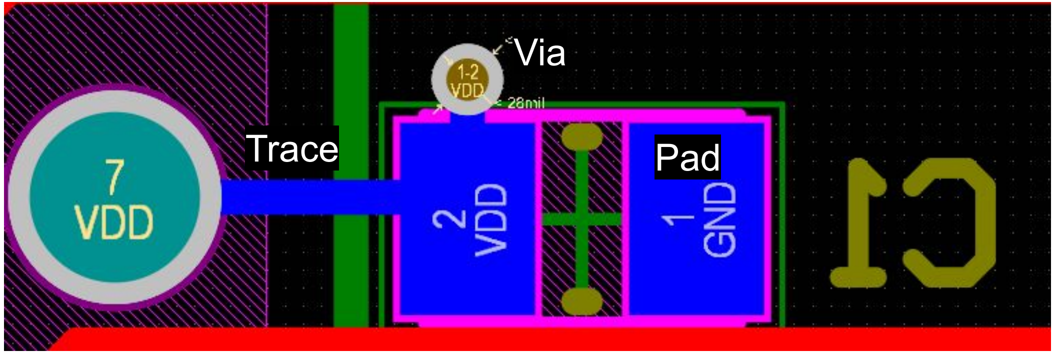
PCB Layout

Terminology Review

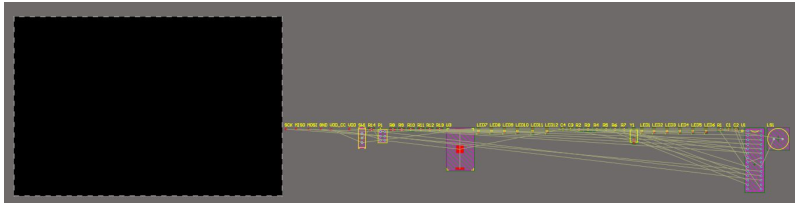
Trace/Track - Copper path that connects pins

Via - plated hole between layers that takes trace to opposite side

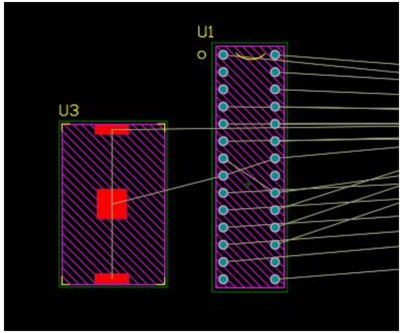
Pad - Piece of copper for soldering components



