Quick an' Dirty PCB Hints

or

"Shadow of the Colossus."

4AM Edition!

Paul Pounds

19 April 2016

University of Queensland

But first…

Some house keeping

Calendar at a glance

Week	Dates	Lecture	Reviews	Demos	Assessment submissions
1	2/3 – 6/3	Introduction			
2	7/3 – 11/3	Principles of Mechatronic Systems design			Problem analysis
3	14/3 – 18/3	Professional Engineering Topics			
4	22/3 – 25/3	Your soldering is (probably) terrible	Progress review 1		
Break	28/3 – 1/4				
5	4/4 – 8/4	Introduction to Teleoperation			
6	11/4 – 15/4	Q&A 1			
7	18/4 – 22/4	PCB Hints	Progress seminar	25% demo	
8	25/4 – 29/4	Switch to			
9	2/5 – 6/5	Q and A		50% demo	
10	9/5 – 13/5	sessions	Progress review		
11	16/5 – 20/5			75% demo	Preliminary report
12	23/5 – 27/5				
13	30/5 – 3/6	Closing lecture		Final testing	Final report and reflection

You are here

Progress Seminars

- Mostly done… only a few more to go!

- Most groups have done pretty well so far
  - But not everybody…

- But overall teams are much better prepared!
  - Yay!

FAQ Roundup

- **None as yet**

Right.

Onwards to PCB design!

"Quick and dirty"

- Printed Circuit Board design is a huge topic and I am in the middle of writing a much longer lecture on all the intricacies of it.

- The complete monstrous shambling thing will probably be 200+ slides long and push the class to the brink of insanity.
  - The Garhugeian Cometh…

What is a PCB?

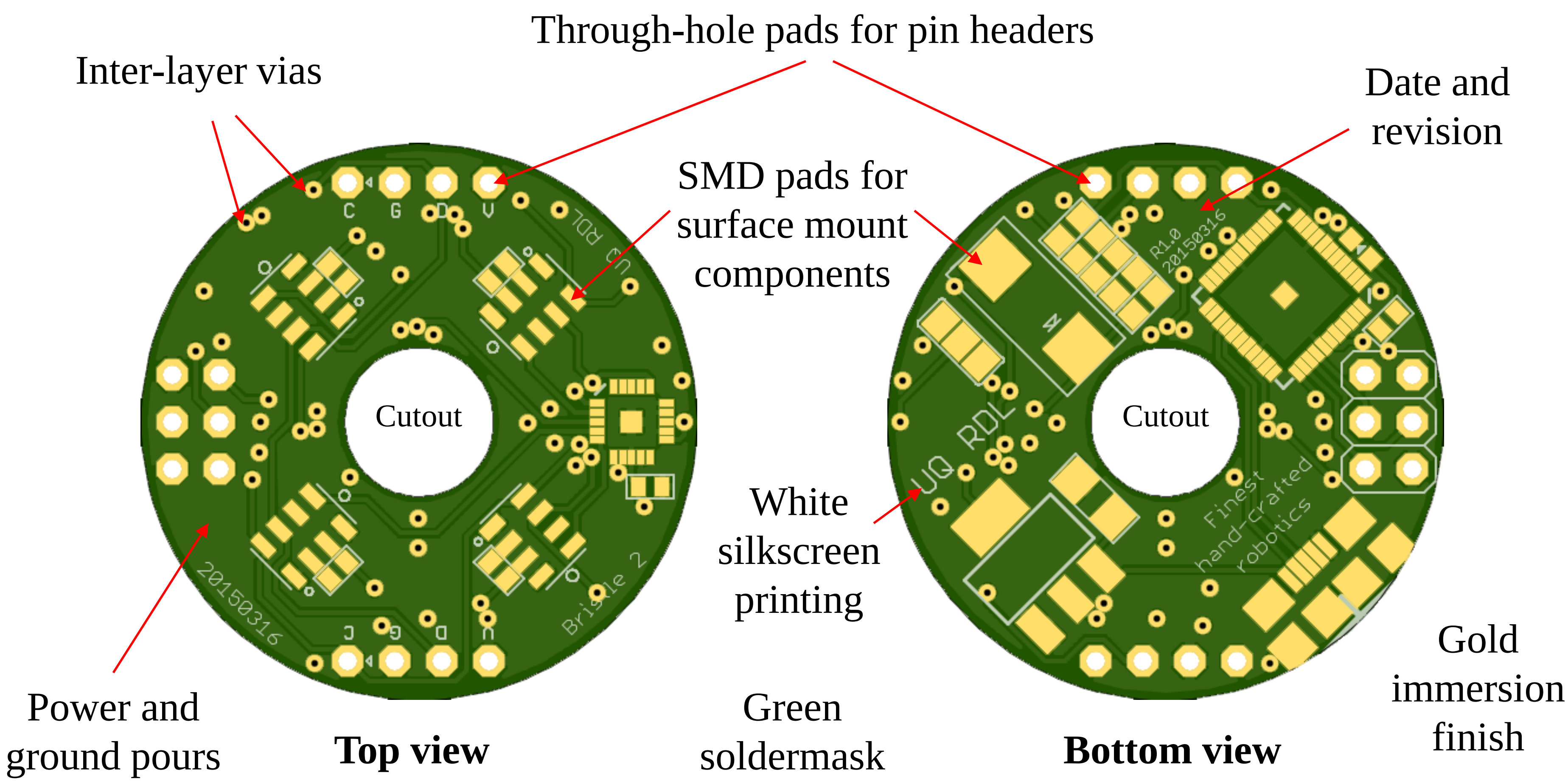
You should know what a PCB is/does.

But what goes into it? And how is it made?

And why would they even call it a “printed circuit board”, anyway?

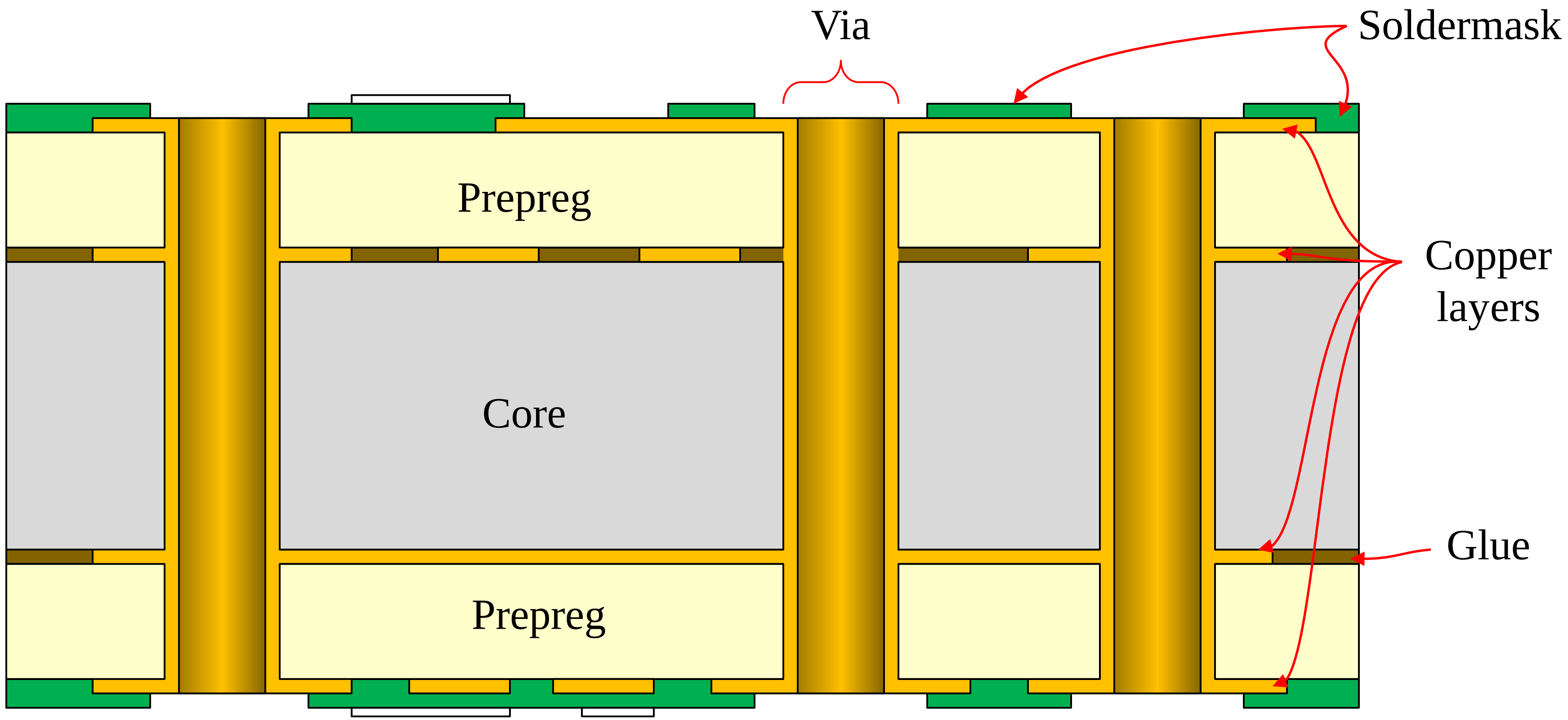
PCB structure

- Stuff on a PCB – example 4 layer board



PCB structure

- Stack of conductive and insulating material

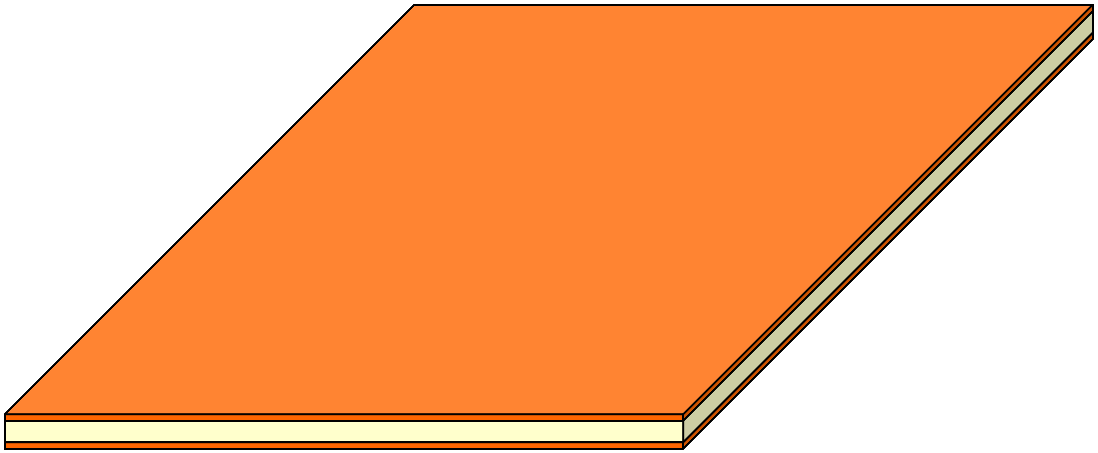


Handy summary of terms

- Layer: Horizontal sheet of conductive metal
- Substrate: Horizontal sheet of rigid insulator
- Soldermask: Resistive surface coating
- Silkscreen: Surface writing and graphics
- Track: Conductive line on a layer
- Pad: Solder point for component mounting
- Via: Vertical conductor for connecting layers
- PTH: Pin Through-Hole; parts with pins, leads, legs
- SMD: Surface Mount Device; flat mounting parts

How a PCB is made

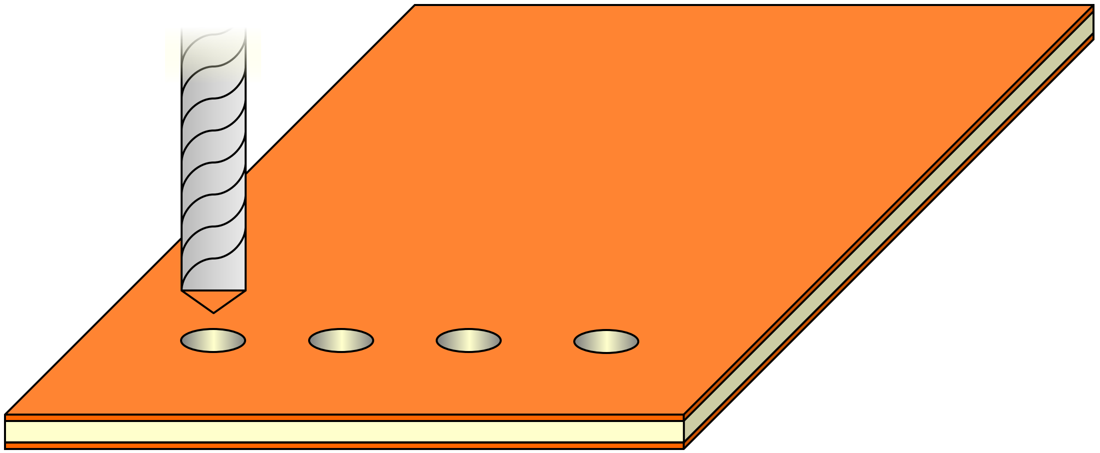
- Start with copper-clad prepreg sheet



Start!

How a PCB is made

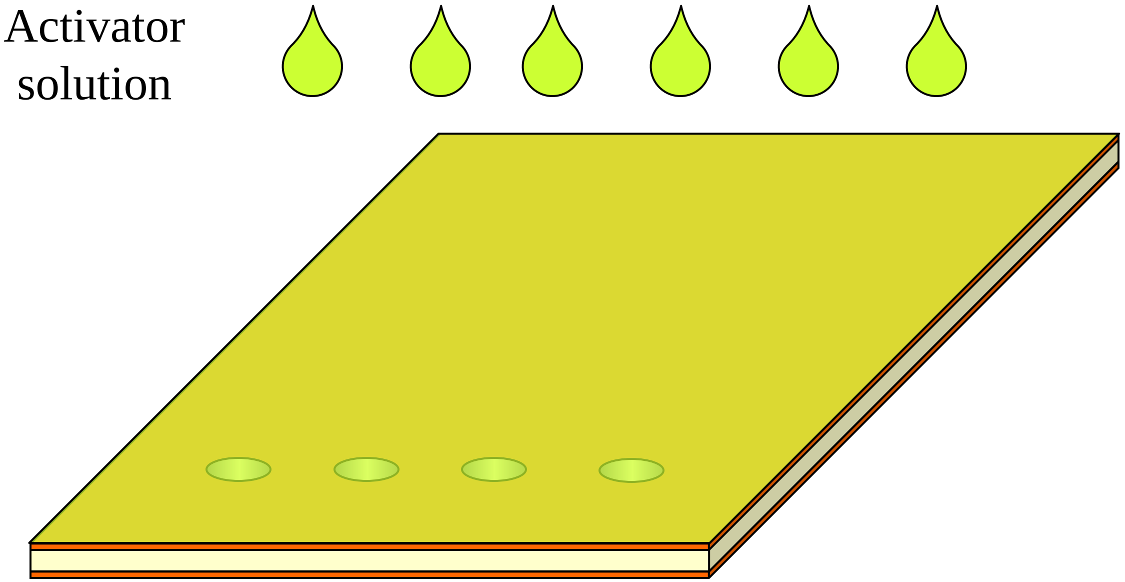
- Drill where holes will go



“Drilling”

How a PCB is made

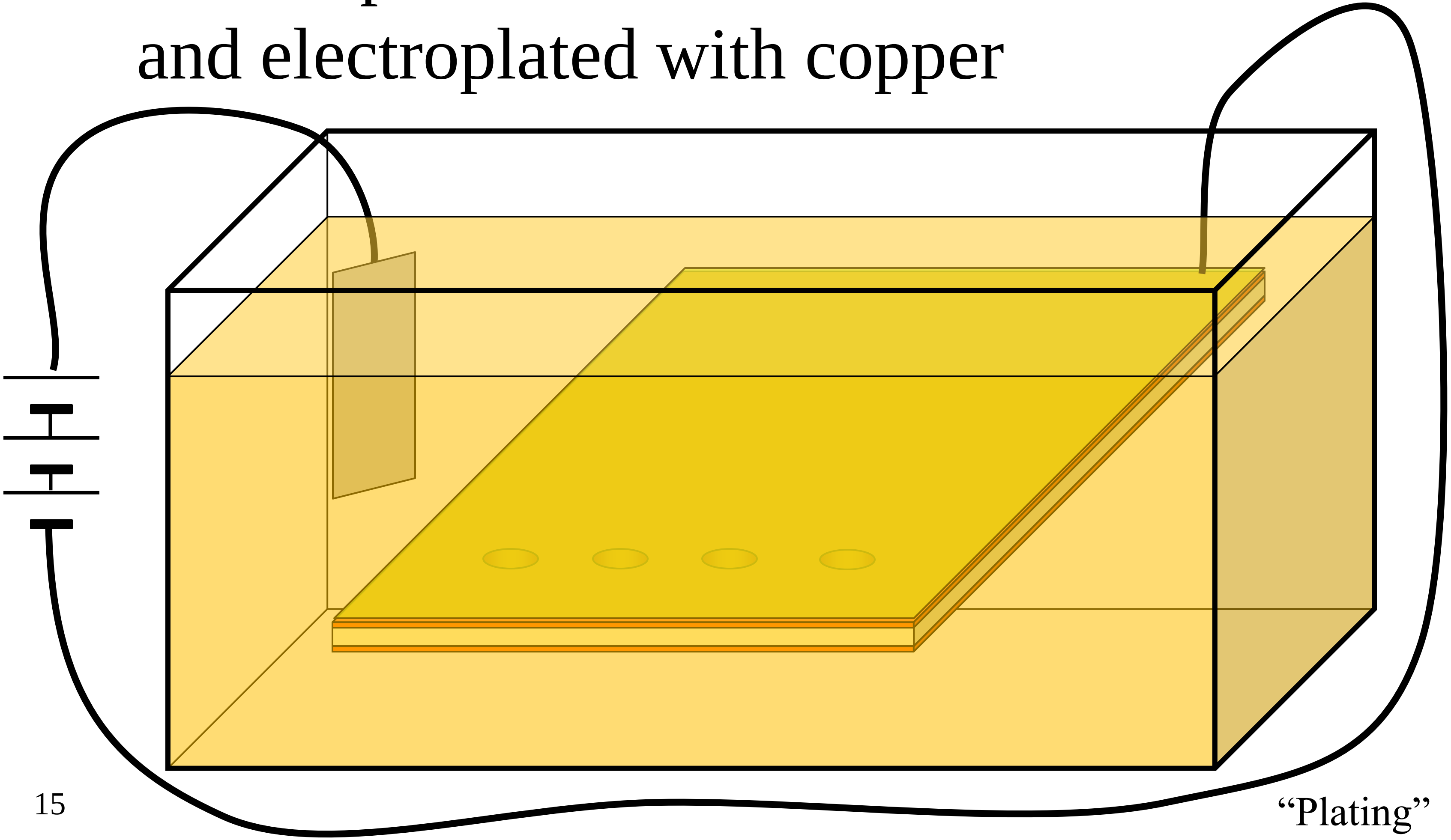
- Wash the board with an electrochemical activator solution



