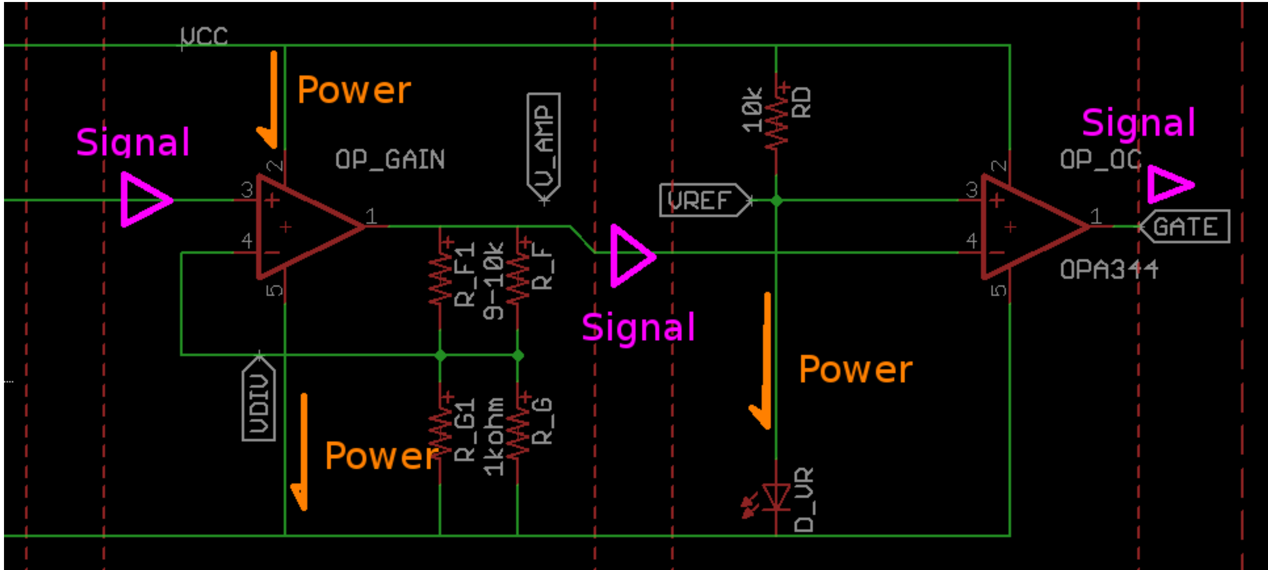
Schematic: Name nets

- Defaults are stupid: n$7
- Human-readable means human-checkable
- Can be checked during PCB Routing too
- Allows meaningful labels



Schematic: Conventions

- Power flow goes top to bottom
- Signal flow goes left to right
- Most parts follow this (not all)



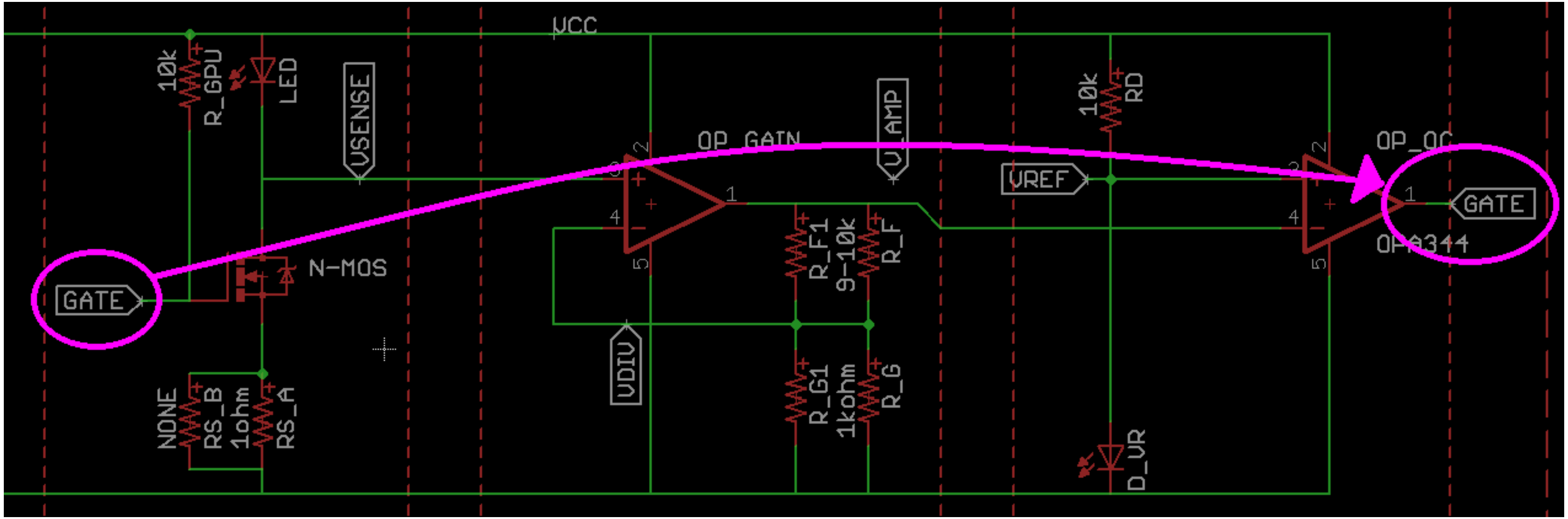
Schematic: Conventions

- Active low nets: Prefix name with “n”
  - eg: nEnabled, nDisabled, nActive,nFloating
- Avoid long lines to power/ground
  - Use labels or Power/Ground symbols

- Use descriptive names to avoid confusion
  - PWM_RED_LED
  - I2C1_SDA
  - UART1_TX_OUTPUT

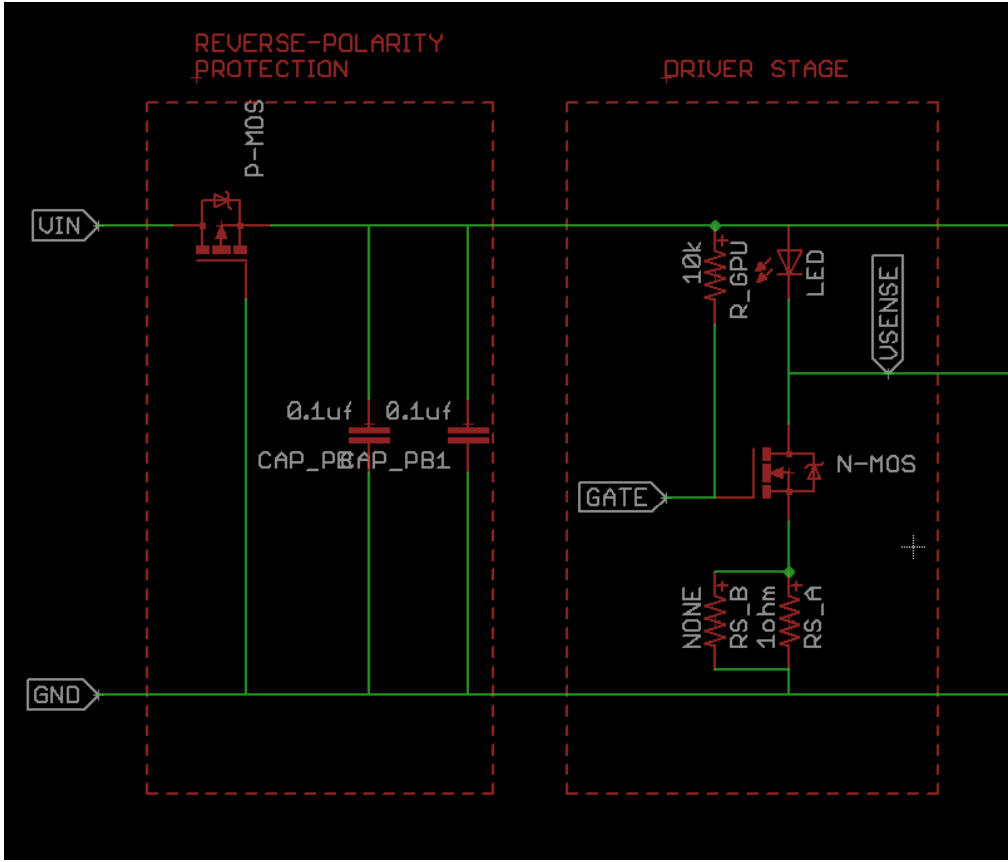
Schematic: Don’t worry about lines

- Use net names and labels
- Reduced clutter = better checking



Schematic:Create Functional Blocks

- Group related parts
- Label nets between blocks



Schematic: Tips for Beginners

- Add Extra status LEDs
  - Add one per power net
  - One for microcontroller to blink/toggle
- Add test points
  - Give you a place to test voltages
  - Makes cutting/rerouting traces easier
  - Add in schematic so you don’t forget
- Make UART signals easy to fix
  - Add jumpers or test points to swap pins