Component placement

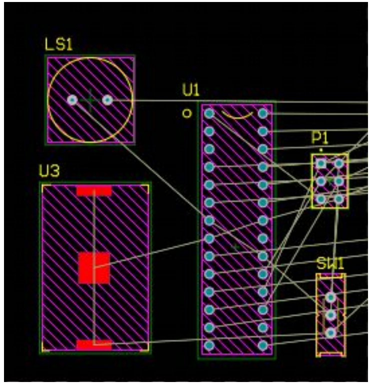


Place major components first - MCU, ICs



Place mounting holes in corners, evenly spaced for logical mechanical design

Place headers in accessible locations - board edges



Decoupling caps close to pins

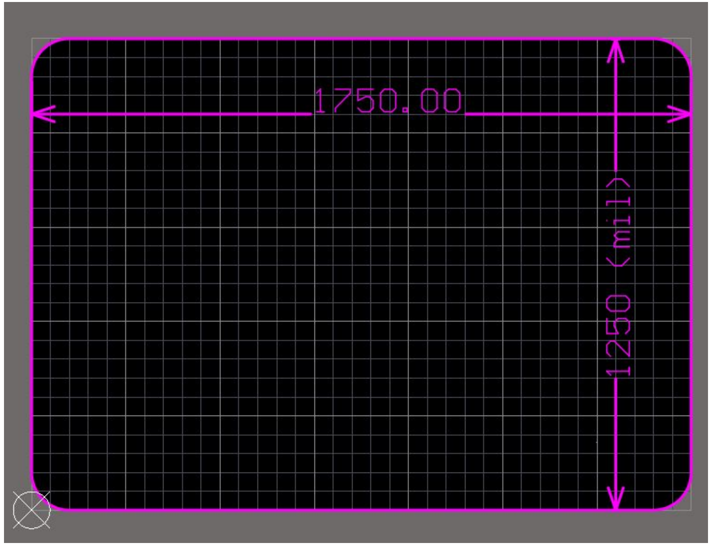
Passives last - resistors, caps

Dimensions

Logical - 50mmx60mm, not 48.6mmx59.87mm

Rounded edges around mounting holes are nice

Define board outline on mechanical layer



Layers

Top/Bottom - component placement and routing layers

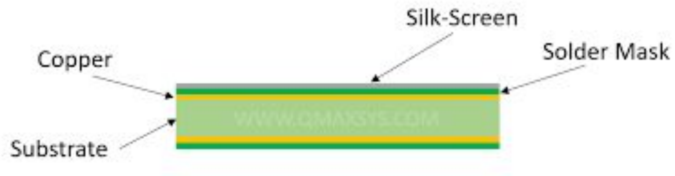
- Can have internal signal layers as well

Solder mask - protective layer coating PCB

Mechanical Layers - board outline, 3D bodies, courtyards

Silkscreen - text and images

Solder paste - Where paste will be applied to easily connect components



Routing

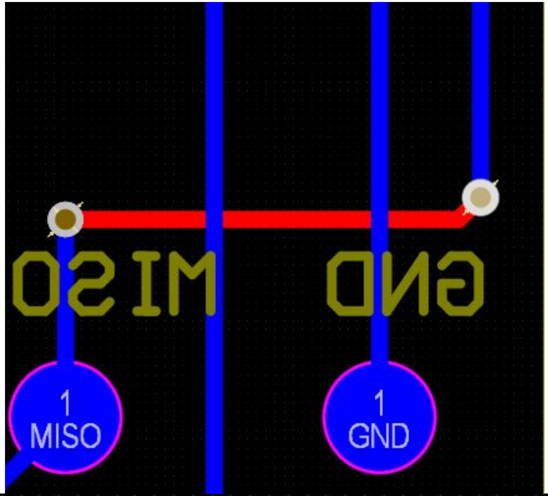
Have to connect all the pins in net to one another

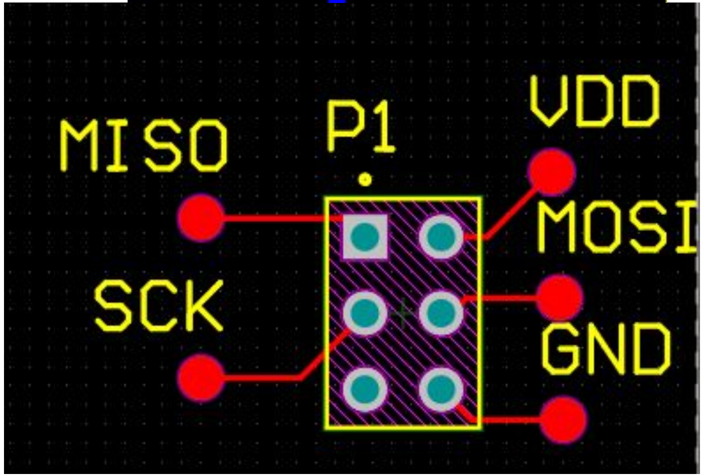
Software will show you what must be routed

Can route on top, bottom, and internal layers

To switch layers in trace, use via

Try to route top vertical, bottom horizontal or vice versa for complicated designs