“Activation”

How a PCB is made

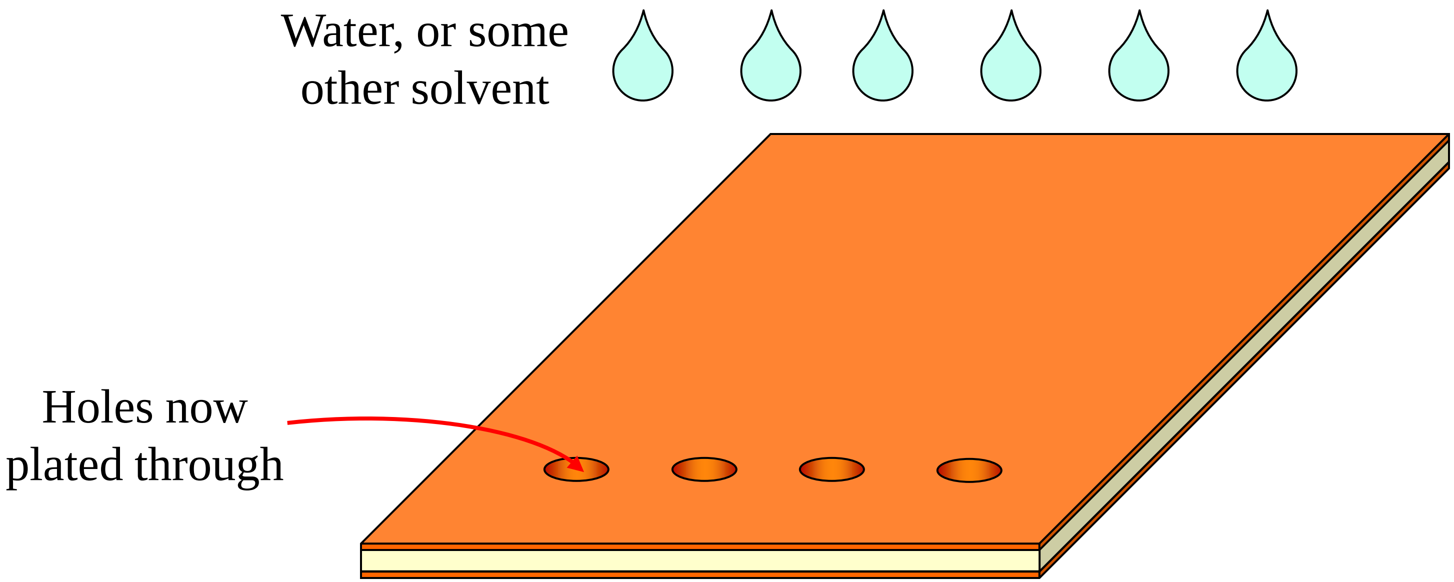
- PCB is placed in an electrochemical bath and electroplated with copper



“Plating”

How a PCB is made

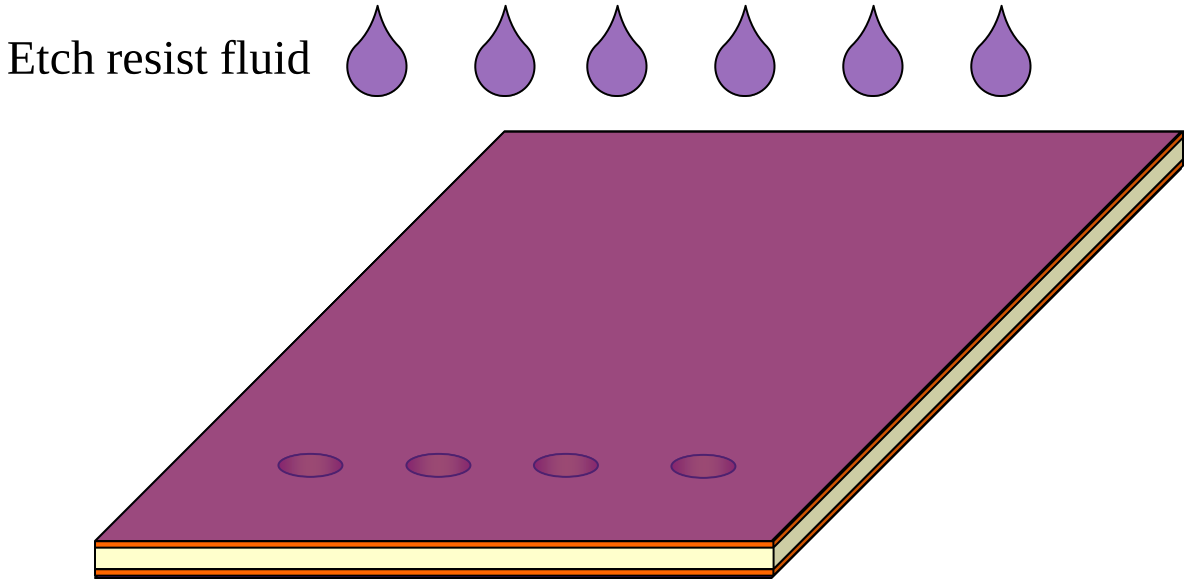
- PCB is removed from tank and washed



"Washing"

How a PCB is made

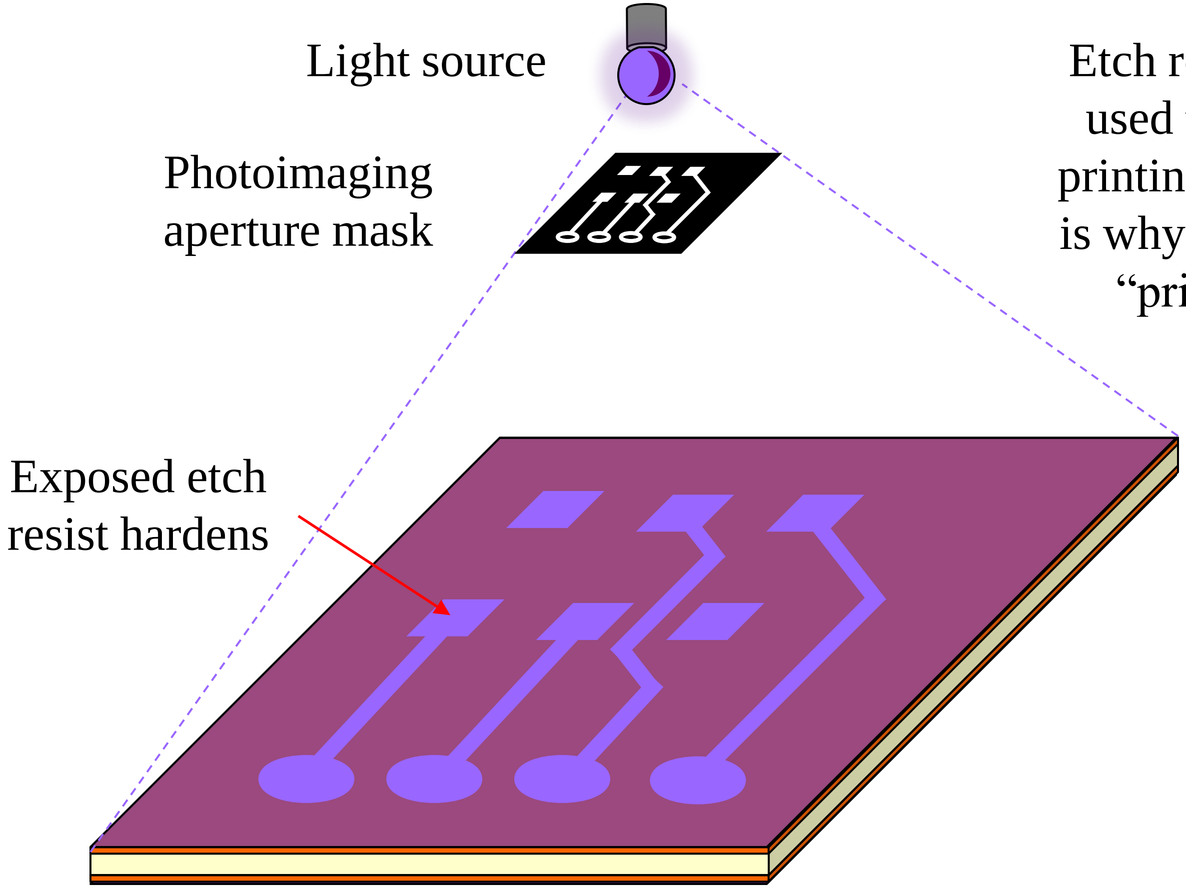
- Coat with photo-curing etch resist



“Resist coating”

How a PCB is made

- Selectively expose to light (one or both sides)

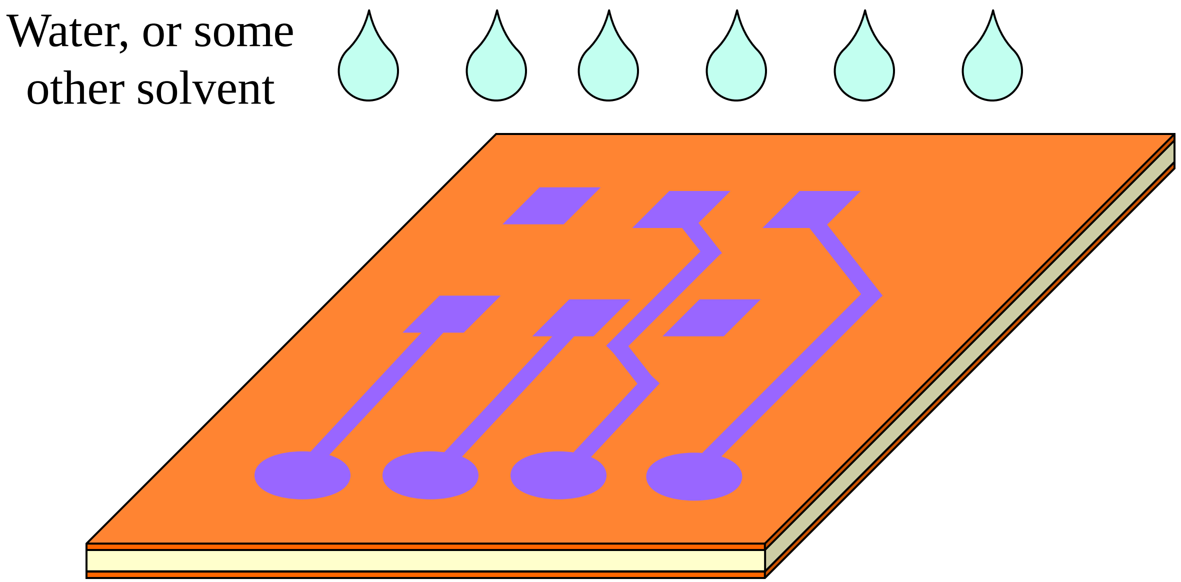


Etch resist application used to be done by a printing method, which is why PCBs are called “printed” circuits

“Photoimaging”

How a PCB is made

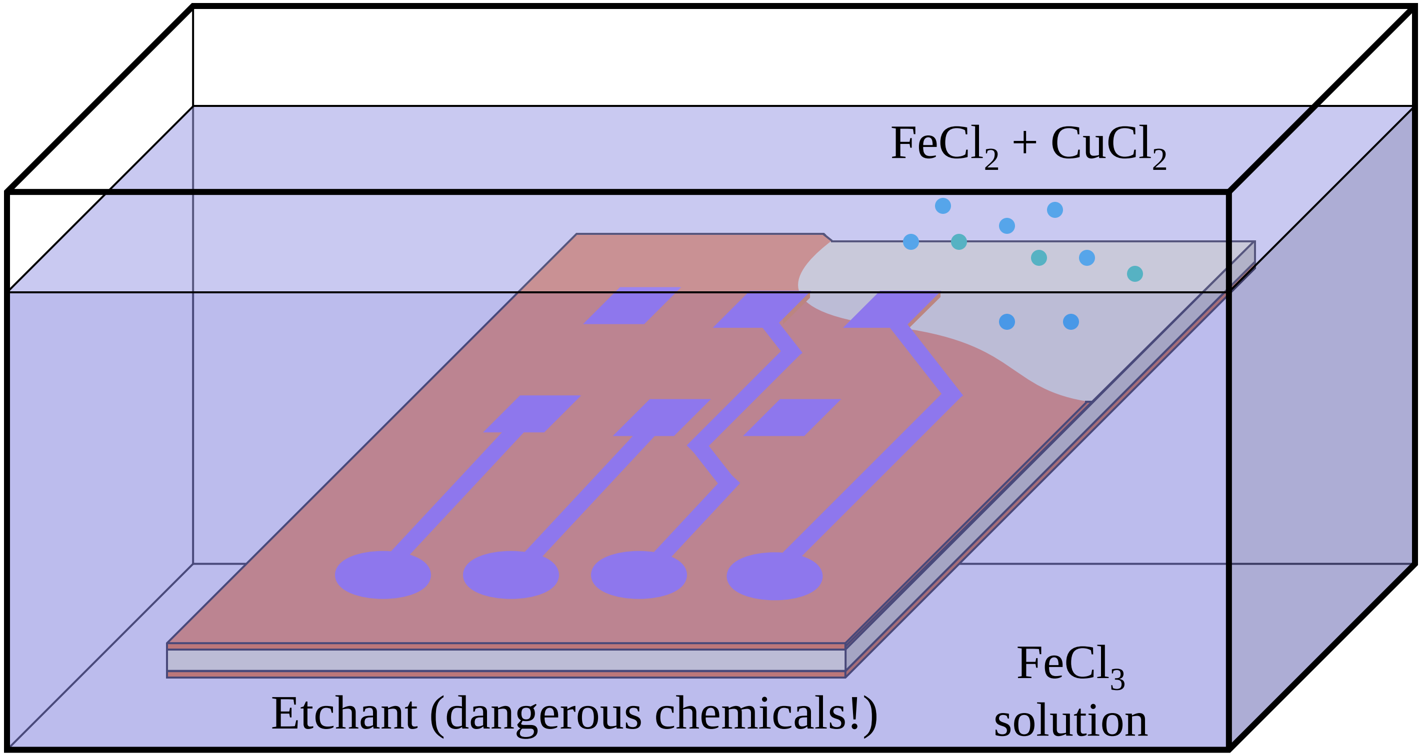
- Unhardened etch-resist is washed away, exposing uncoated copper



"Cleaning"

How a PCB is made

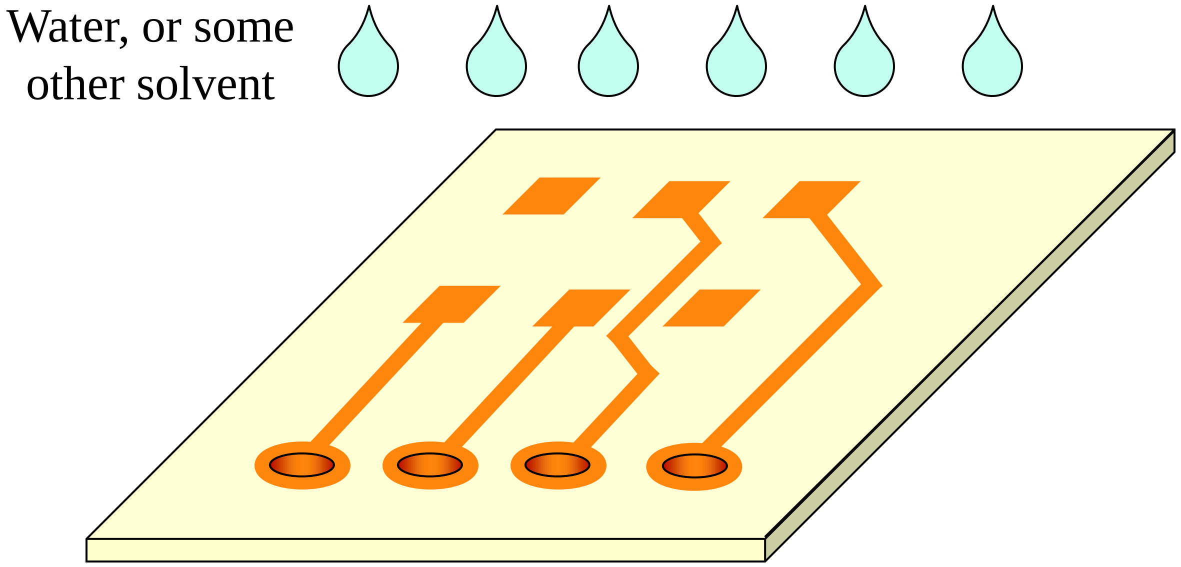
- PCB is placed in a chemical bath of etchant that reacts to dissolve exposed copper



"Etching"