Libraries

- Broken into 3 parts

Symbol

- Schematic-side
- Connects internal and external nets

Footprint

- Layout Side
- Connects pads to traces

Device / Component

- Connect Symbol to Footprint
- (or, connects nets to pads)
- Often has multiple footprint options for identical symbols

Libraries

Community libraries:

- Trust, but verify
- All libraries have an error
- Won’t have every part
- Can get you started quickly

Libraries

Make your own!

- You’ll need to
- You’ll get the hang of it
- Can save you work
- Can (usually) be trusted

Design Decisions: Providing power

- Off board power
  - Such as USB, 5V wall wart, or Arduino
  - Cheap + easy, but tethers your project to outlet
  - Ideal for many projects
- Batteries
  - Lipo? NiCAD? NiMH? Alkaline?
  - Large voltage ranges
  - Charging?
  - Forces low-power designs

Design Decisions: Regulating power

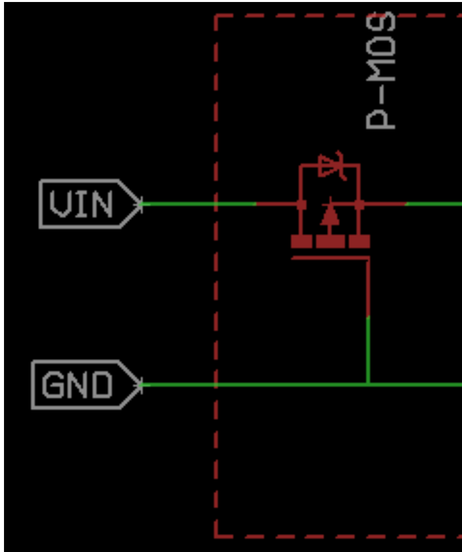
- Linear Regulator
  - Pro: Cheap and simple
  - Con: Watch the voltage drops
  - Con: Can be hot and inefficient
- Switching Regulator
  - Con: More parts, and much more complex
  - Con: Often needs more PCB area
  - Pro: Can provide large voltage drops efficiently
  - Pro: Can increase voltage source provides

Design Decisions: Regulating power

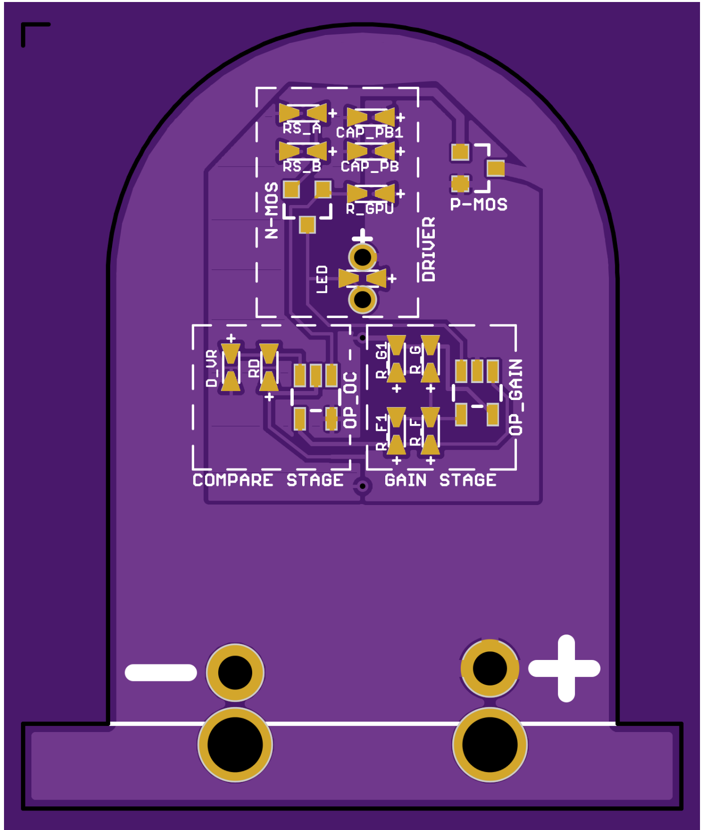
- Unregulated battery power
  - HERE BE DRAGONS. May cause unexpected issues. Sometimes serious.
  - Voltage range will drive component selection
  - Can be good for simple blinky or analog circuits

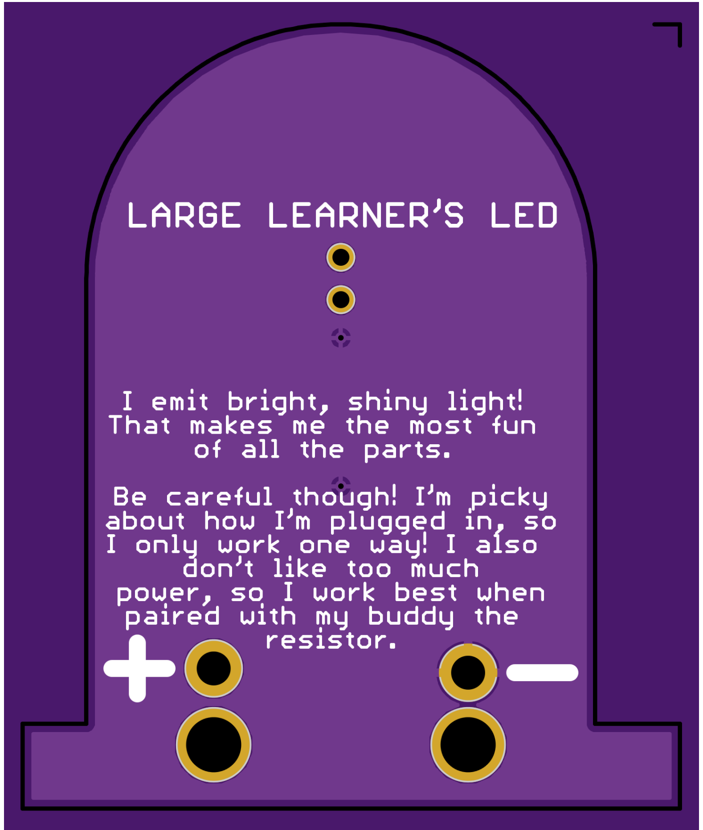
Design Decisions: Regulating power

- Input Power Protection
  - Good idea when moving past prototype stage
  - Can be very simple:
    - Reverse Polarity can be 1 part
    - Overcurrent or overvoltage can be 1 IC, or a few parts
    - Some ICs do everything for you