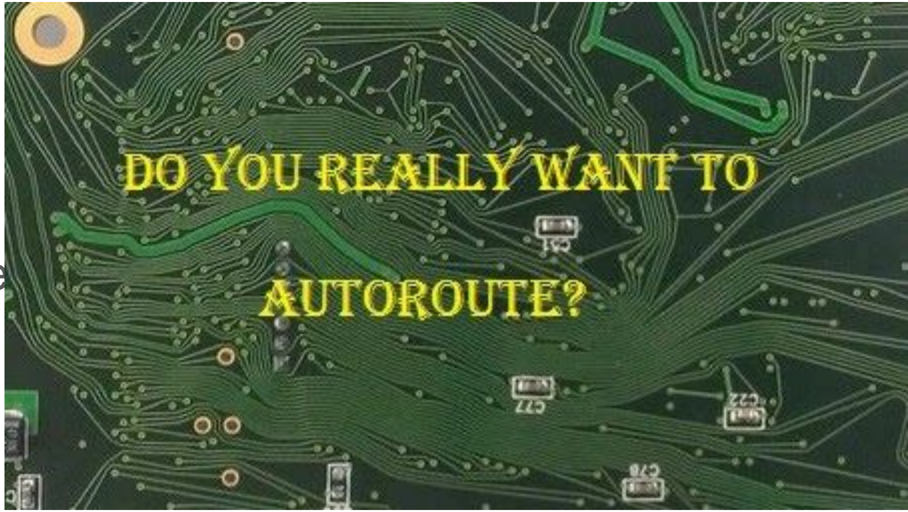
Autorouting

Once components are placed, software can route for you

If you define rules well, can be not bad

For small designs, best to just do it yourse

Especially as you are learning!



Ground Planes

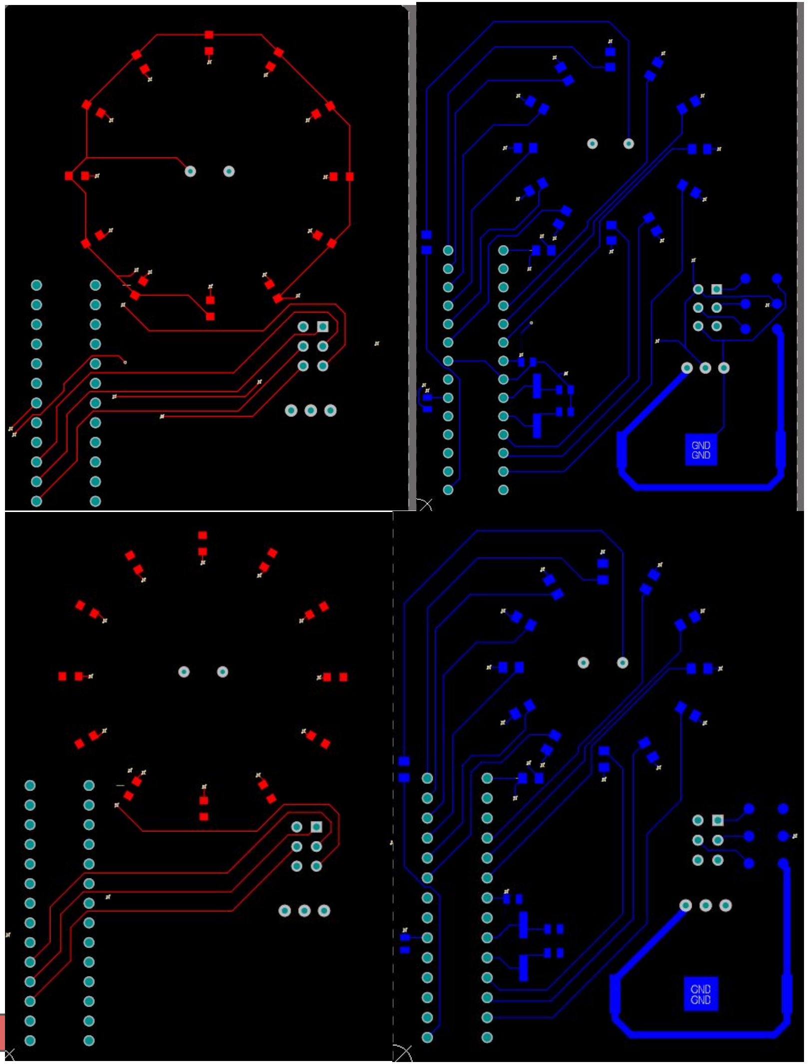
Almost every component will be connected to ground

Ground should be continuous

Can lay ground over entire board, will smartly connect only to what it should

Provides short ground return loops

Can also create power plane on opposite side, reduce routing and increase thermal relief