How a PCB is made

- Removed from etchant, washed, and the etch-resist is removed

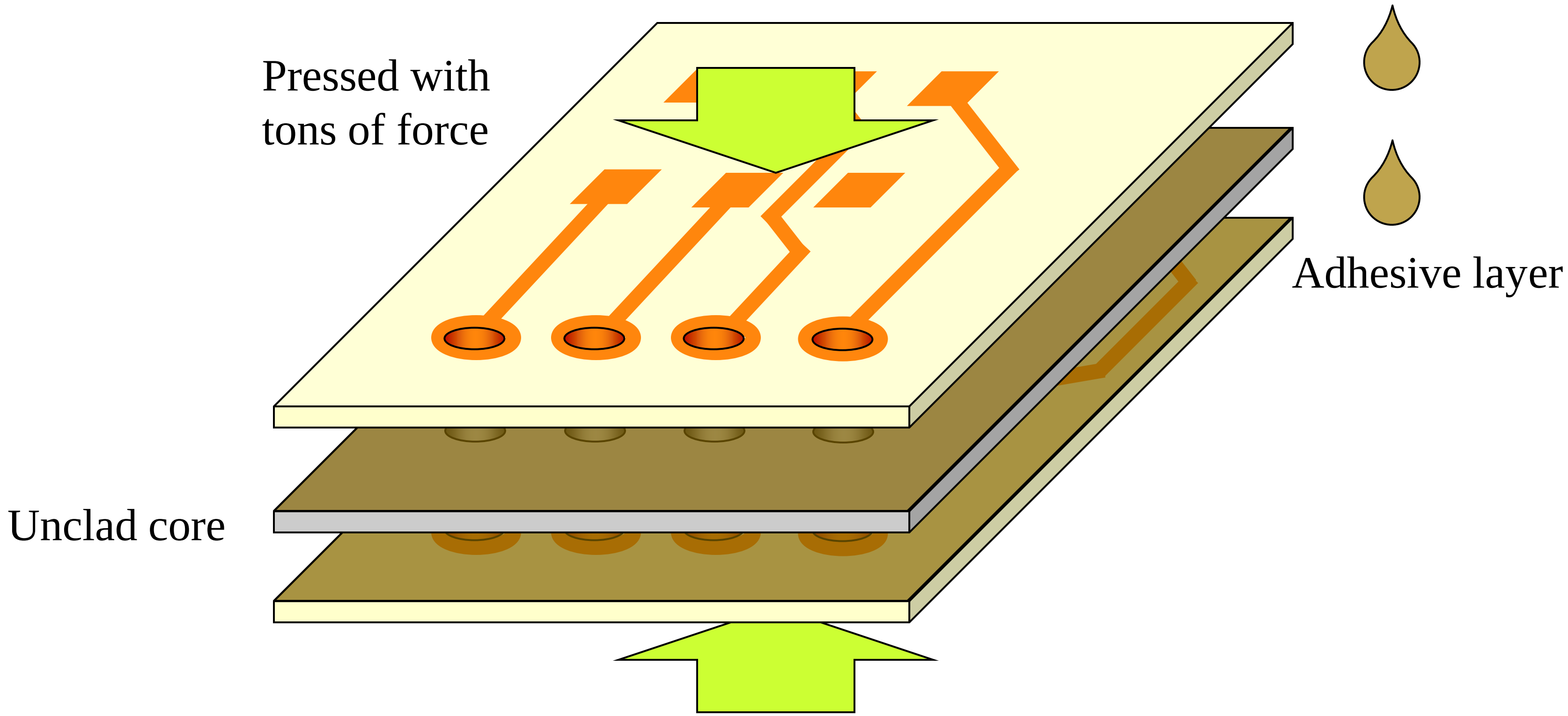


Simple single-layer boards could stop here

“Cleaning”

How a PCB is made

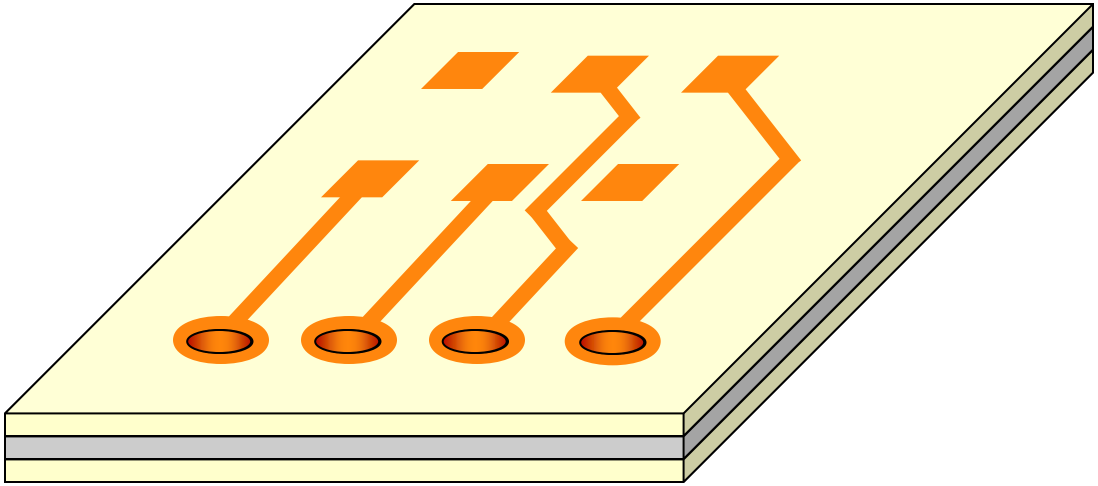
- Multilayer board layers are coated with glue, aligned and then pressed together



“Laminating”

How a PCB is made

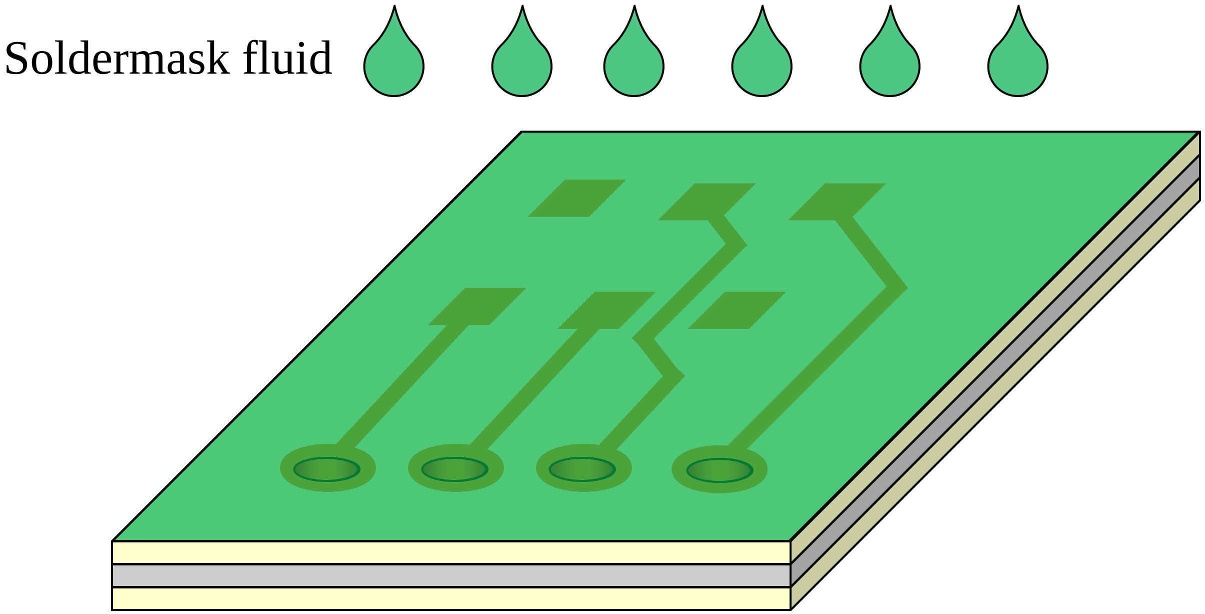
- The board stack is allowed to cure under pressure (may require application of heat)



“Curing”

How a PCB is made

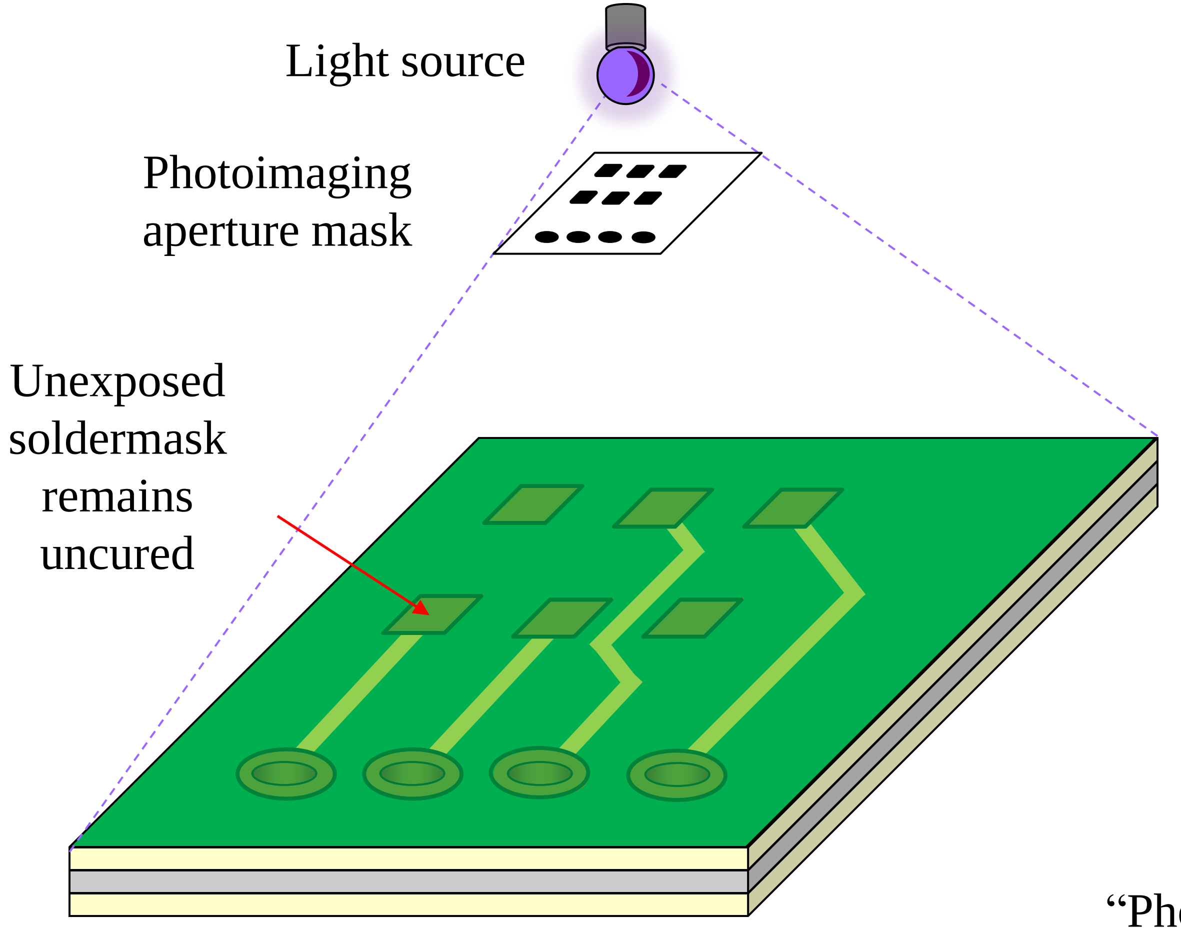
- The board is coated with liquid soldermask



“Soldermask”

How a PCB is made

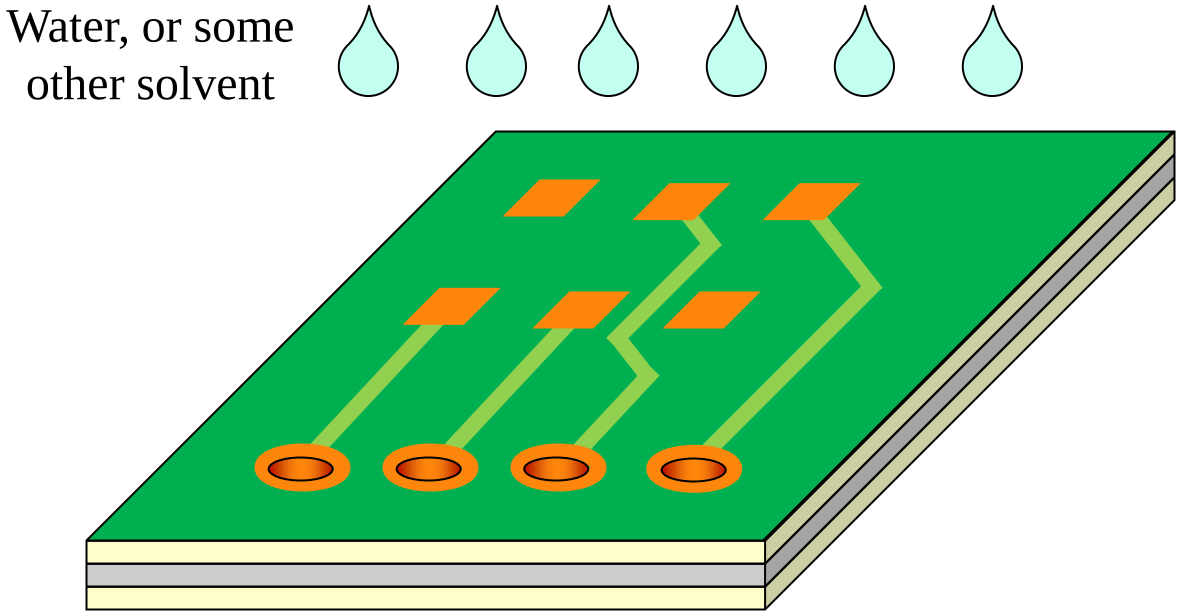
- Selectively expose to light (one or both sides)



“Photoimaging”

How a PCB is made

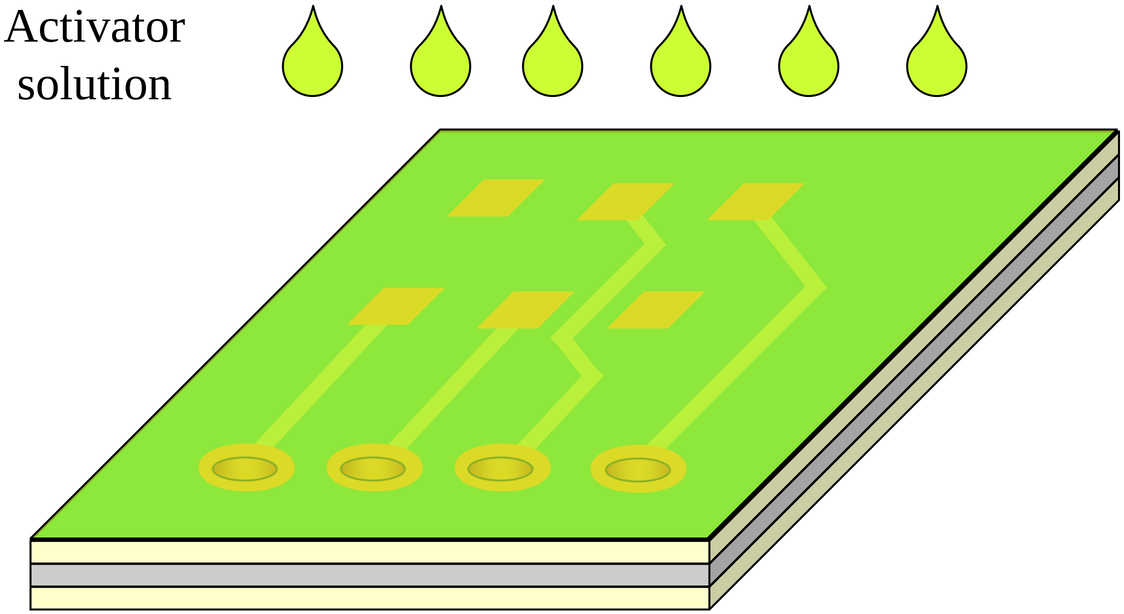
- Wash uncured soldermask away, exposing copper pads, vias and holes



“Washing”

How a PCB is made

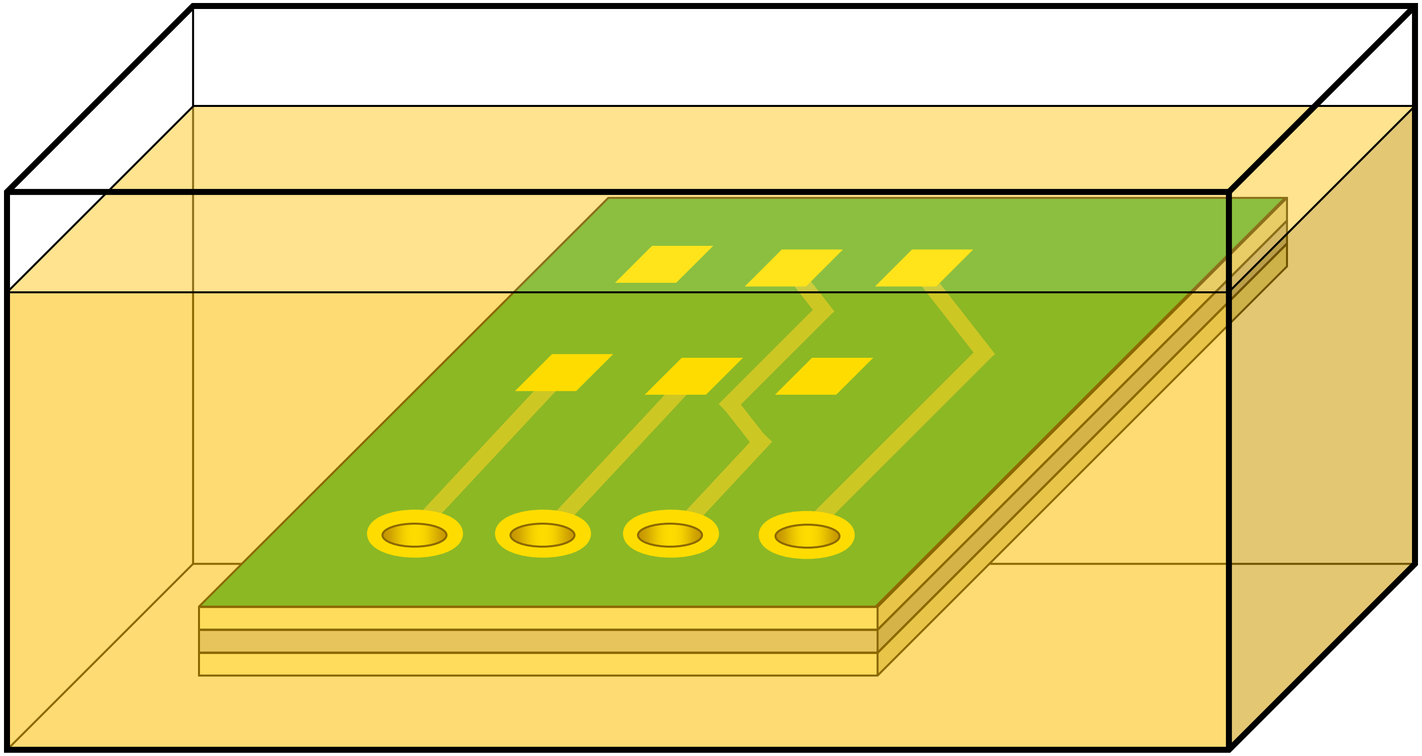
- Wash the board with an electrochemical activator solution



“Activation”