PCB Routing: Physical Connections





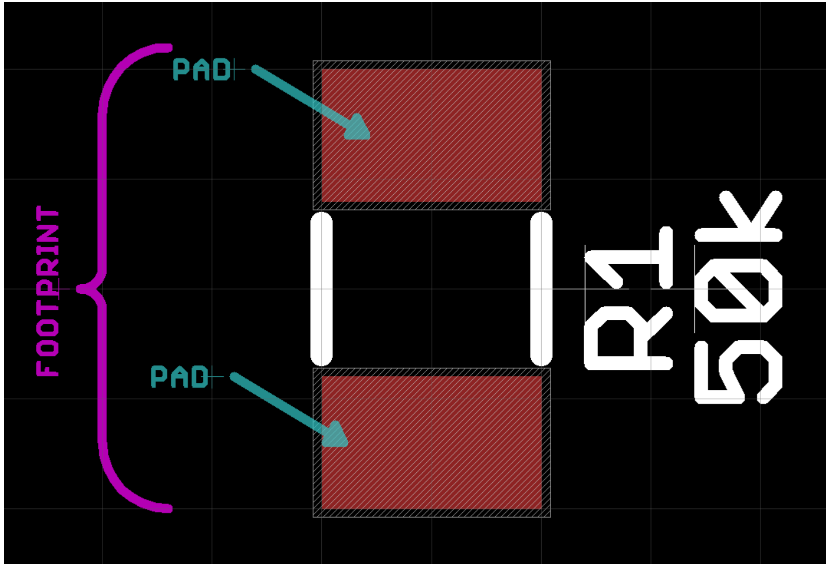
PCB Routing: Glossary

AKA Board Layout

- DRC: Design Rule Check
  - Uses specifications to verify manufacturing requirements
- DFM: Design For Manufacturing
  - Guidelines and suggestions for preventing errors
  - Usually not enforced by design tool
  - Covers wide selection of design and layout areas
  - Critical as volumes and complexity increases

PCB Routing: Glossary

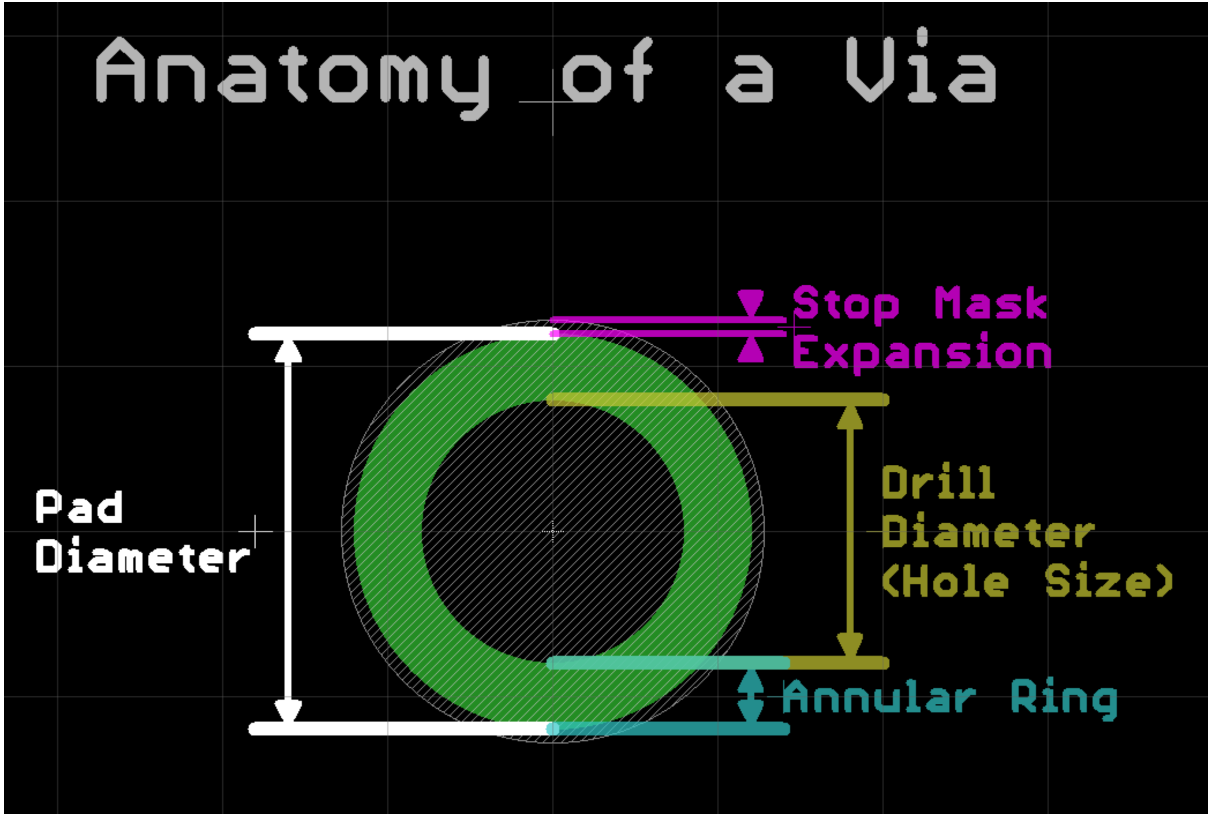
- Stop Mask: Indicates a section of the board that should be exposed
- Pad: A small exposed section of copper.
  - Typically refers to where you'll be attaching a part
- Footprint: A set of pads that match a component's pin arrangement



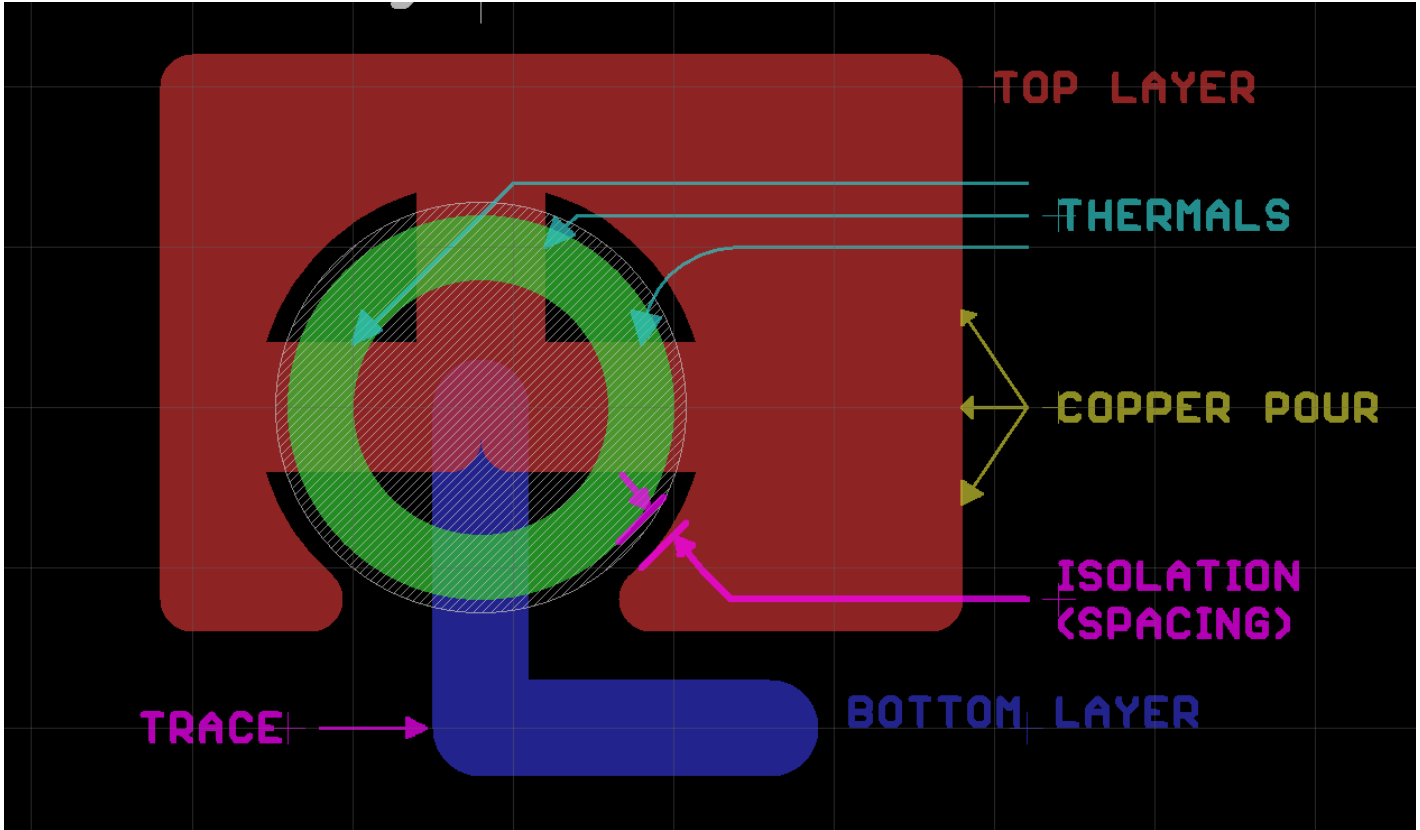
PCB Routing: Glossary

- Package: Physical size and shape of a component
  - Most parts have multiple packages
- Pinout
  - How the nets from an IC connect to the pads
  - Parts with the same footprint can have different pinouts.