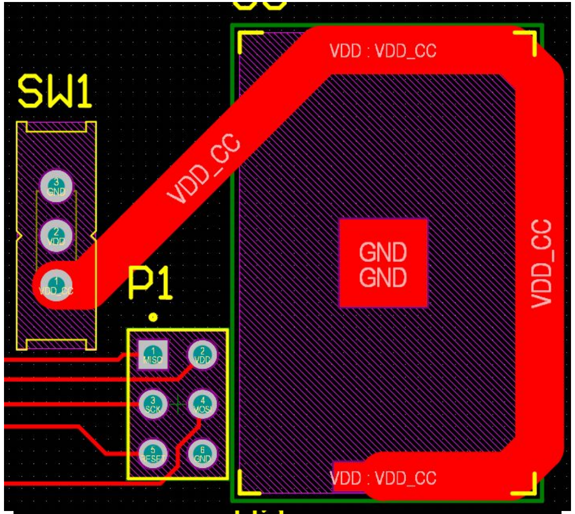
Polygon Pours

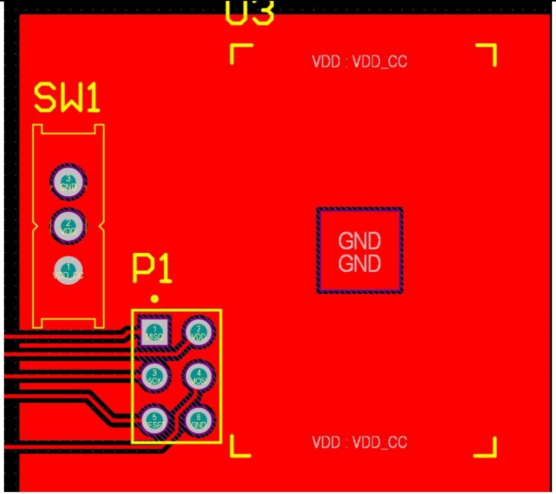
How you make a ground plane on signal layer

Can also be used for other signals

Useful for high power, thermal requirements

Can simply use a high width trace, but won't smartly pour around non-net objects





Trace Rules

Size of trace determines current capacity - use trace width calculator

Don’t change size of trace connecting two points

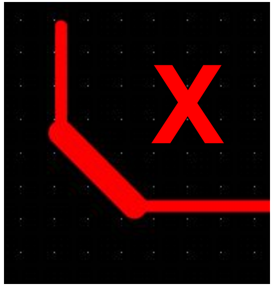
Inner layers must use thicker traces

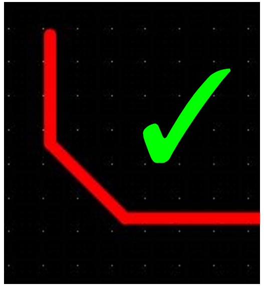
- Insulated by dielectrics
- Often thinner, check stackup

Longer traces need to be thicker

Controlled impedance for certain types of signal (USB, HDMI, Ethernet, etc) - more on advanced