How a PCB is made

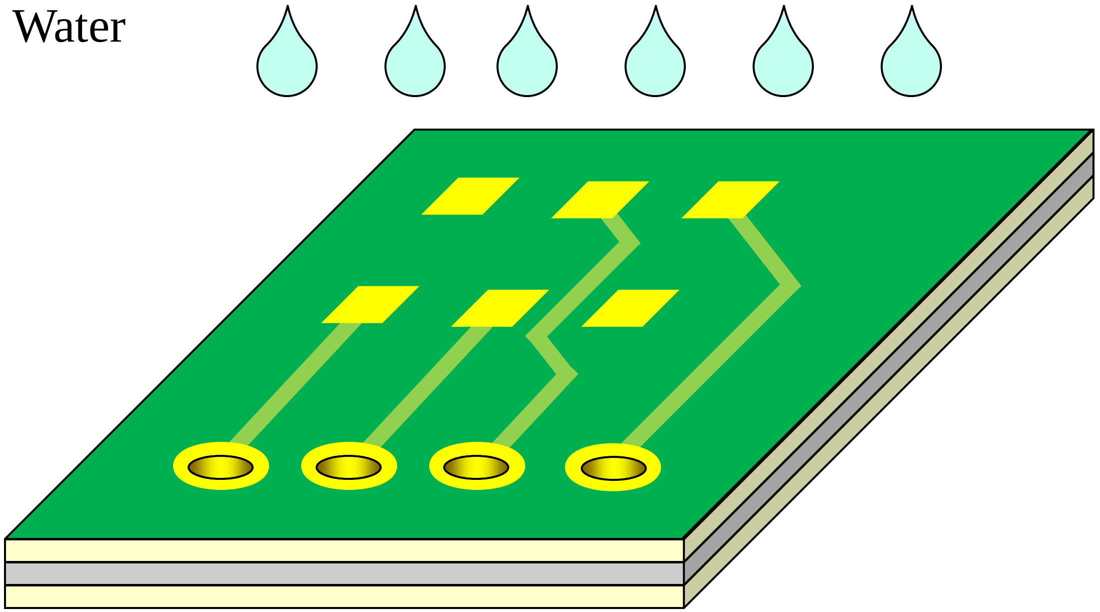
- PCB is placed in a chemical bath and plated with nickel, gold or other metals



"Plating"

How a PCB is made

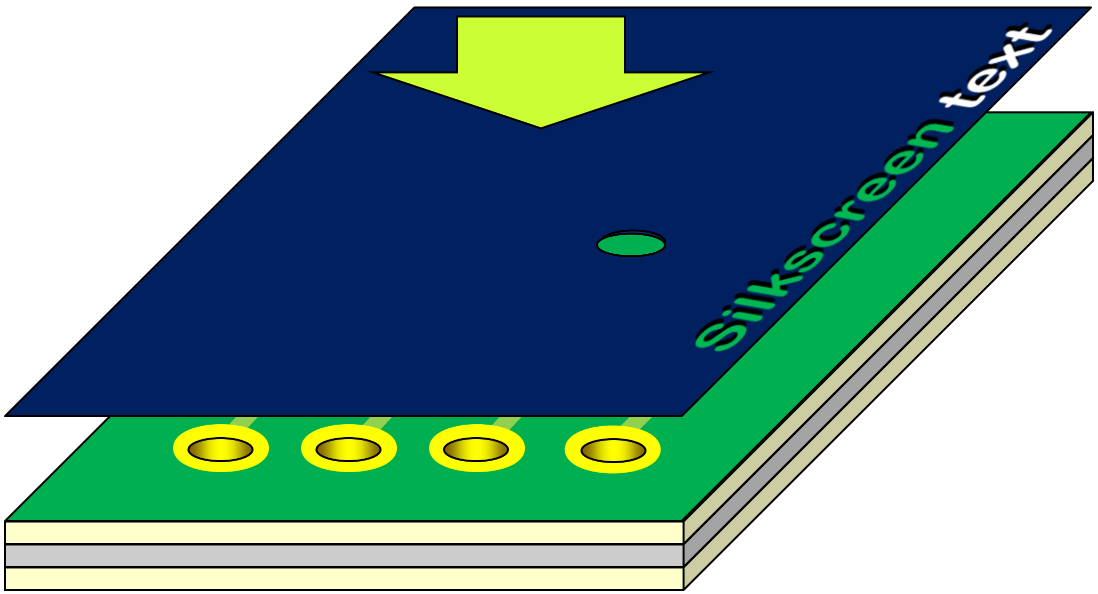
- PCB is washed clean of chemicals



“Washing”

How a PCB is made

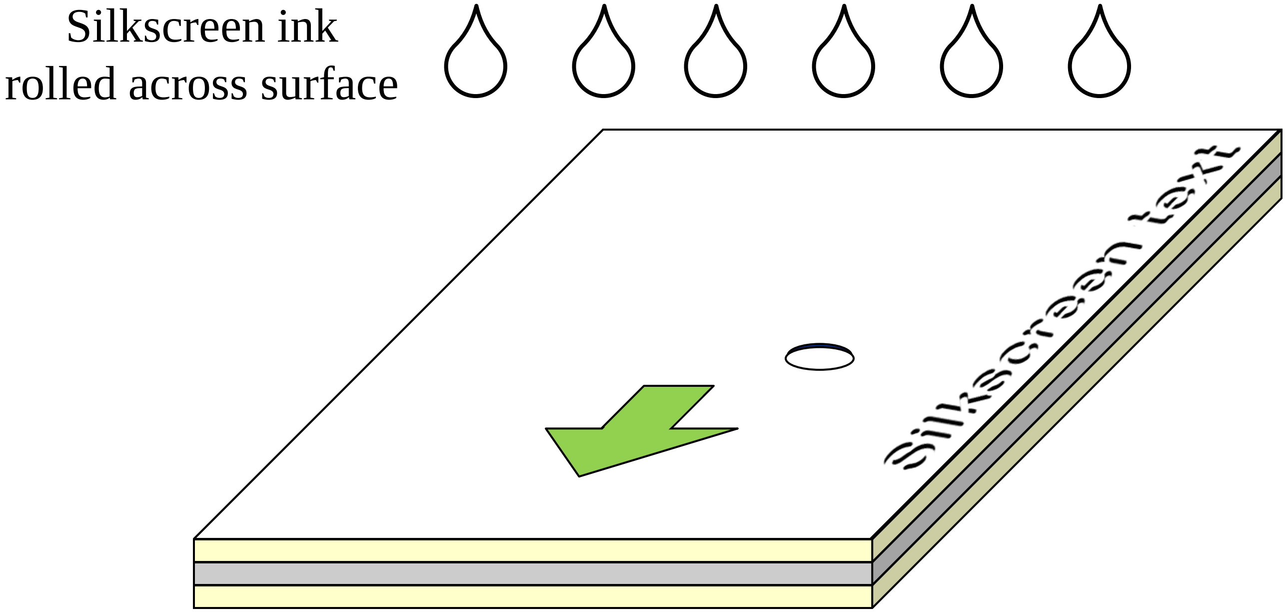
- Silkscreen stencil is laid on; made using a photo-cured emulsion



"Stencilling"

How a PCB is made

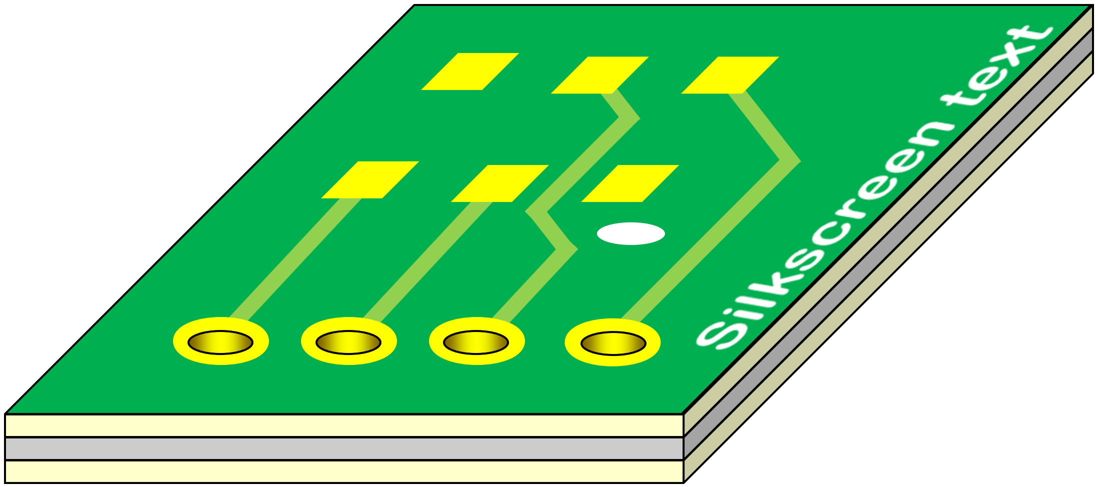
- Ink is applied to the stencil, filling the voids



“Inking”

How a PCB is made

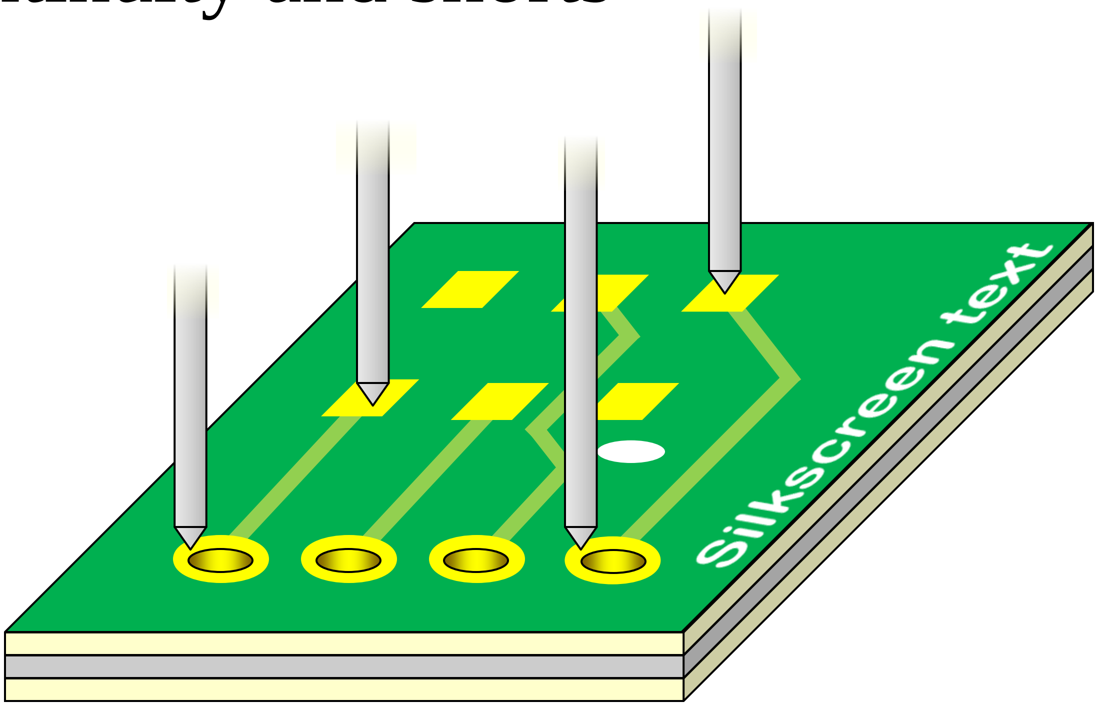
- Silkscreen stencil is removed and ink allowed to dry



“Drying”

How a PCB is made

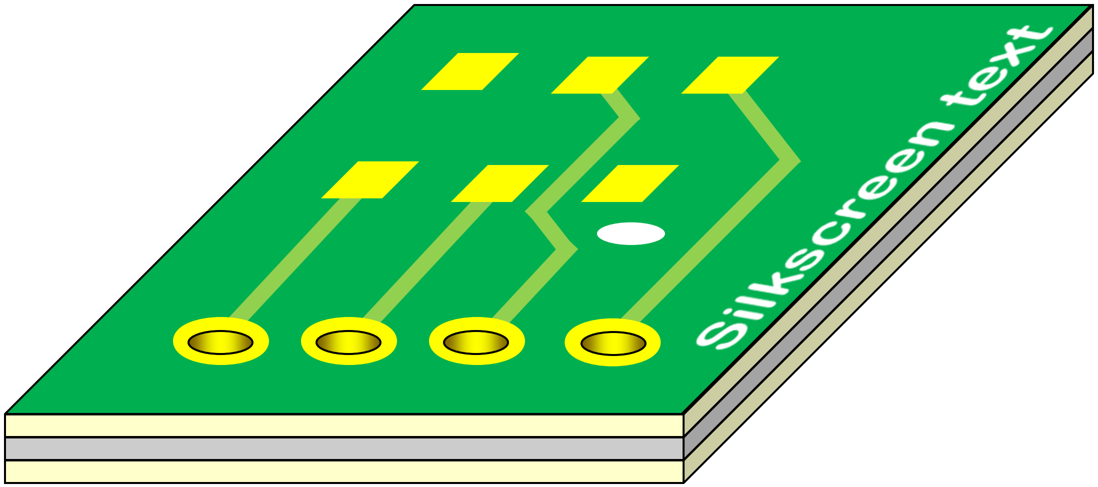
- Finally, the electrical connections are tested for continuity and shorts



“E-test” or “Bed of nails”

How a PCB is made

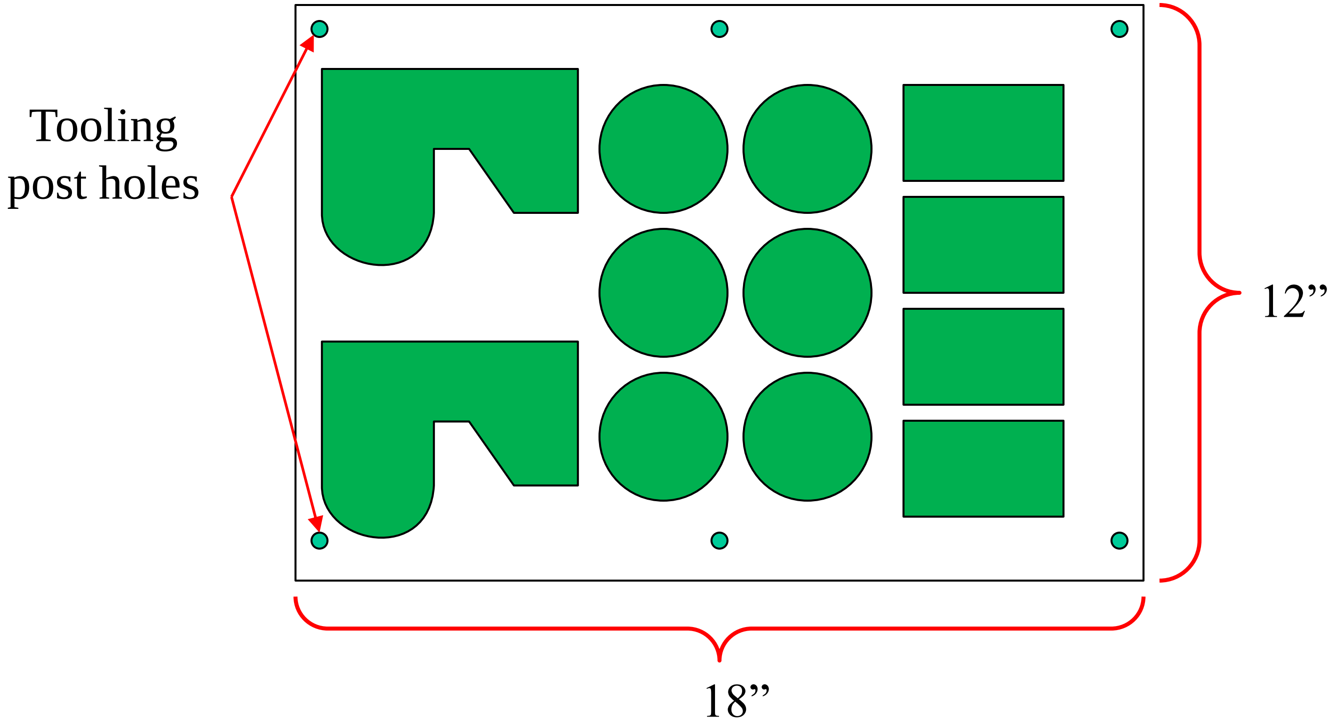
- Boards that pass all tests are complete



Done!

Panelisation

- A lot of work goes into each board!
- Virtually all boards are made in panels, with multiple units fabricated together



Critical design parameters

Stuff to know about your design:

- Thickness of copper layers and total lay-up
- Board material (typically FR4)
- Minimum track width and spacing
  - How thick is your smallest track? How much space between copper with different nets?
- Minimum drill diameter and via resting
- Clearance between copper and board edge

Common design specs

Spec	**Easy**	**Typical**	**Hard**
Layer	1-2	4	8+
Tracks	0.4 mm (16 mil)	0.2 mm (8 mil)	0.1 mm (4 mil)
Spacing	0.4 mm (16 mil)	0.2 mm (8 mil)	0.1 mm (4 mil)
Drills	0.6 mm (24 mil)	0.4 mm (16 mil)	0.3 mm (12 mil)
Annual ring	0.4 mm (16 mil)	0.2 mm (8 mil)	0.1 mm (4 mil)
Board thickness	1.6 mm (63 mil)	1.6 mm (63 mil)	0.8 mm (63 mil)
Copper thickness		35 μm (1 Oz)	70 μm (2 Oz)
Board material		FR4	RF substrates

If your stack up is unusual, you'll need to specify it in detail – sometimes you'll even require a technical drawing just so there is no misunderstanding about what you want.

What specs do I need?

- It's hard to know what you need until you're actually layout out your design
  - But it's good to start laying out with close to the right size – can usually guesstimate how challenging your board will be to route

Generally, the more complex and cramped your board, the tighter your specs will be

On that note…

What stuff do you actually have to provide the when you're submitting a PCB?