PCB Routing: Glossary



PCB Routing: Glossary



PCB Layout: Design Rules

- Critical specs:
  - Minimum Drill size
  - Annular Ring
  - Trace Spacing
  - Trace Width
- Less Critical specs
  - Board-edge clearance
  - Mask expansion/retraction
  - Minimum mask web

PCB Layout:

Datasheets: A dummies guide

- If routing matters, ICs often have examples and notes.
  - High Current
  - RF
  - Low-power
  - Analog

10.1 Layout Guidelines

The VM and VCC terminals should be bypassed to GND using low-ESR ceramic bypass capacitors with a recommended value of 0.1 µF rated for VM and VCC. These capacitors should be placed as close to the VM and VCC pins as possible with a thick trace or ground plane connection to the device GND pin.

10.2 Layout Example

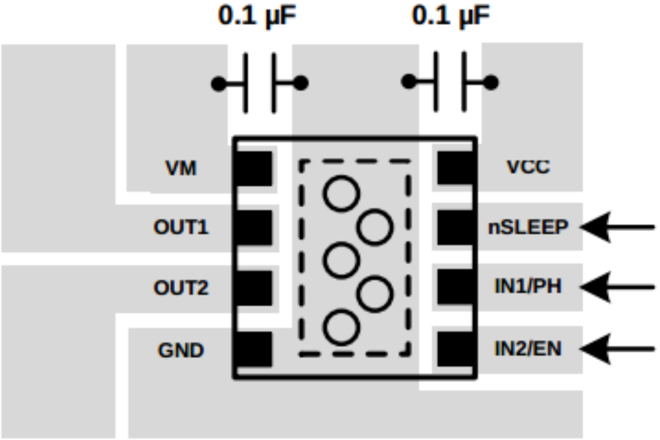


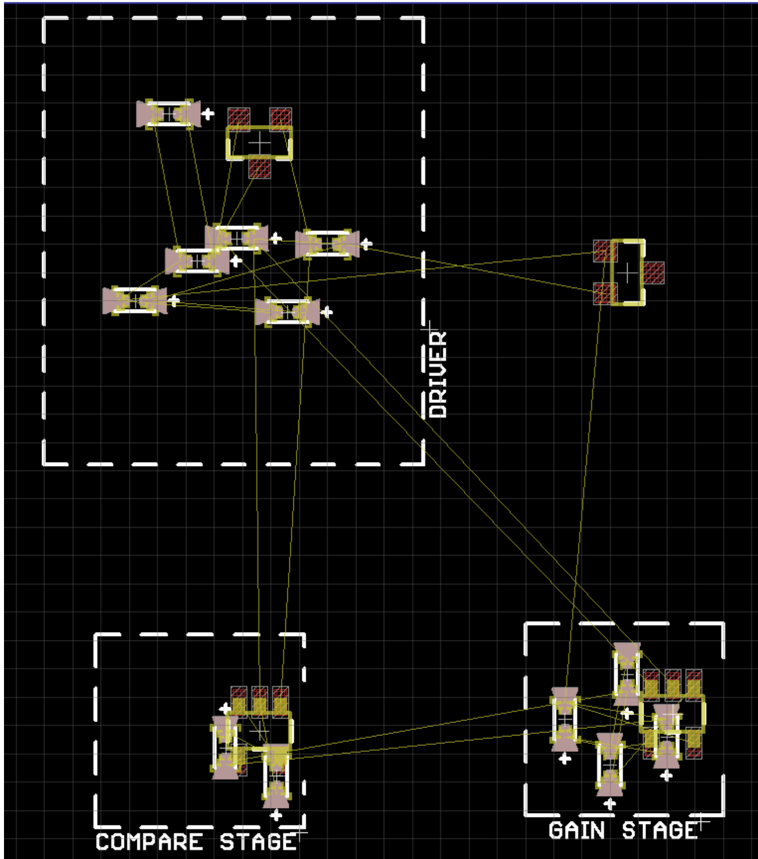
Figure 11. Simplified Layout Example

PCB Layout: The general process

- Separate parts into blocks you made earlier
  - Organize parts
  - Route power and ground
  - Route Signals
- Combine blocks on desired board shape
  - Route power and ground
  - Route Signals
- Make corrections

PCB Layout: Forming Blocks

- Sort components out
  - Just like the schematic!



Design Decisions!

Components on one or both sides?

- Both: Can Ease Routing
- Both: MUCH harder to assemble

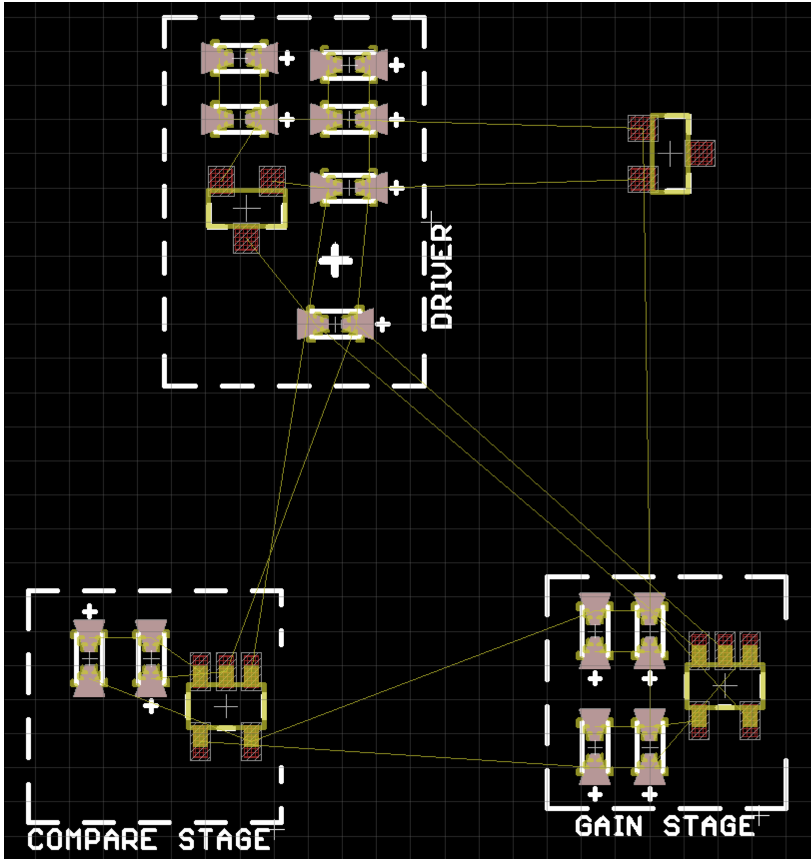
2 or 4 layer board?