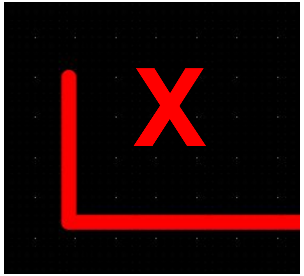
Right angles bad, 45 degree angles good

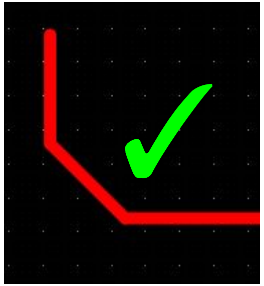




$$R = \frac{\rho L}{A}$$

ρ = resistivity
L = length
A = cross sectional area





Vias

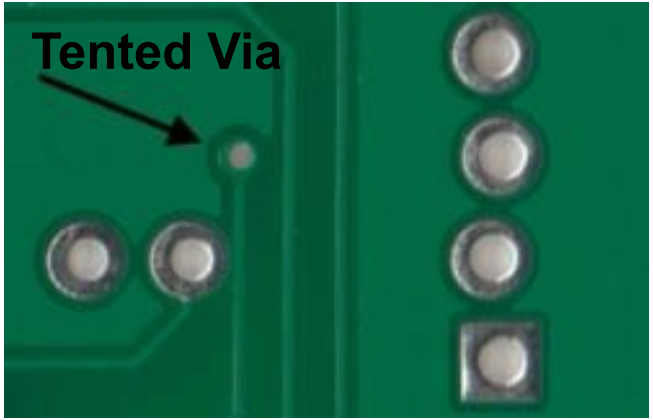
Not on pads - bad for manufacturing

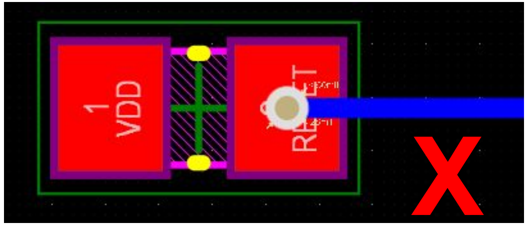
- Solder paste can wick into via during reflow, leaving little solder on the pad for good connection
- You can, for more $$, do this if necessary

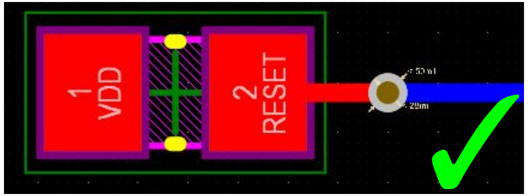
Tenting vias to prevent damage/oxidation

- Covers them in solder mask
- Also prevents solder wicking issue near smt pads
- If not covered, can be used as test points

Try to use consistent sizes - determines number of drill bits used ($$)







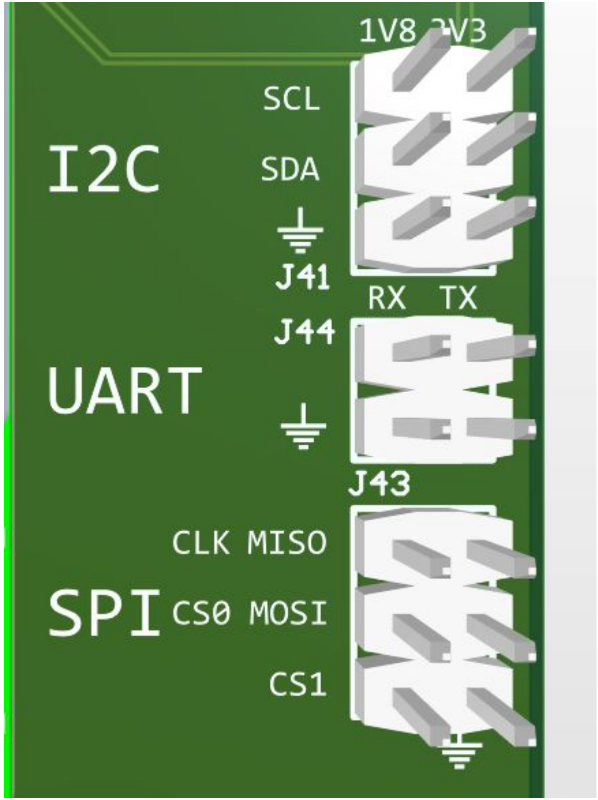
Use the Silk Screen

Plugs - very clear ground/directional label

LEDs - if indicators, what do they mean?

Designators should be visible - don't put them below pads

You can also put fun artwork and such to personalize your pieces :)



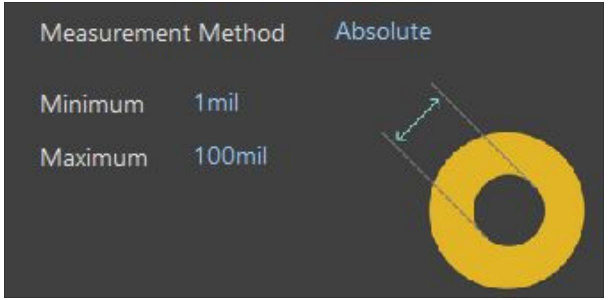
Design Rules

Define how you want board to operate

Examples: min trace width, min distance between components, etc

Should check with manufacturer on their limitations

Run DRC (Design Rule Check) frequently



Other tips and tricks

Clock line symmetry - need good clean clock lines for good timing

Make sure headers have room, including mating parts

Easy to get CAD goggles - switch between 2D and 3D frequently to ensure it is logical