Anatomy of a PCB submission

- Spec sheet
- Mechanical layer (outline)
- Copper layers
- Stop layers (soldermask)
- Silkscreens
- Cream layers (solder paste stencil)
- Part origins
- Bill of materials (BOM)

Spec sheet

Part number: uq_samara_05
Rev: 1
Quantity: 10 min
Time Required: Fastest turnaround
Number of Layers: 4
Material: FR4
Board Thickness: <1 mm
Line width (min): 0.2mm
Line width (norm): 0.2mm
Line Spaceing (min): 0.2mm
Line Spaceing (norm): 0.2mm
Copper thickness: 35 um
Soldermask color: Any
Soldermask type: Photoimageable
Silkscreen color: White
Silkscreen sides: One side

File Type: Gerber

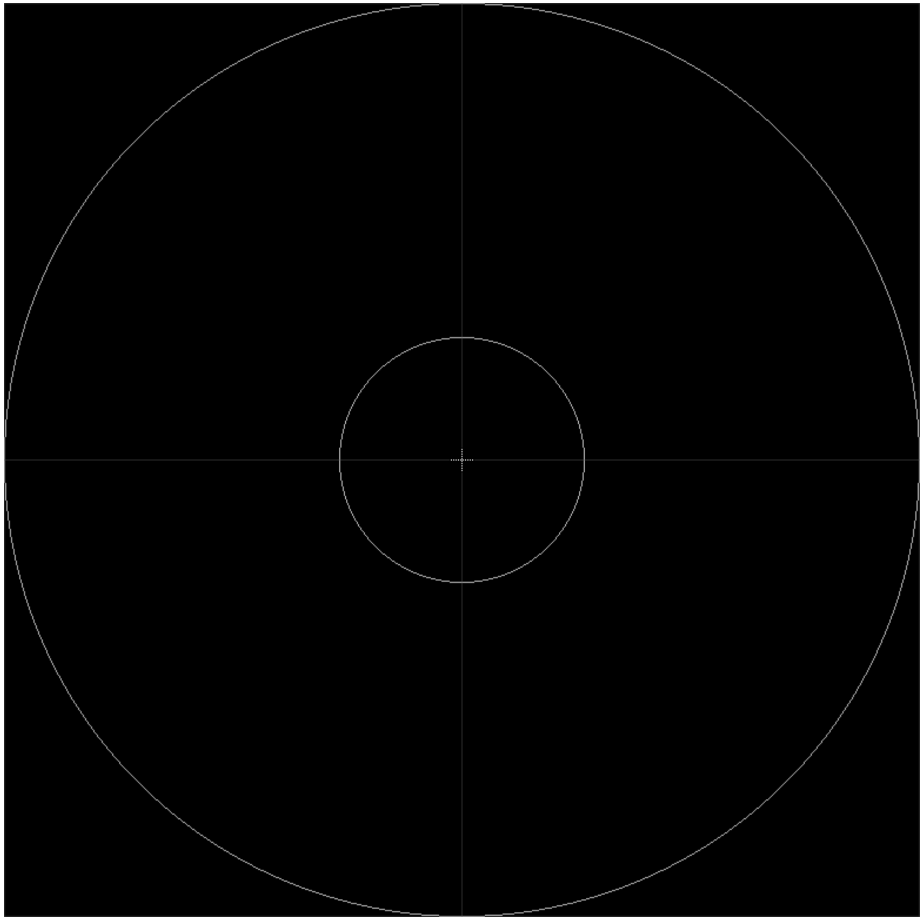
Gerber Information
Character Set: ASCII
Drill file: Excellon

Files:

uq_samara_05.cmp	Layer 1 - Top Signal
uq_samara_05.l02	Layer 2 - Signal Layer
uq_samara_05.l03	Layer 3 - Signal Layer
uq_samara_05.sol	Layer 4 - Bottom Signal
uq_samara_05.stc	Top soldermask
uq_samara_05.sts	Bottom soldermask
uq_samara_05.plc	Top silkscreen
uq_samara_05.pls	Bottom silkscreen (empty)
uq_samara_05.dri	Drill file