- 2 layer is less expensive
- 4 layer makes routing much easier
- 4 layer may not be an option

PCB Layout: Routing Blocks

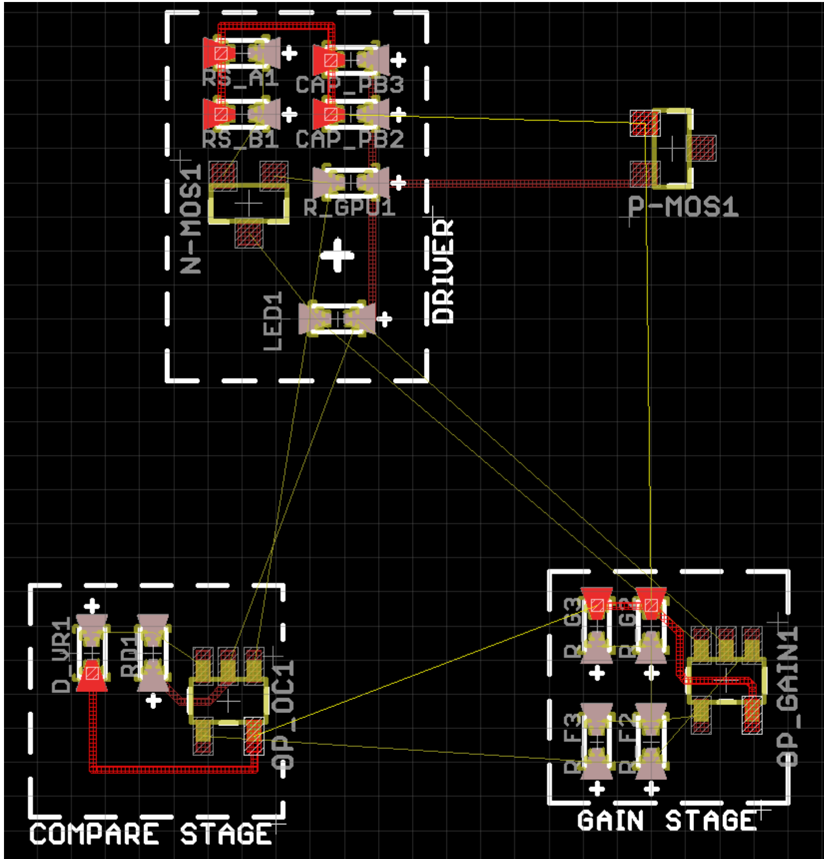
Untangle Airwires

- Minimize crossover
- Minimize routing length



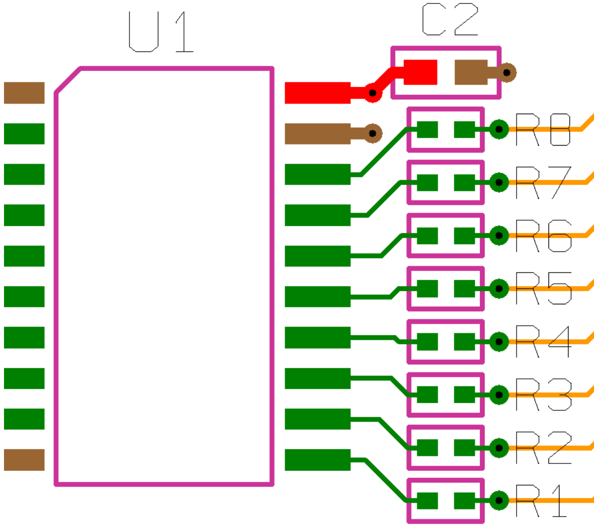
PCB Layout: Routing Blocks

- Power and ground first
- Bypass caps second
- Important Signals
  - Analog
  - communication
  - high-power
- Everything else



PCB Layout: Bypass caps

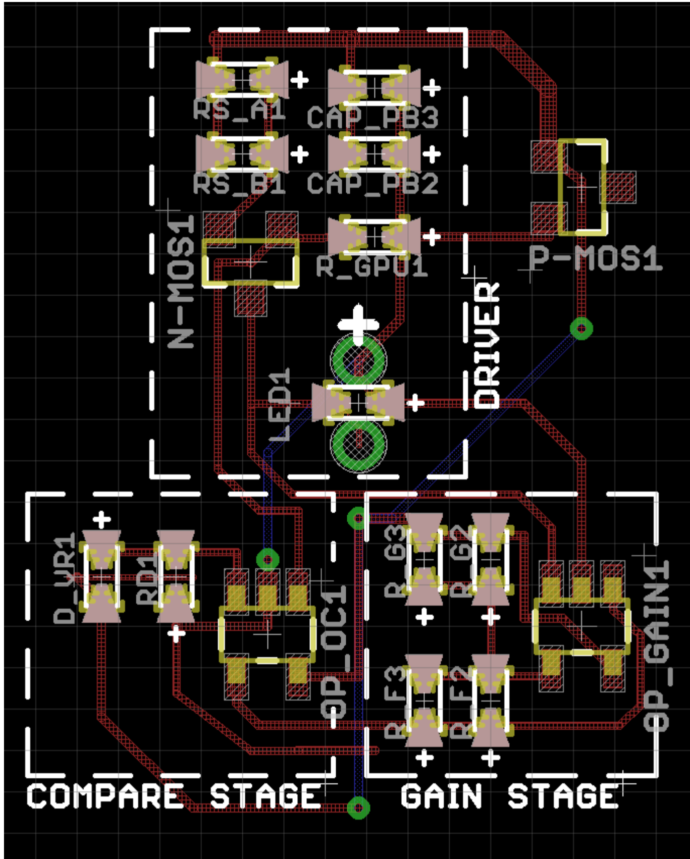
- Magic sprinkles of electronics
- Every IC should have one
- Requires intentional routing
    - Minimize “loop area” between IC Power, IC Ground, and cap pads
    - Avoid vias (generally)
    - Should be physically close to IC



http://blog.optimumdesign.com/how-to-place-a-pcb-bypass-capacitor-6-tips

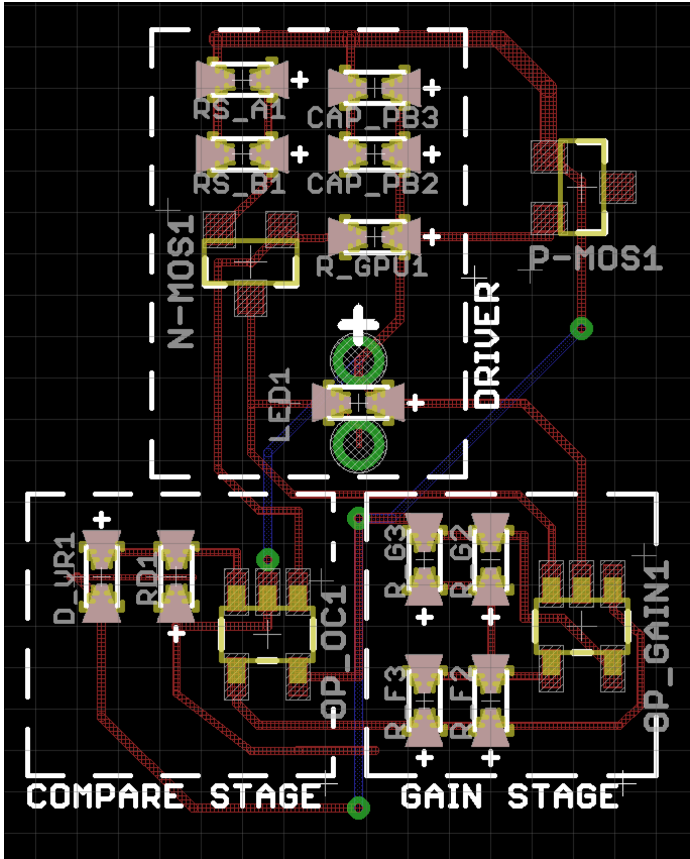
PCB Layout: Connecting Blocks

- Arrange blocks
  - Minimize airwire distance
- Plan for mechanical
  - Odd PCB shape?
  - Offboard connections?
  - Mounting?
- Connect!
  - Power, then signals