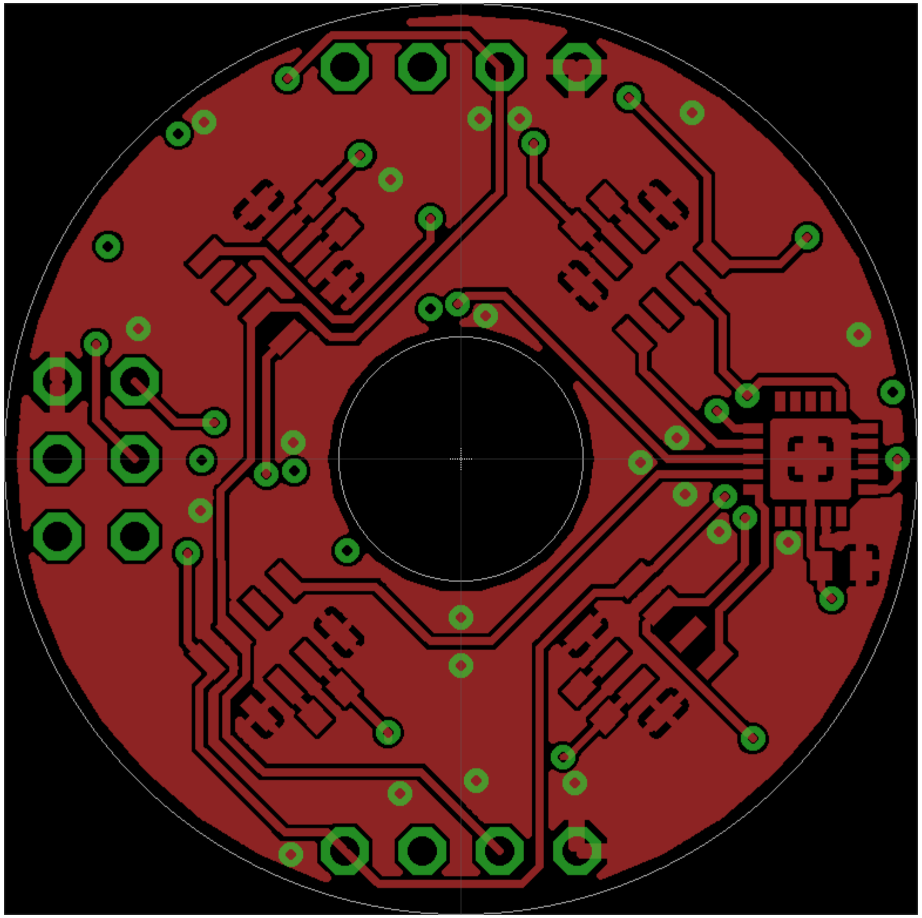
Mechanical layer

- Boring simple drawing of the board outline



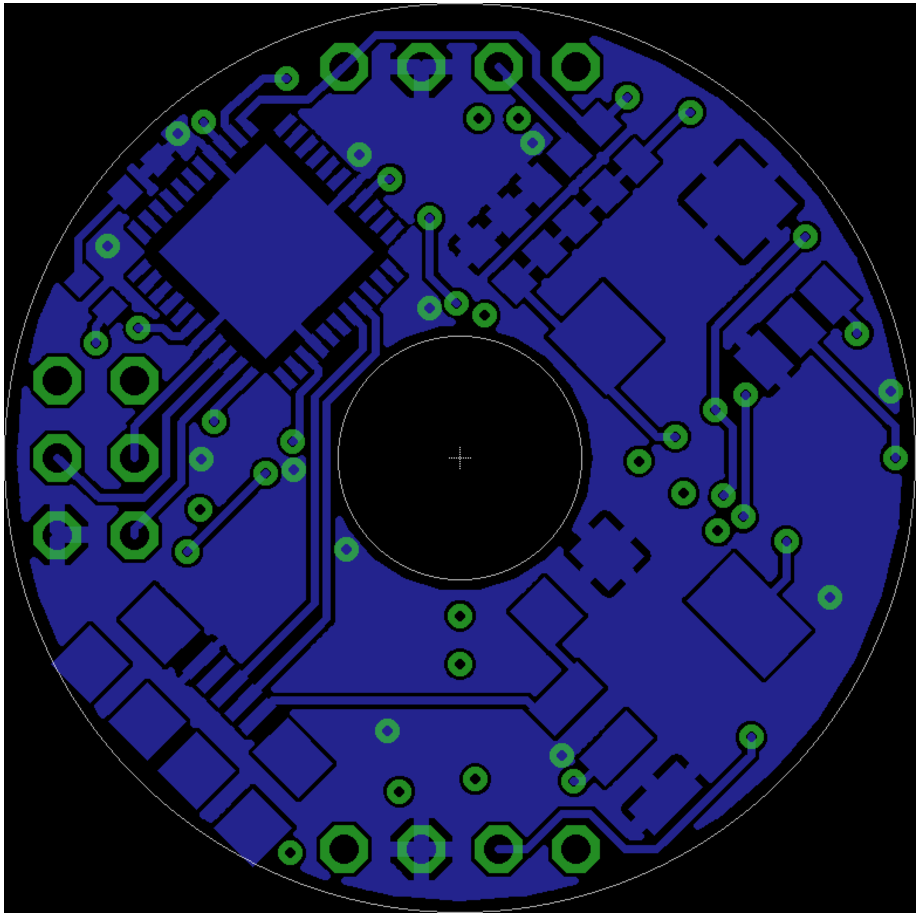
Top layer

- Aka "component side" – tracks plus pads,vias



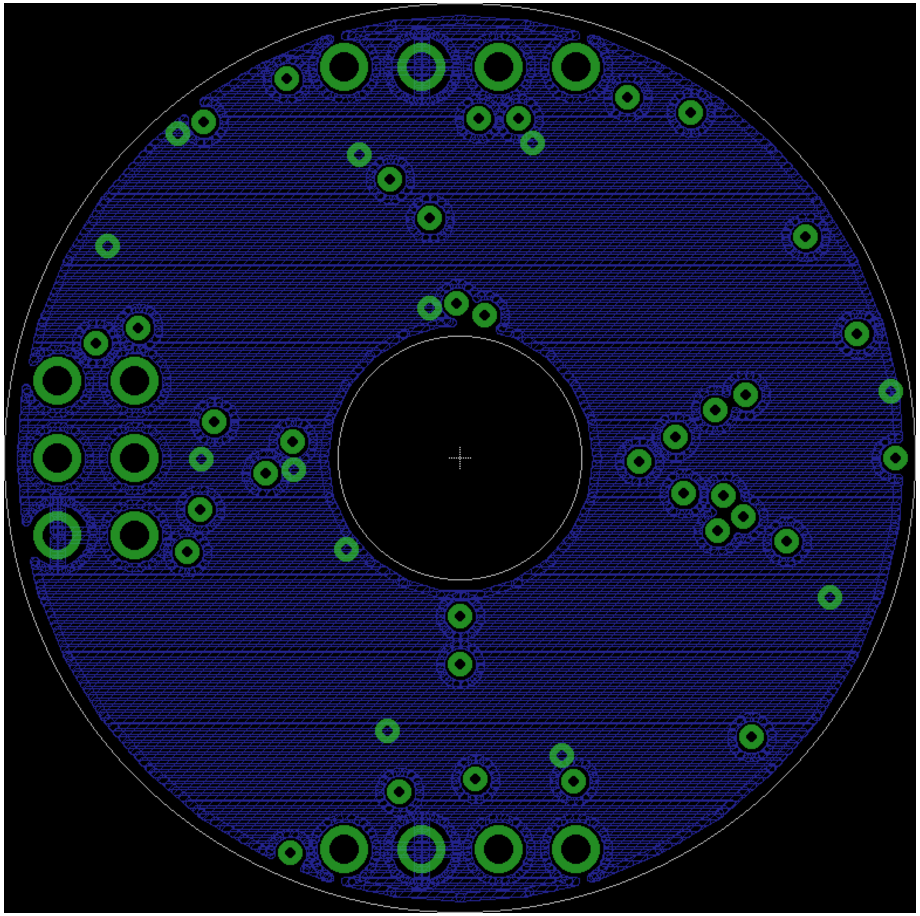
Bottom layer

- Aka "solder side" – tracks plus pads,vias



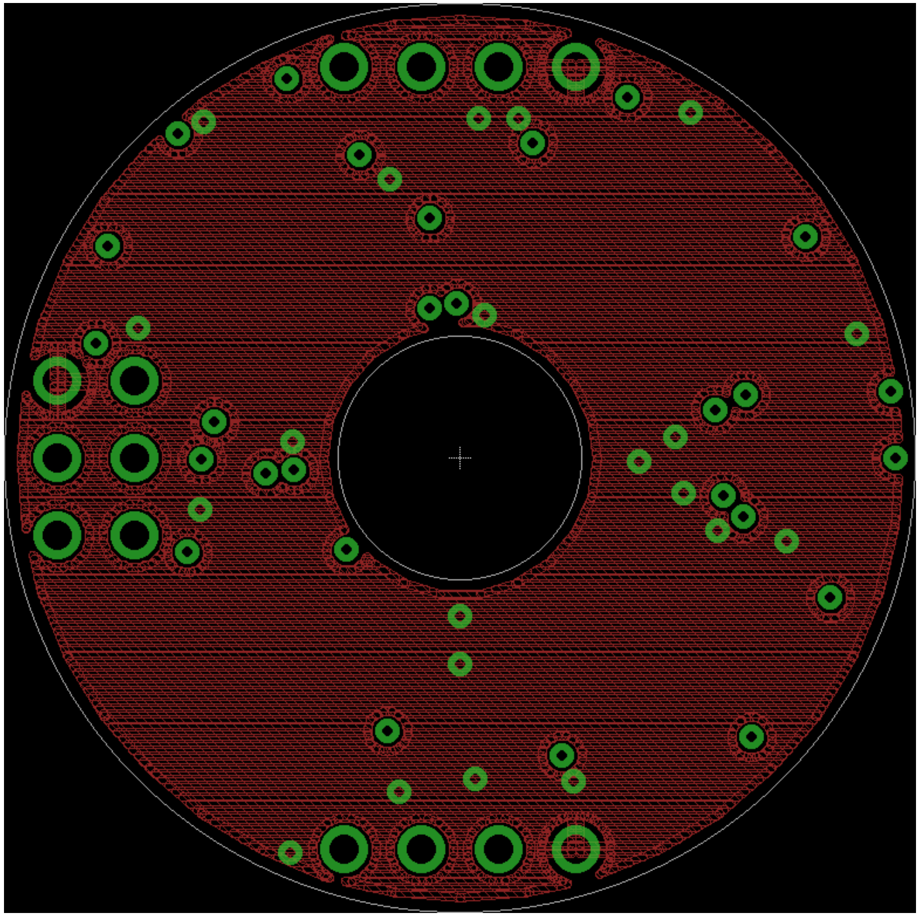
Inner layer 1

- Middle ground plane



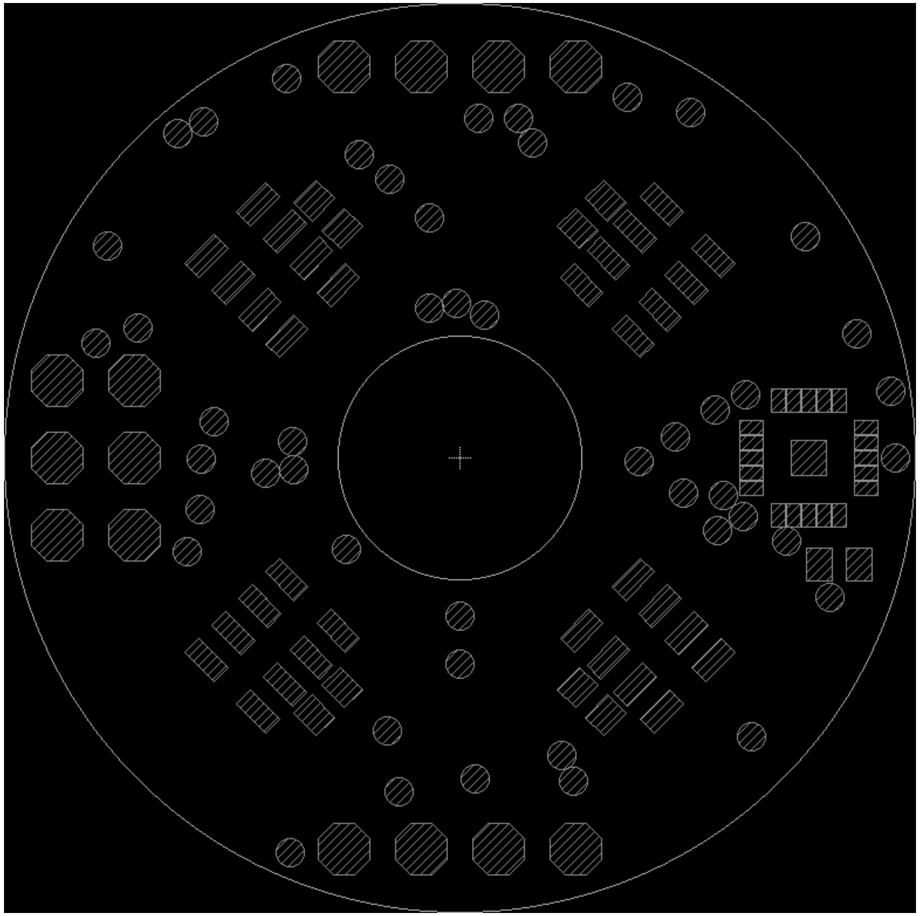
Inner layer 2

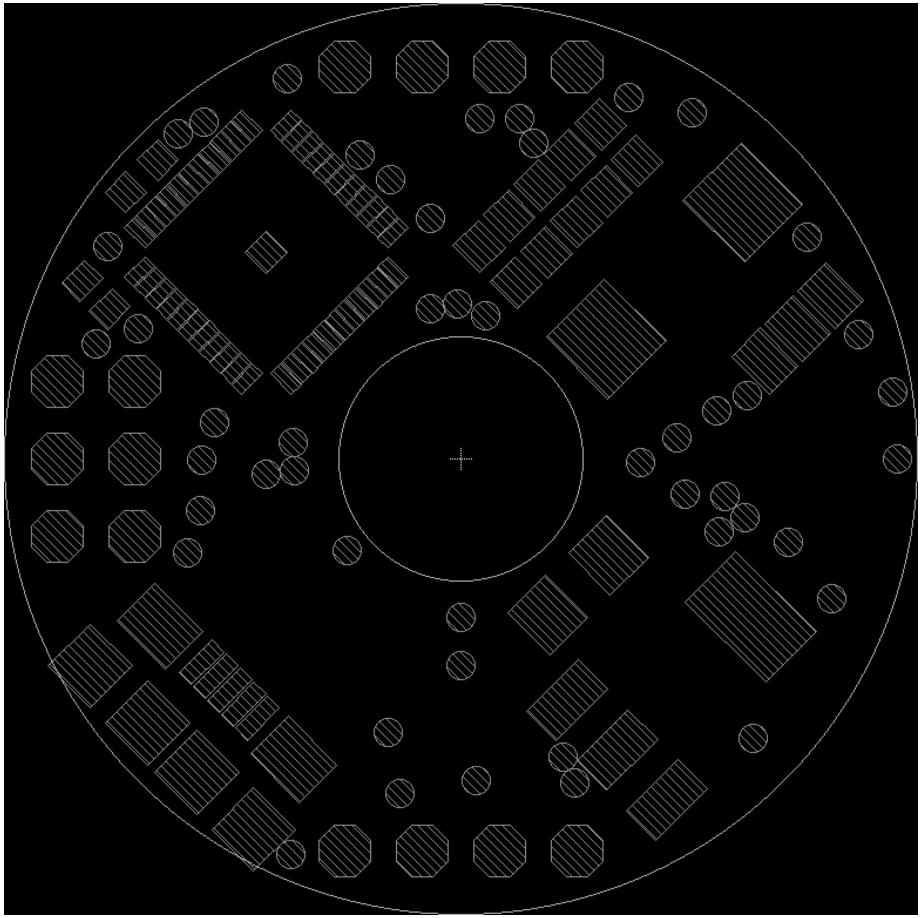
- Middle power plane



Top and bottom stop layers

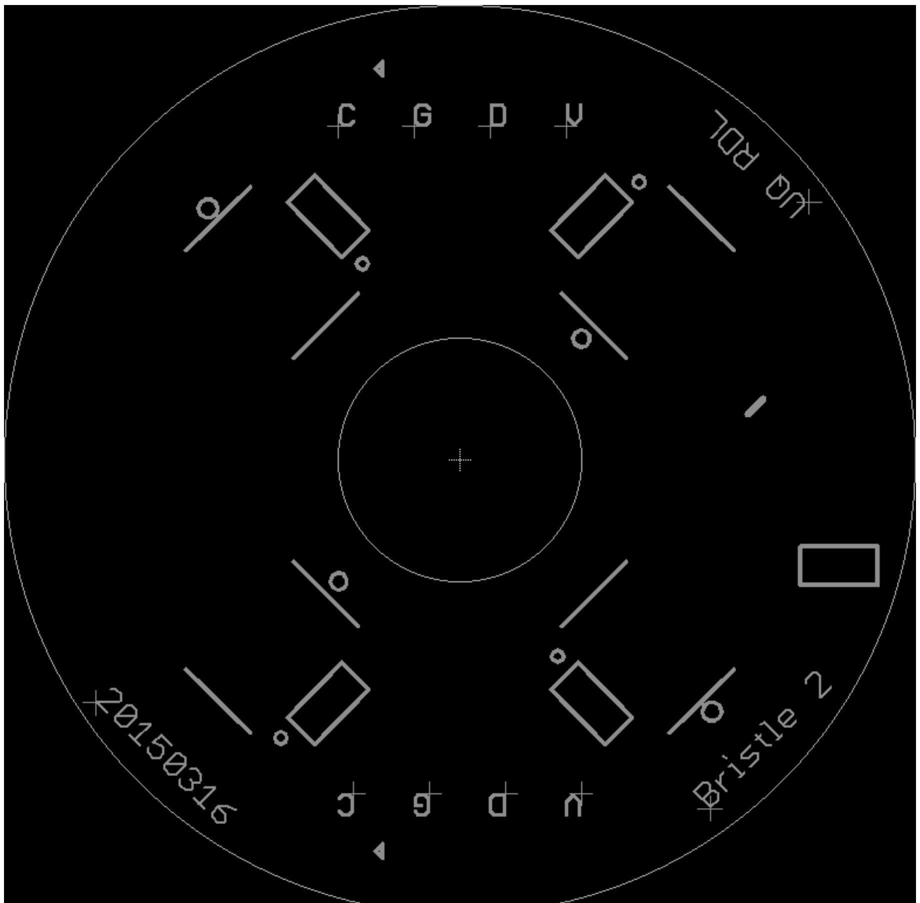
- Soldermask apertures – places without mask

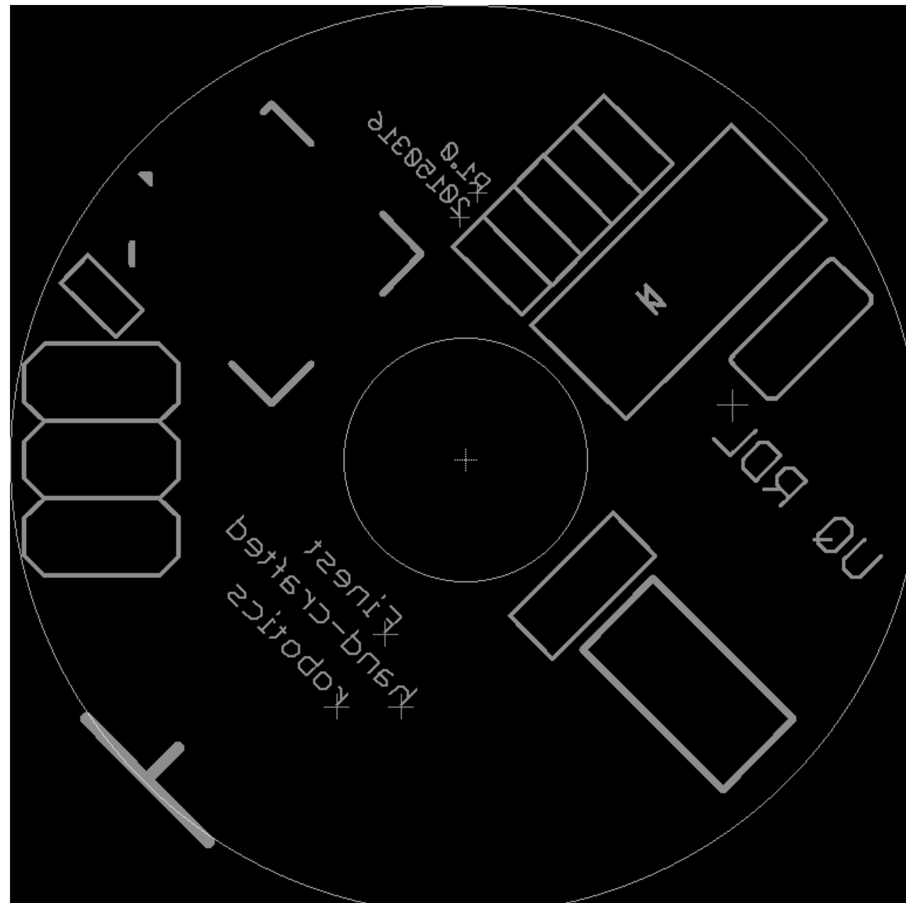




Top and bottom silkscreens

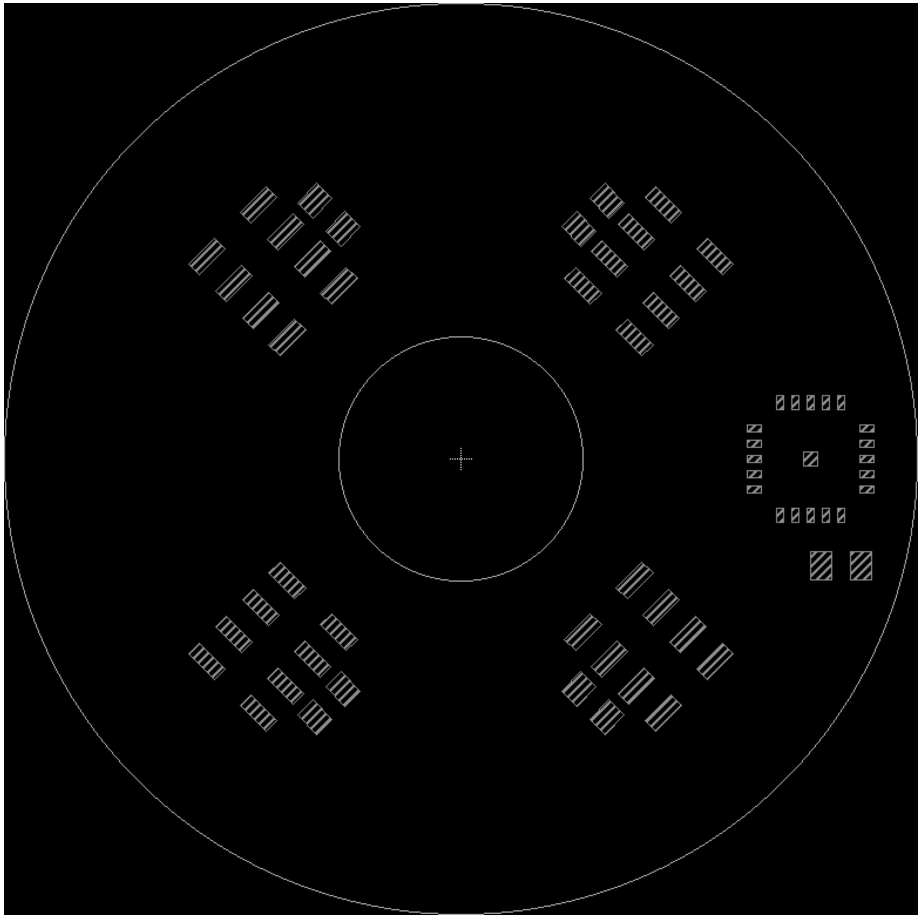
- Text and annotations (reversed on bottom)

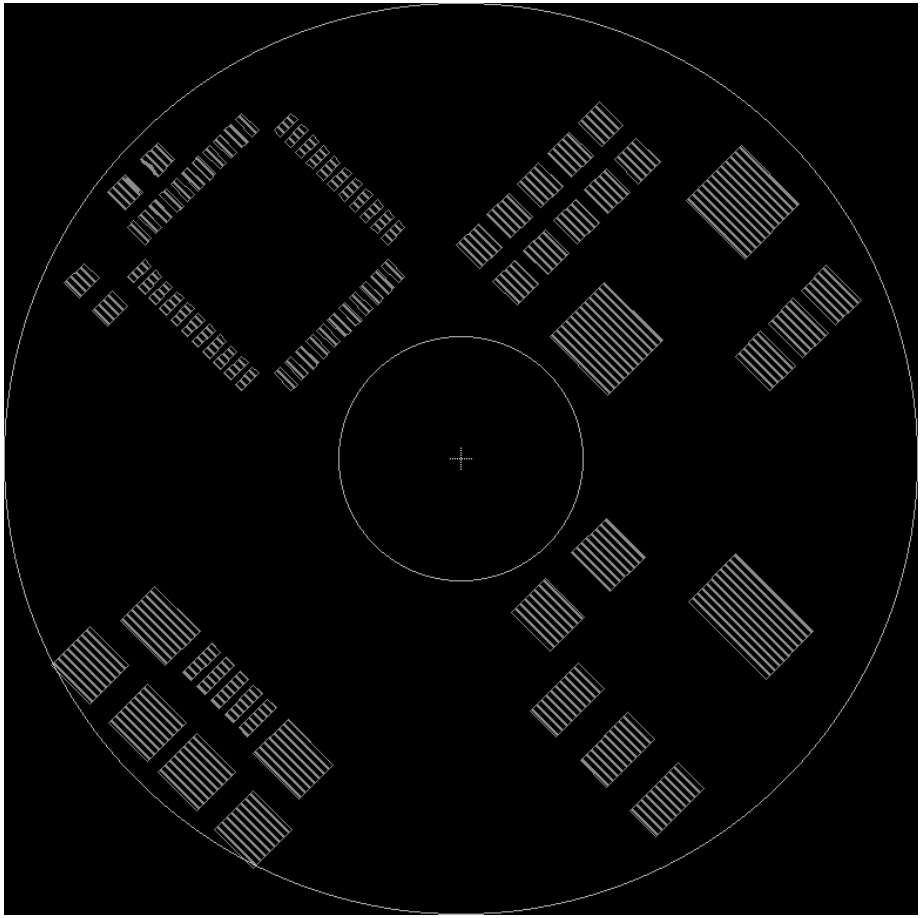




Cream layers

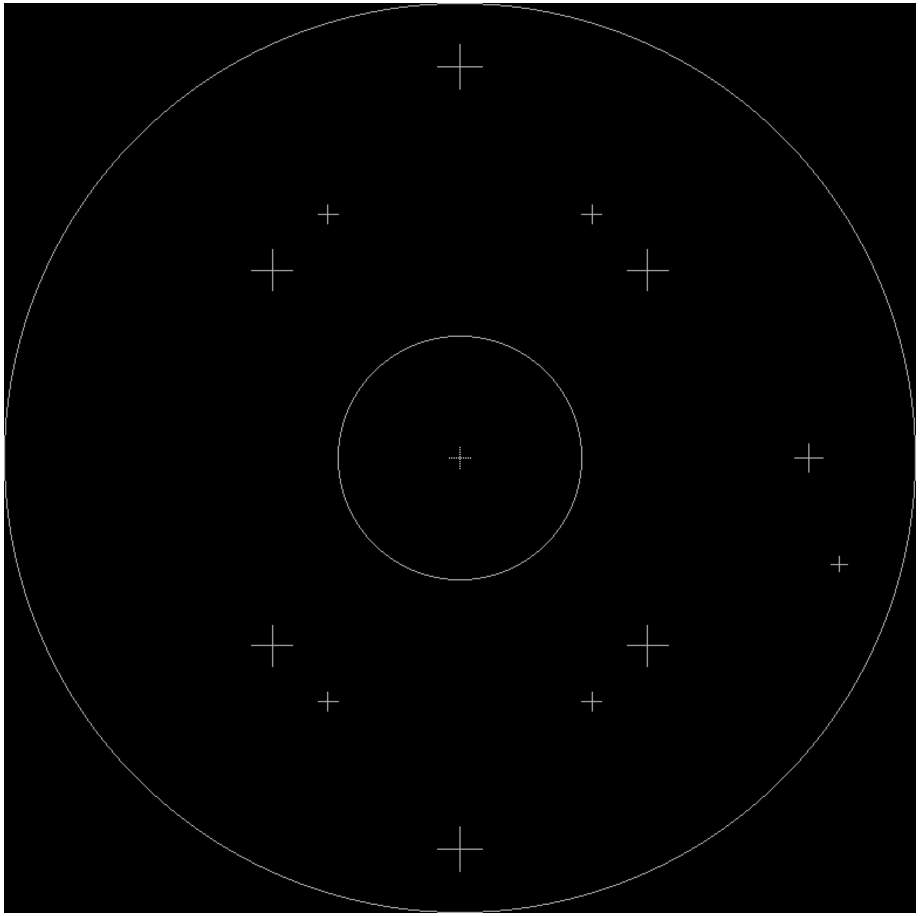
- All the places solder is deposited for SMD

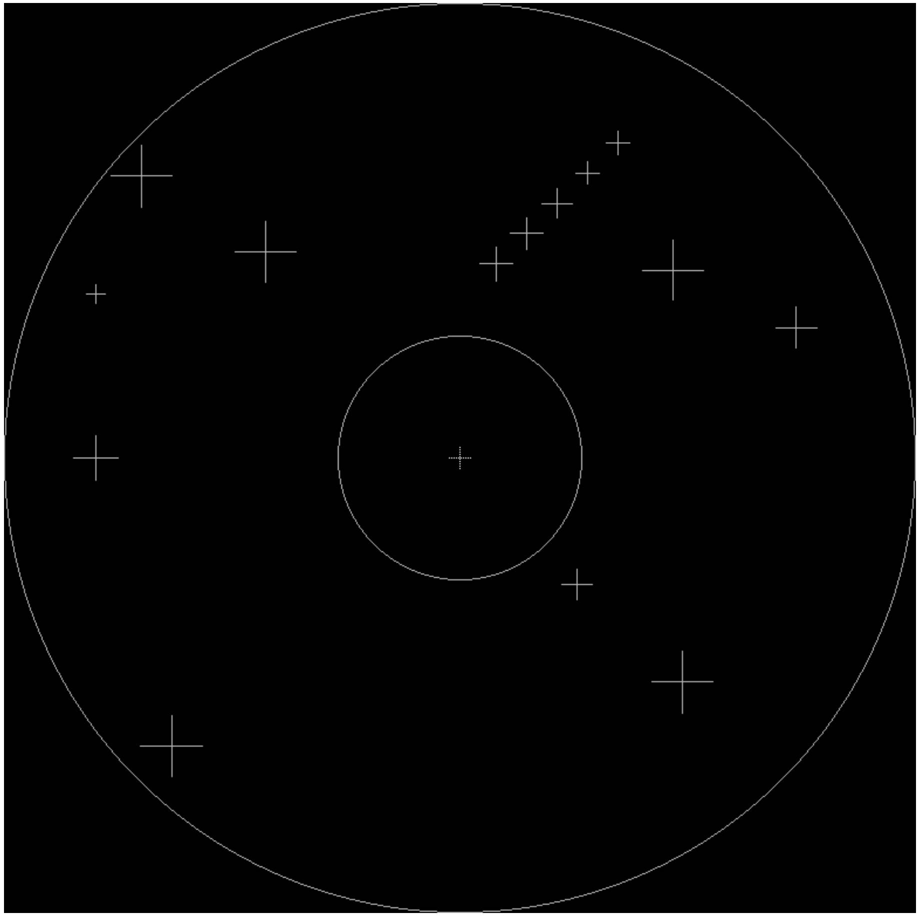




Component origins

- Used for automatic board pick-and-place





Bill of materials

Partlist exported from K:/work/UQ/development/projects/bristle/circuit/bristle2.sch at 19/04/2016 1:49:57 AM

Assembly variant:

Part	Value	Device	Package	Description
01	MPL115	MPL115	MPL115	
02	MPL115	MPL115	MPL115	
03	MPL115	MPL115	MPL115	
04	MPL115	MPL115	MPL115	
3.3V	VREGAP1117	VREGAP1117	SOT223	
C01		CAP_CERAMIC0402	0402	Ceramic Capacitors
C02		CAP_CERAMIC0402	0402	Ceramic Capacitors
C03		CAP_CERAMIC0402	0402	Ceramic Capacitors
C04		CAP_CERAMIC0402	0402	Ceramic Capacitors
C06		CAP_CERAMIC0402	0402	Ceramic Capacitors
C10	1uF	CAP_CERAMIC0603	0603	Ceramic Capacitors
C11	.1uF	CAP_CERAMIC0603	0603	Ceramic Capacitors
C12	.01uF	CAP_CERAMIC0603	0603	Ceramic Capacitors
C52	470uF	CAP_TANTALUMD/7343_REFLOW	EIA7343-31/D-R	Tantalum Capacitors
C53	10uF	CAP_CERAMIC1206	1206	Ceramic Capacitors
CN1		USBMICROB	USB-MICROB	USB Connectors
JP1	4X1_PIN_HEADERPTH	4X1_PIN_HEADERPTH	PIN_HEADER_4_PTH	
JP2	4X1_PIN_HEADERPTH	4X1_PIN_HEADERPTH	PIN_HEADER_4_PTH	
JP3		PINHD-2X3	2X03	PIN HEADER
LED2	CHIPLED_0603S	CHIPLED_0603S	CHIPLED_0603	
R1	120o	RESISTOR0402	0402	Resistors
R2	4.7K	RESISTOR0603	0603	Resistors
R3	4.7K	RESISTOR0603	0603	Resistors
SJ2		SJ2W	SJ_2	SMD solder JUMPER
UC01	ATTINY24QFN	ATTINY24QFN	QFN20	
XM1		ATXMEGAXXA4U-MH	QFN-44	ATXMEGA A4U Series (XMEGA+USB)

A whole bunch of stuff…

Lots of technical specification in a PCB!

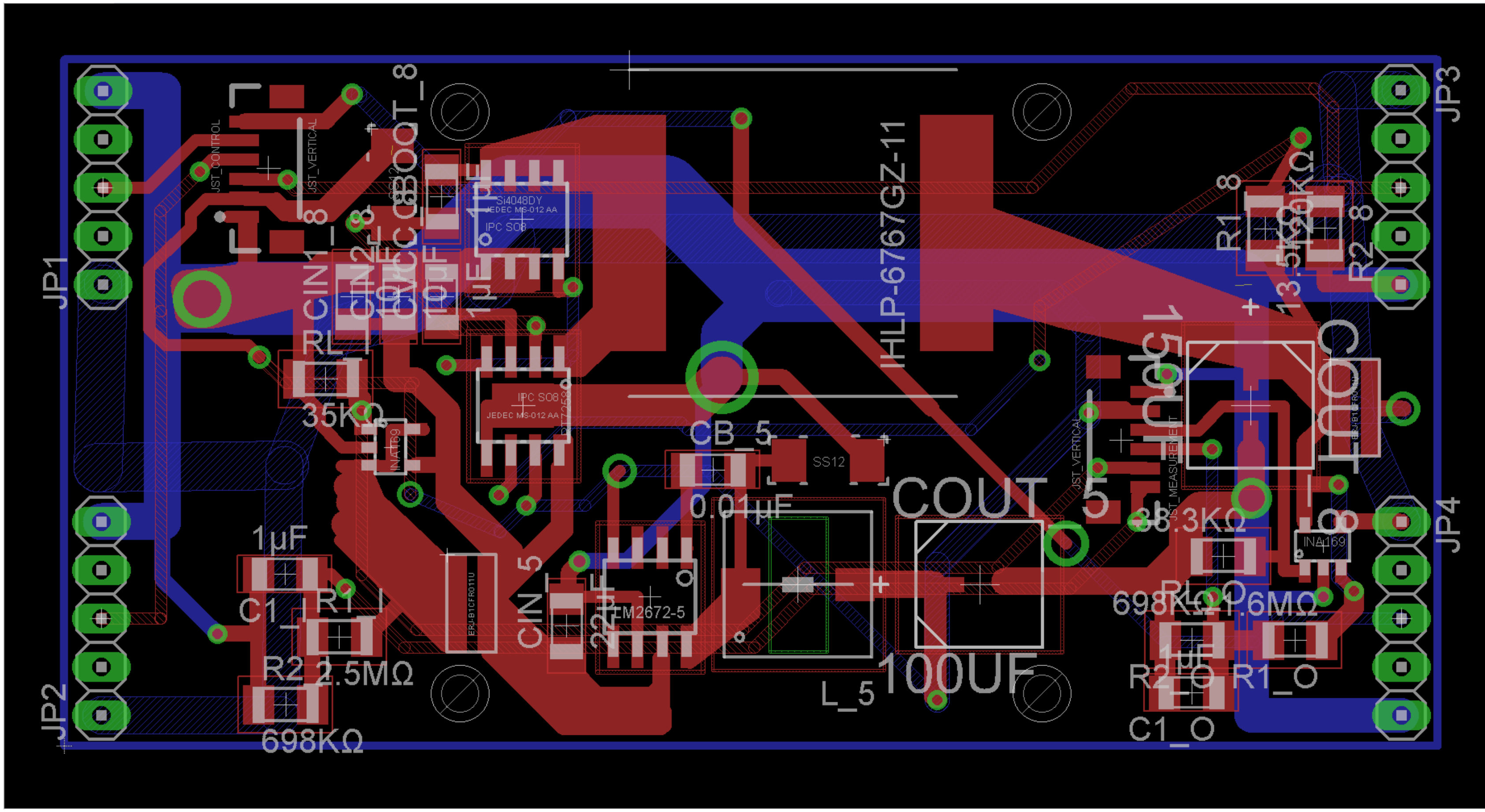
And if any one part of it is wrong, your board probably won't work.

Hey, that's great…

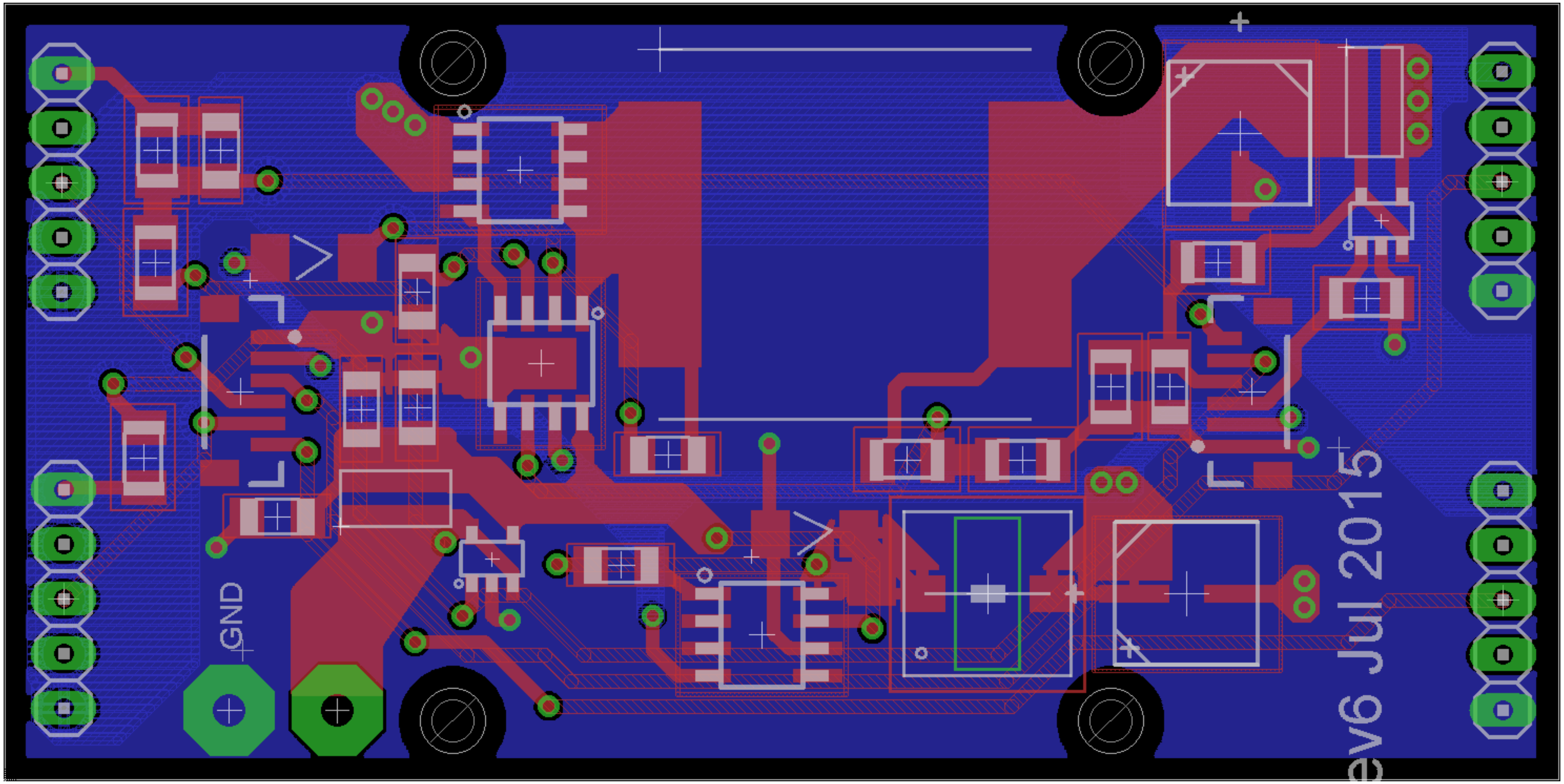
So what about actually designing a PCB?

How hard can it be?

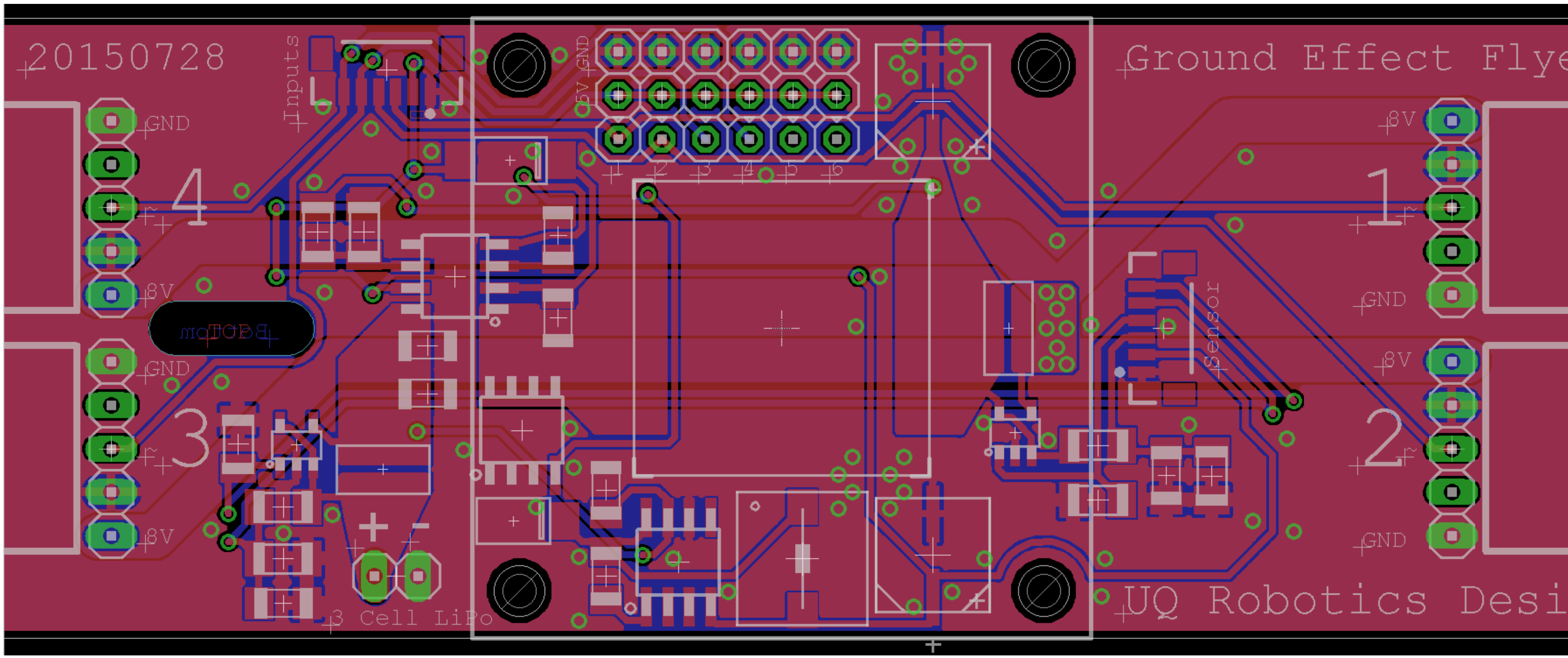
Absolute beginner's layout

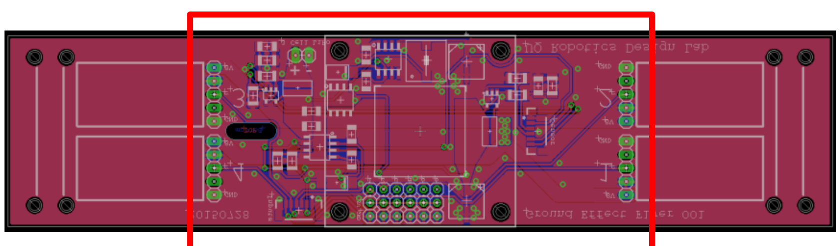


Same designer with more experience



Experienced designer layout





Routing PCBs

- Laying out PCB is broadly called "routing"
  - Usually the hardest part of PCB design
  - Tight routing = less board space, more boards per panel, lower costs: take time to get it right!

- Finding optimal routing is a challenging task and machines are not very good at it

<u>DO NOT WASTE YOUR TIME</u>

<u>OR MONEY WITH AUTO-ROUTING</u>

Routing PCBs

- If you become very good at routing you will win the acclaim and respect of your peers!
- If you are lazy or terrible at it, you will be silently (or not so silently) cursed by everyone who has to deal with your designs
- Pretty much any idiot can route a low-speed board (<1 MHz) and expect it to work…

The performance gotcha

- But! There is no mercy in high-speed digital, low-voltage analog or RF *anything!*
  - *Mixed circuits?? Fetch the Necronomicon…*

- Obey all the rules or physics will punish you

Most boards you'll be making anytime soon will be pretty forgiving

Worth bearing in mind

- In general, the PCB and circuit design should be developed in parallel
  - Can be quite easy to design impossible circuits
- If the circuit just won't work on the board as designed… try tweaking the circuit?
  - A small circuit design can drastically simplify the routing problem
  - You'd be surprised how often this works!

General routing process

Here is a set of guides for PCB layout

Do not adhere to them mindlessly

Not an excuse to stop thinking…

General routing process

1. Specify your board dimensions (if known)
2. Place constrained components first
   – Usually things like large parts and connectors
3. Route power, ground and major signal busses
   – High power, high frequency, high sensitivity
4. Shuffle components to reduce net complexity
5. Make connections between components
   – Revise layout on the fly

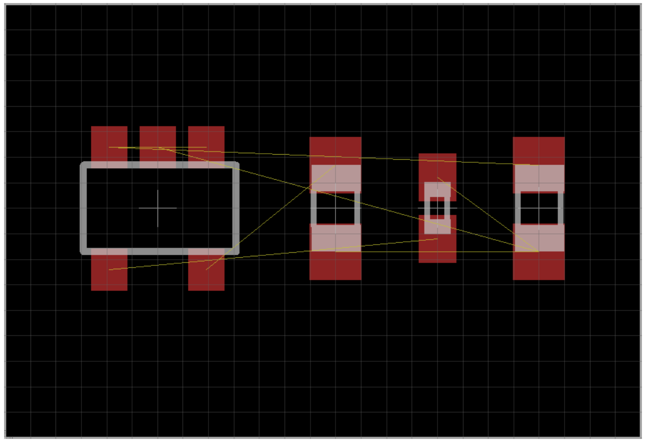
Guiding principles

- Minimise the number and variety of drills
- Maximise track width and spacing
- Preserve ground layers
- Preserve spacing between components
- Maximise clearance with the board edges
- Minimise turns and maximise corner angles

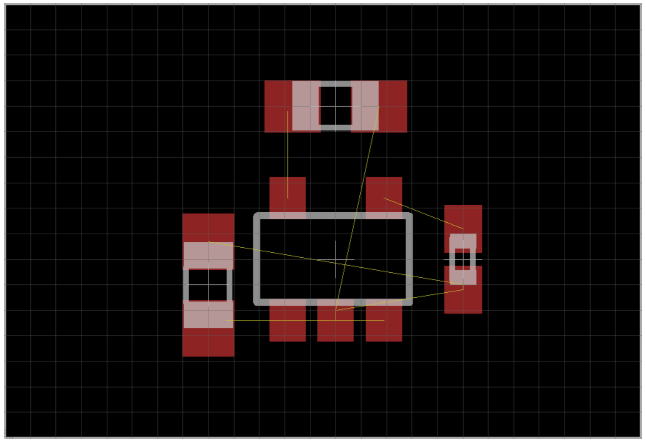
If it looks crap, it probably is crap. If it looks like somebody loved it, it's probably ok

Part placement

- Good part placement makes your life simple
- If you can see it all laid out, you can route it!
- Follow the reference design if available
  - Depart from the reference design at your peril!



Default part layout – not helpful!