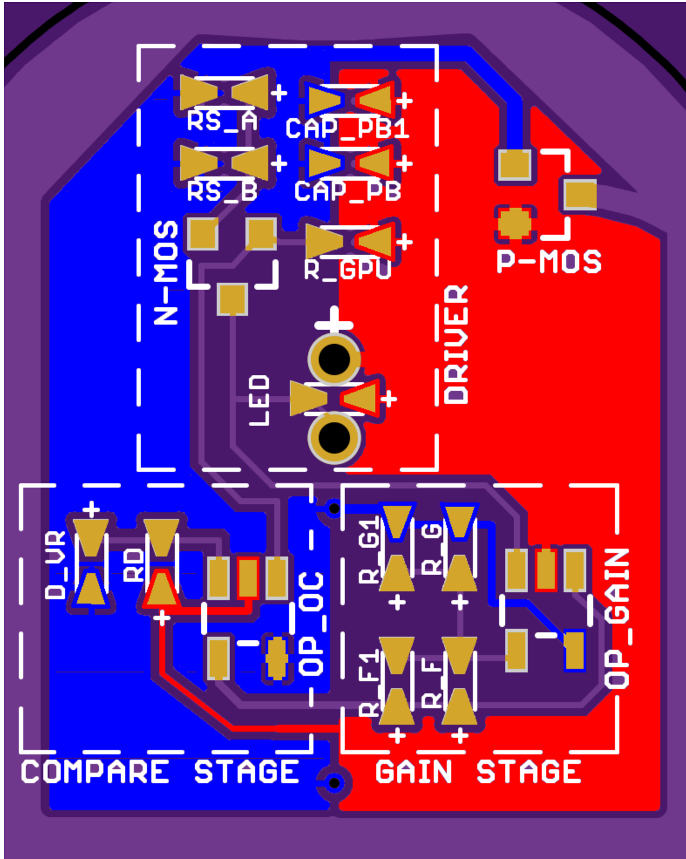
PCB Layout: Routing Strategies

- Short + Long
  - Put Parts + short traces on top
  - Long connections on bottom
  - Ground pour on bottom
- Up/Down + Left/Right
  - Put up/down traces on top
  - left/right traces on bottom
- Varies by design!

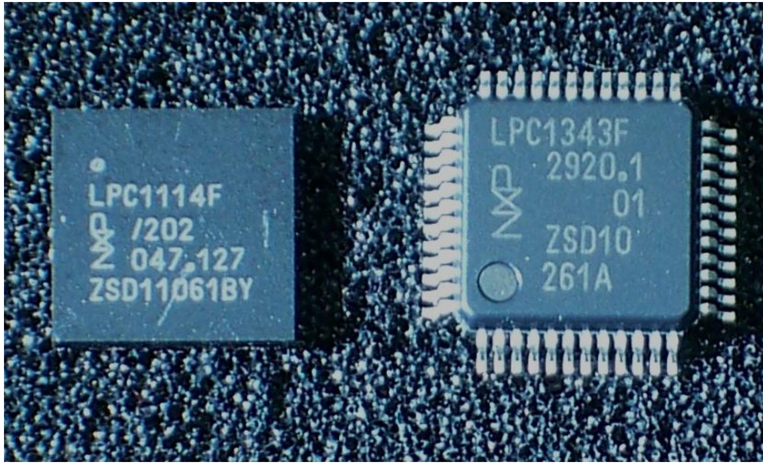


PCB Layout: Design Decisions

- Ground/Power Planes?
  - Route first! (for now)
- 4 layer Routing Strategy:
  - Top: Parts + short connections
  - Ground plane
  - Positive power plane
  - Bottom: Longer connections



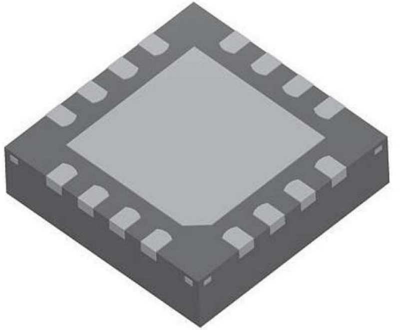
PCB Layout: Design Decisions



Surface Mount Parts

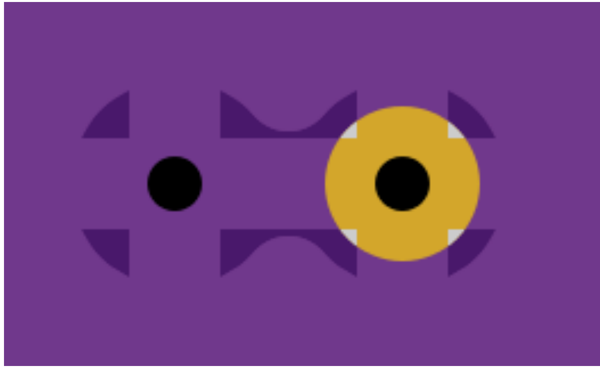
- Lots of advantages
- Need to use them eventually

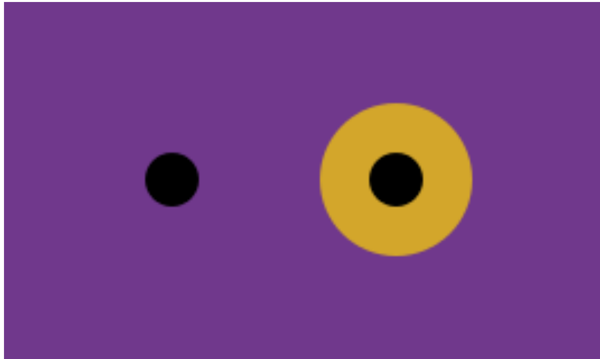
- Beginner Friendly
  - Passives: 1206, 0805
  - IC: Leadless packages, MSOP-*, SOT-*
- Less friendly
  - 0402: Really tiny and hard to place
  - BGA, pitches 0.5mm and lower



PCB Layout: Design Decisions

- Via Tenting?
- Thermals?
- Assembly consideration
  - Space for rework
  - Leave room for silk
- Heatsinks?
  - Voltage Regulators
  - Motor drivers





PCB Layout: Routing Busses

- Bus: A group of signals performing the same function
  - Common for chip-to-chip communication
- Should be routed at the same time
  - Try to keep together
  - Length differences may matter
- Consult routing guides for the protocol
  - More/less picky, voltages, pull ups, oh my.

Routing: Notable Signal types

Signal type	General Analog	Precision Analog (<10mv precision)	High Speed Signals (>1Mhz)	High current (>500mA)	High Voltage (AC, or >48VDC)
trace width (beyond fab spec)		Y	Y	Y	Y
trace spacing (beyond fab spec)		Y			Y
Trace length	Y	Y	Y	Y	
via size		Y	Y	Y	Y
multiple vias		Y	Y	Y	
trace placement / adjacent signals	Y	Y	Y		
parasitic capacitance + inductance		Y	Y		
heat		Y		Y	Y
Solid ground connections (ground loop path)	Y	Y	Y	Y	Y

General analog:

- Typical hobby-level stuff, with voltage swings of 100+mV causing no issues.
- Greater than/less than voltage comparisions usually fall in here

Precision Analog:

- Usually for fine logging or measurements.

High Speed:

- Common for clock signals, Wireless radios, some serial busses, video signals

DFM: Design for Manufacturing

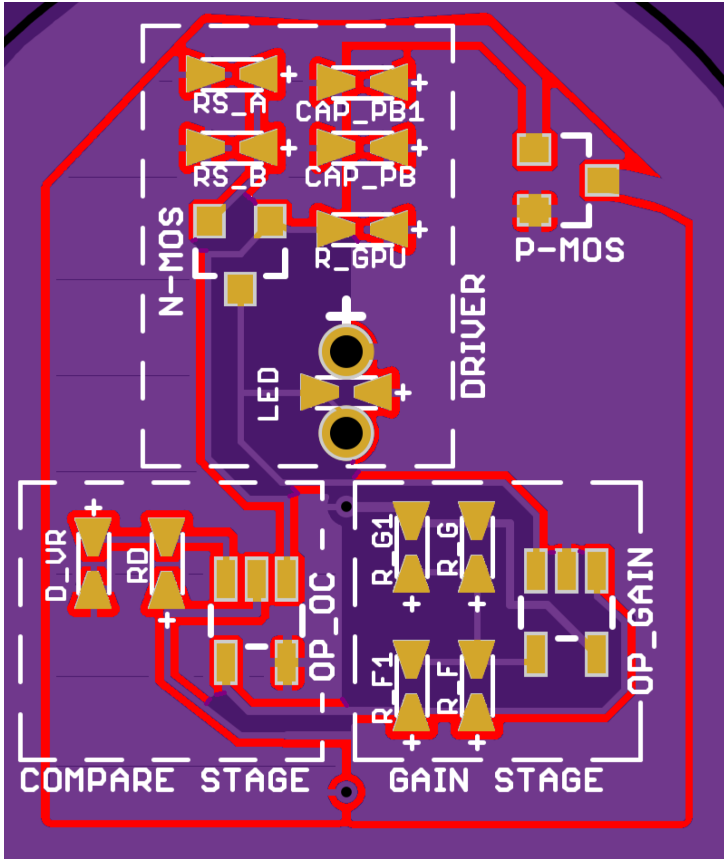
Or, How to minimize fabrication problems

- Fab specs are minimums
  - Opt for larger traces, and larger spacing
  - Increase ground plane isolation
- Avoid placing parts needlessly close
- Doubly important for open designs
  - Can’t always plan for specific fab specs/QC

DFM: Ground Plane Isolation

- Set above fab min spec
- Touches lots of traces
- Hard to find and troubleshoot

- Example Short:

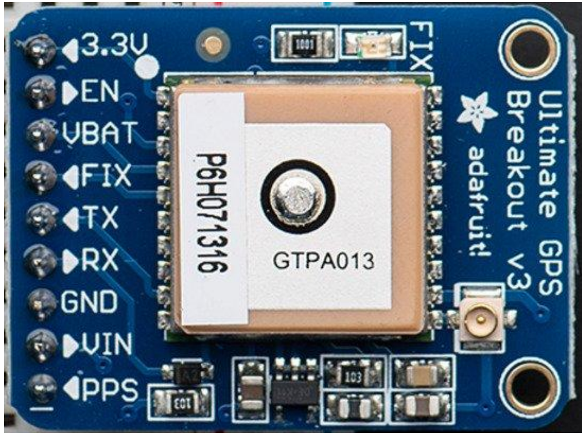


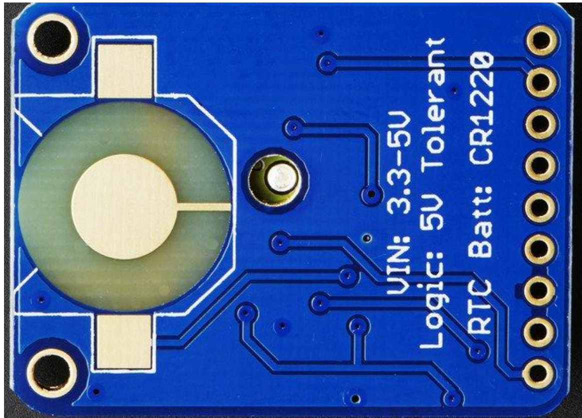
Fabrication: Getting prepped

- Run DRC
- Double check pinouts
- Verify availability of components
- Put date, version, and name on board
- Mechanical Concerns
    - Mounting holes?
    - Enclosures?
    - Part sizes?

Fabrication: Getting prepped

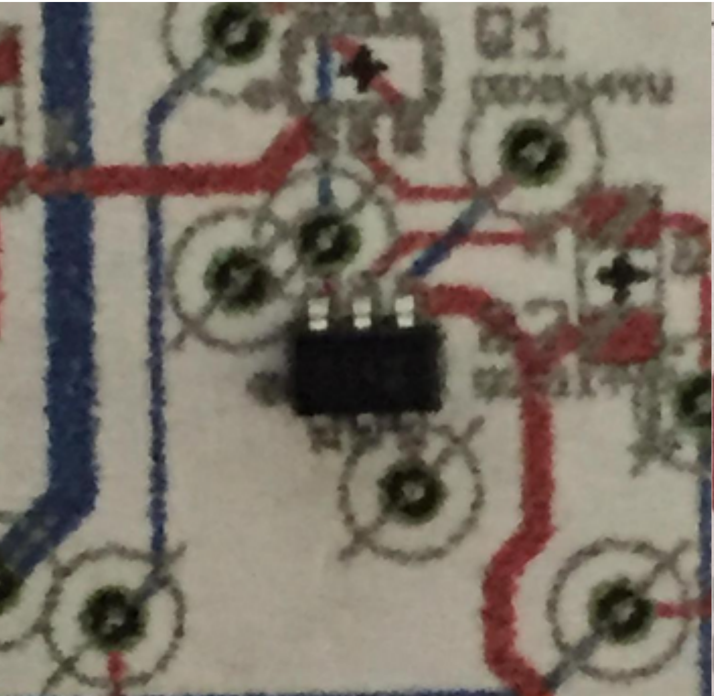
- Clean up silkscreen
  - Component names
  - Label off-board connections
  - List power input voltage ranges
  - Mark polarity of components
  - Make silk readable
    size (>0.035”)

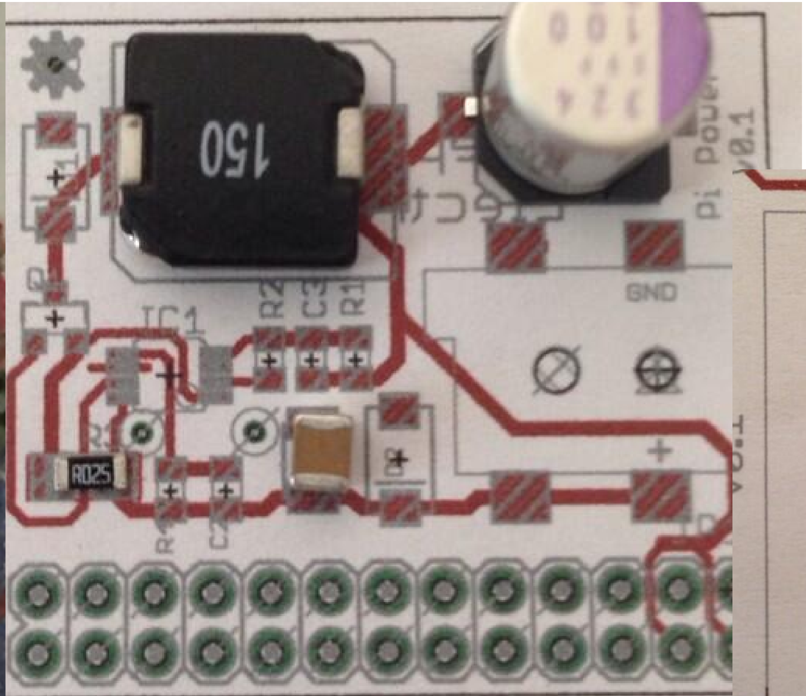


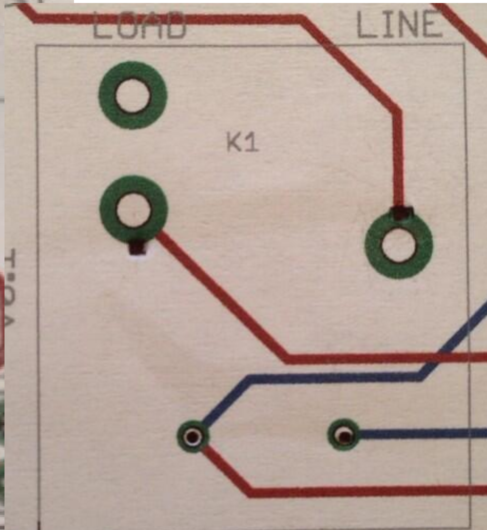


Fabrication: Getting prepped

- 1:1 Scale Printout







Fabrication: Ordering Boards

- Run the CAM processor
    - (usually)
    - Generates gerbers and drill files
    - Verify expected drill format for your fab
        - Drill formats are stupid.
    - Verify board shape / outline gerber
        - Dimension layer on Eagle
        - Edge_Cuts on KiCAD
        - Mechanical 1 on most other tools

Fabrication: PCB Layers / Gerbers

- Simple files corresponding to a single part of the fabrication process
    - Copper Placement
    - Hole placement
    - Solder resist placement
    - Silkscreen printing
-

Fabrication: PCB Layers / Gerbers

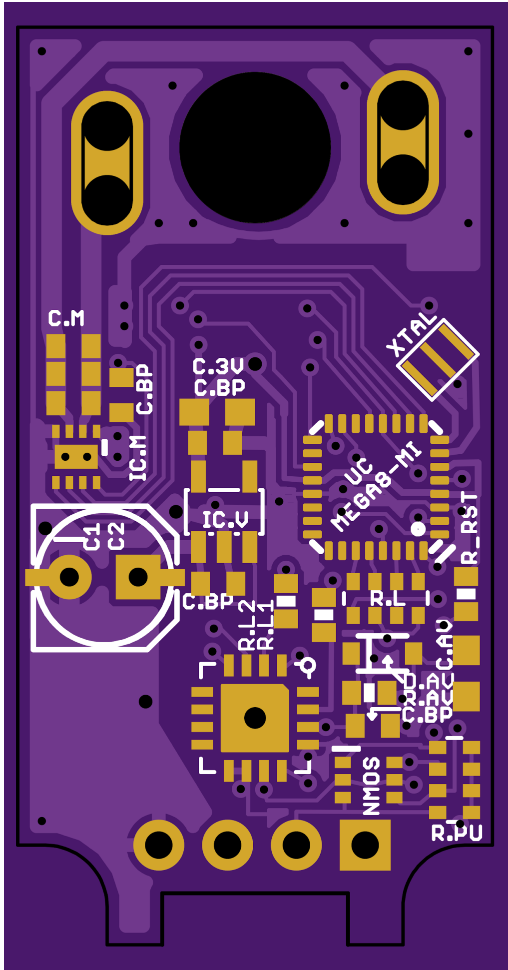
- Complexities come from format ambiguity
  - Plated drills?
  - How is the edge defined?
  - How is the drill format handled?
  - Positive or negative internal layers?
- If in doubt, ask your fab!
  - Folks get paid to help you. Take advantage!

Assembly: What you'll need

- A decent iron
- Solder
- Desoldering braid
- For SMD work:
    - Fine point stainless steel tweezers (Walgreens)
    - Solder paste (Chipquik is best. cheap Ebay/Amazon paste works, but not suggested)
    - Stencil?

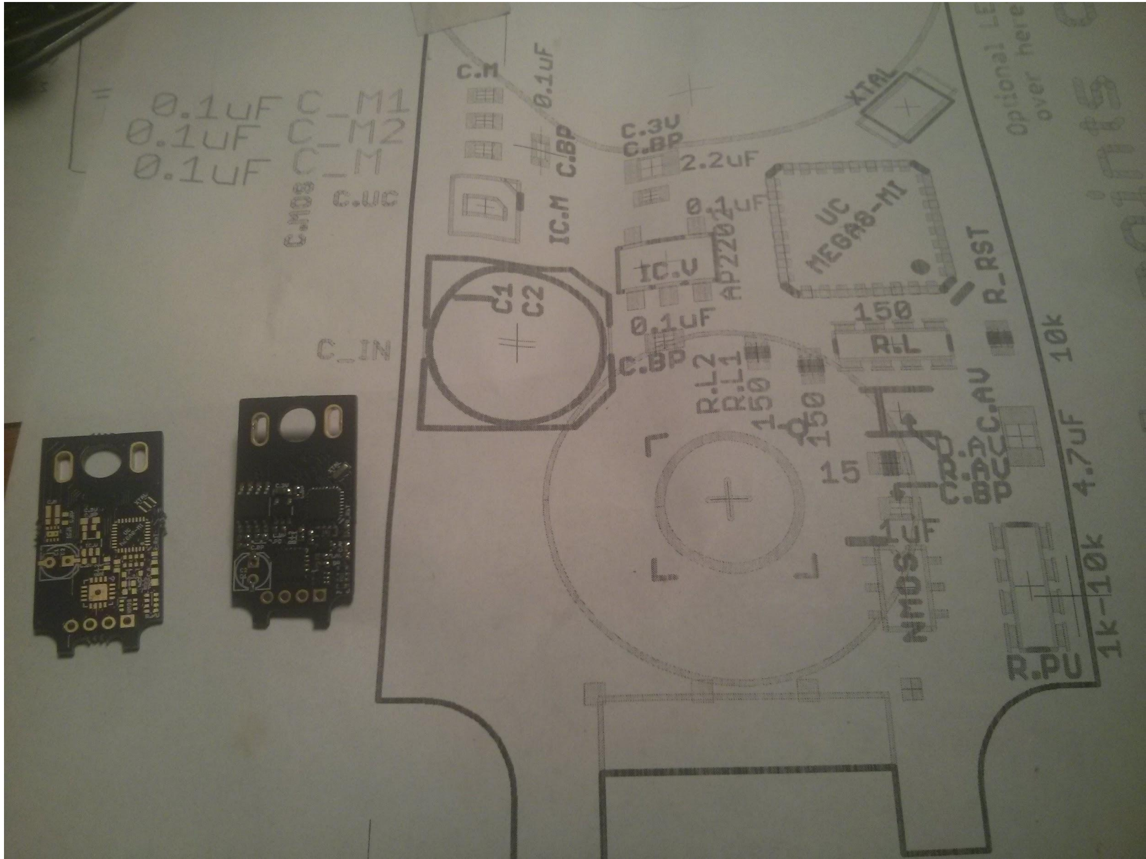
Assembly: SMD Placement

- Expect to bump them a bit
  - Usually pretty forgiving
- Start with hard-to-reach parts
  - Stuff in center, tight fit, etc
- Go to “easy” parts
- End with large or finicky footprints
  - BGA, TQFP, etc



Assembly: Planning BOM

- Giant printout
  - Include values, names, notes
- Parts list



Assembly: Reflow Procedure

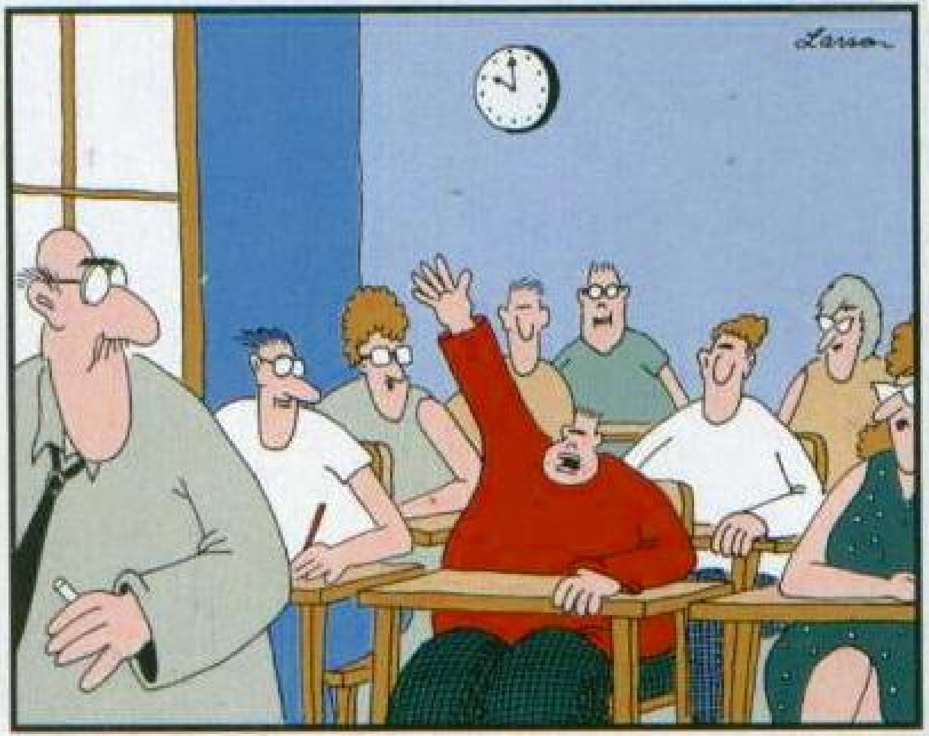
- Minimum required items:
  - Old pan
  - IR Thermometer
- My home procedure
  - Medium til around 250F-300F (flux melts into matte pools)
  - High until solder flows (~250F)
  - Nudge parts if needed (careful!)
  - Remove pan from burner

Assembly: Through-Hole

- Lots of good video tutorials
- Mostly practice
- Lots of flux
- Pro tips:
  - Poster Tack is your friend.
    - Holds board firmly to table
    - Blob around loose parts
  - Sparkfun Locking headers = <3

    https://www.sparkfun.com/tutorials/114

Questions!



"Mr. Osborne, may I be excused?
My brain is full."