Oh hey, that's where that goes…

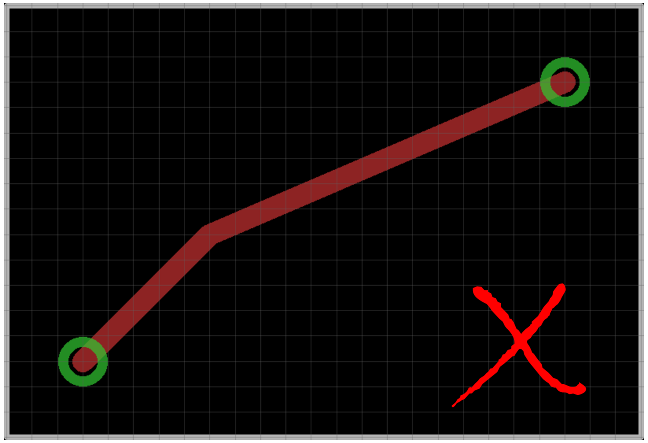
Routing tracks

- Routing tracks refers specifically to laying out the individual traces of a PCB
- Most people do a mediocre job without trying too hard… some people make a mess

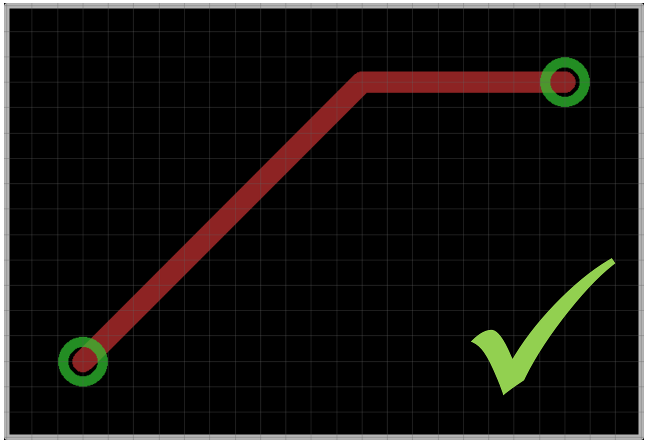
A few simple rules will get you 90% of the way there…

Routing tracks

- Adhere to the 45° routing rule
  - Why? Partly due to tradition, and partly because it makes for neater-looking circuits
  - If the track or space constraints require it, though, it's ok to route direct point-to-point



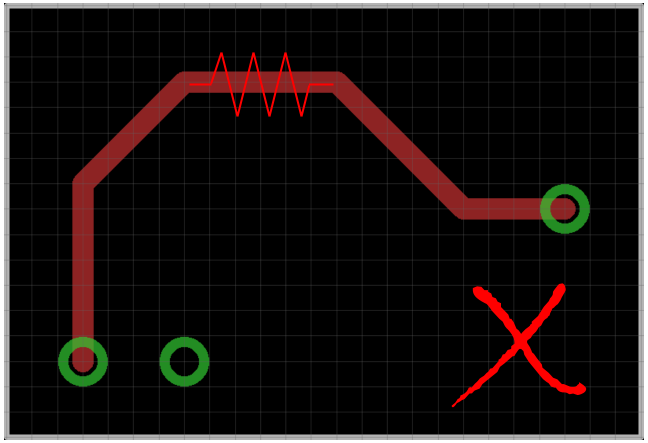
Looks sloppy and lazy



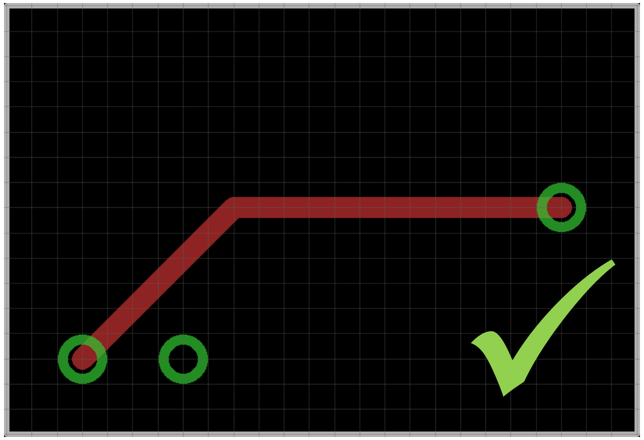
Looks like it's meant to be this way

Routing tracks

- All else being equal, take the shortest route
  - Reduces track resistance, capacitance etc.
  - Classic sign of someone who wasn't paying attention to what they were doing



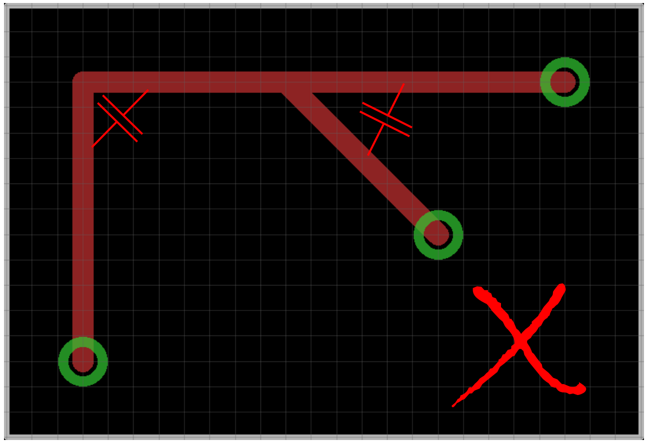
Circuitous circuit



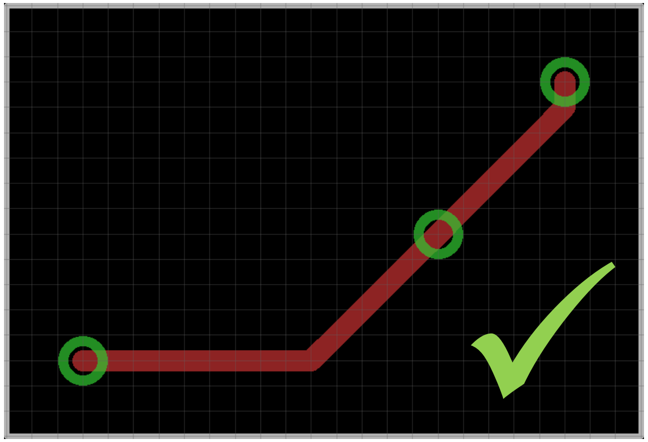
Short track better

Routing tracks

- Avoid acute and right angles; minimise the number of turns and kinks where possible
  - Adds extra capacitance in the path
  - Can be a big problem in high-speed circuits



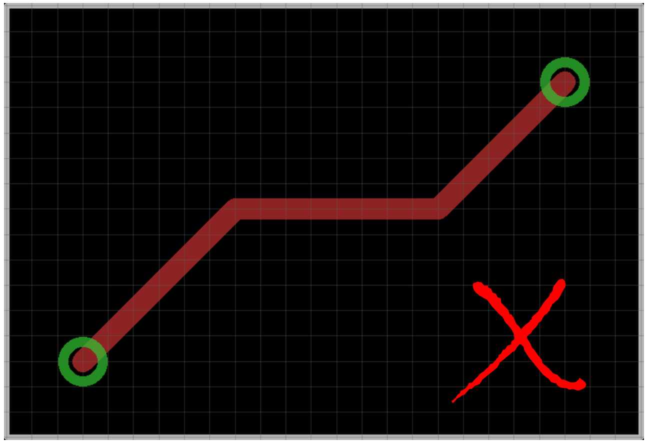
Right angle and acute corners



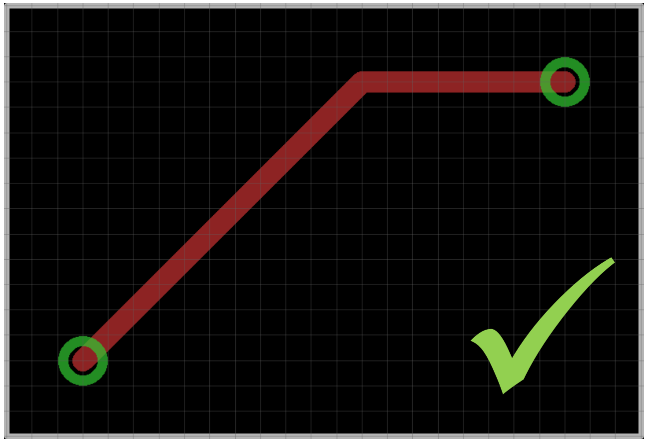
Obtuse corners best

Routing tracks

- Keep an eye out for the ‘dogs leg’
  - Easy to fix…
  - Just take the time to be thorough



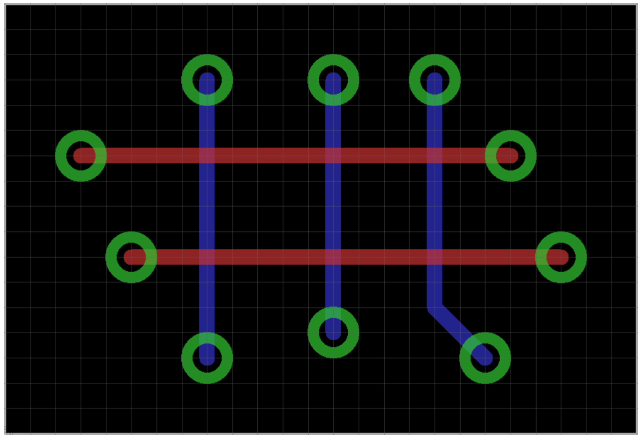
Dog leg track



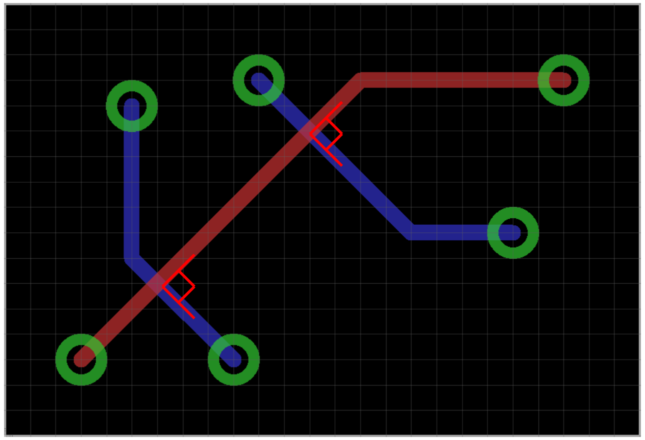
Single kink track

Routing tracks

- Useful to route left-right, up-down on different layers
  - Try to make your digital lines cross at 90 deg on high-speed multilayer boards



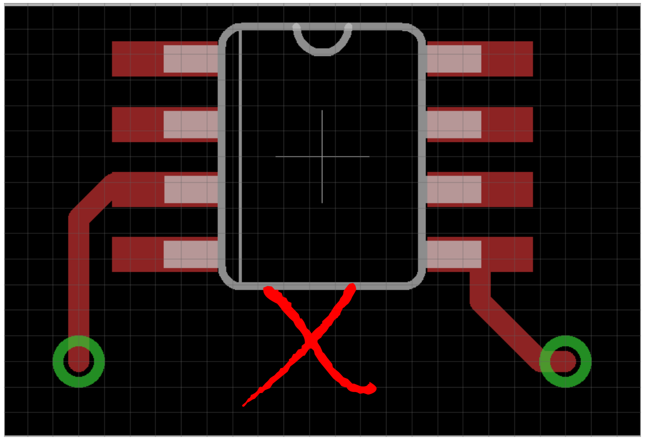
Different layer track directions



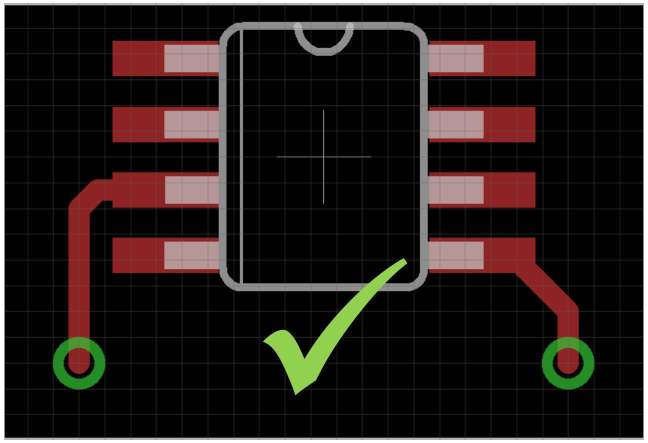
90^o high-freq crossings

Routing tracks

- Enter surface mount pads cleanly
  - Right angles on edges or 45^{o} through corners
  - Avoid acute break-outs from pad edges
  - Can’t always be achieved in practice, though…



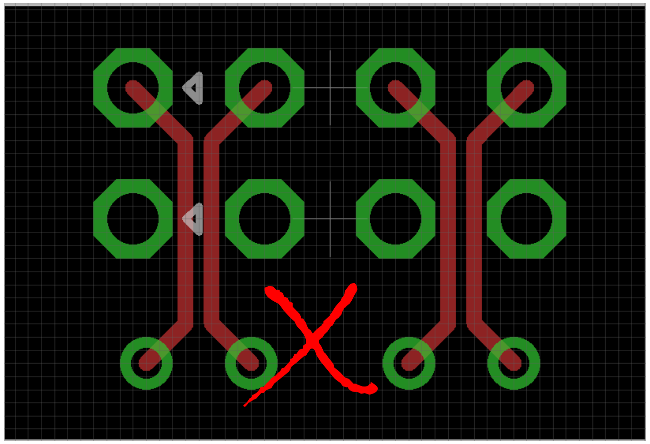
Crappy SMD pad exists



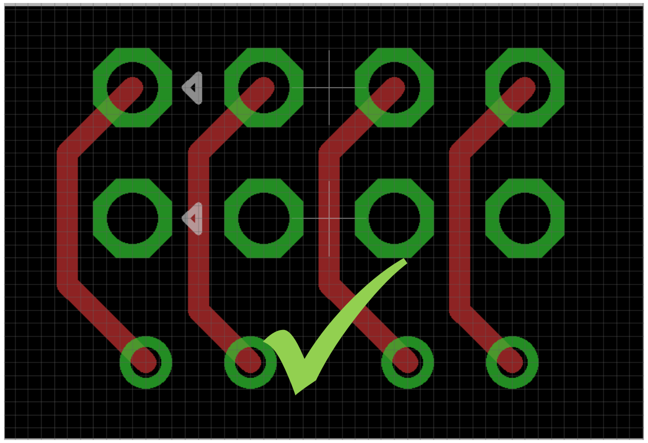
Nice SMD pad exists

Routing tracks

- Don't try to cram stuff through too small a gap… go around if at all possible
  - Maybe you need to rethink your overall part layout or structure?



Avoid cramped buses



Clearance is your friend

Routing tracks

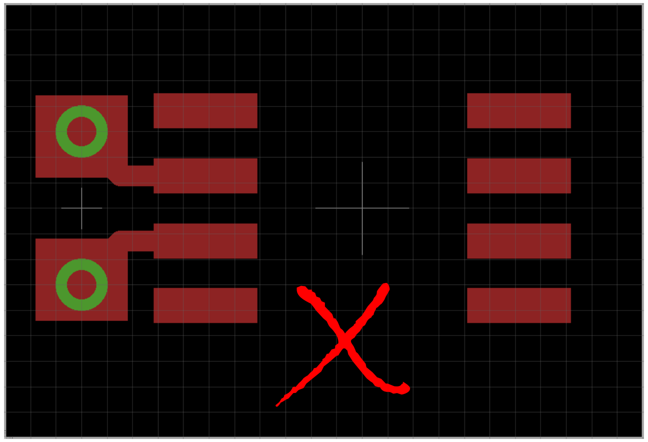
- Higher currents need bigger tracks
  - Lots of online calculators for track size
- Track width rules of thumb for 35 μm
  - Small signal: <0.2 mm
  - 1 A: 0.4 mm external layer (1.0 mm internal)
  - 2 A: 1.0 mm external layer (2.5 mm internal)
  - 5 A: 2.5 mm external layer (9.0 mm internal)
- 40 A is the max current a 70 μm board typically can achieve (without bus bars)

Pads and vias

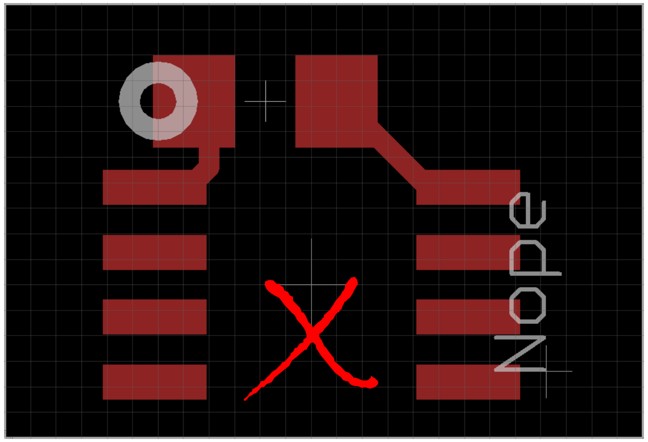
- Drilling is the most costly fabrication step
  - Reduce the total number and variety of drills
- Skinny drills break!
  - Max depth:width ratio no more than 10:1
- Give pads sufficient annular rings to stop pull-through
  - Aim for no less than ~ 25% of hole diameter

Pads and vias

- Don’t put vias in pads (usually)
  - Solder will wick down the hole => dry joints
  - Vias at corners sometimes used for heat sinking
- Don’t silkscreen over pads



Avoid vias in pads



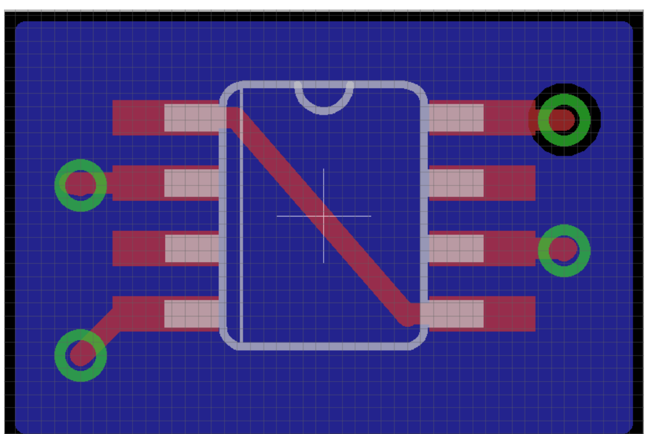
Watch where the silkscreen goes

Pours and ground layers

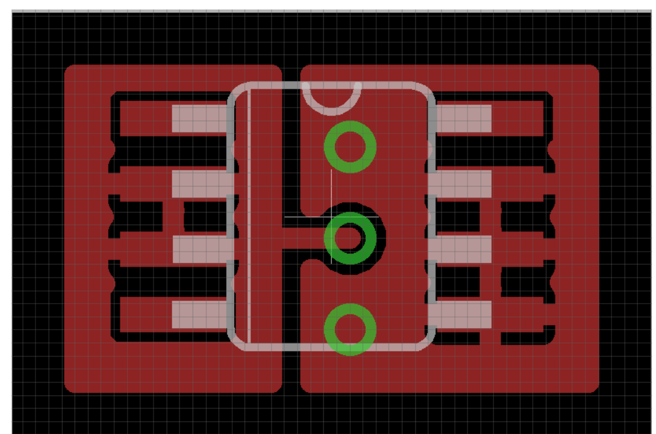
- Pours are large, contiguous copper areas, typically used for ground and power
- So many advantages!
  - Greatly simplify the routing problem
  - Good for buffering power draw and adding thermal relief
  - Ground layer capacitance can help quieten or isolate noisy parts of a circuit

Pours and ground layers

- Very common configurations:
  - Ground pours on both top and bottom layers
  - Power and ground on alternating layers
  - Different signal values pours in different parts of the circuit



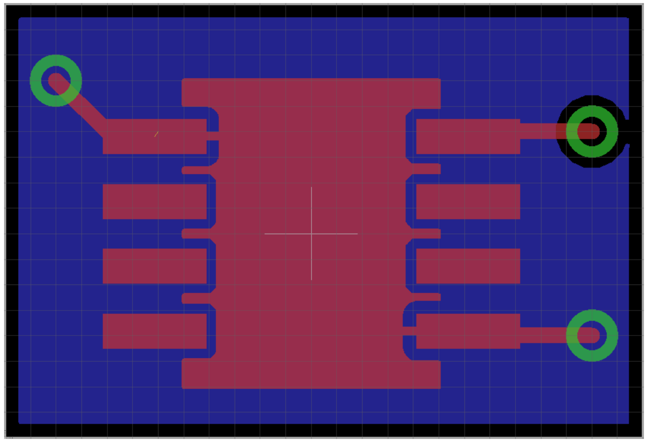
Easy routing of backplane signals



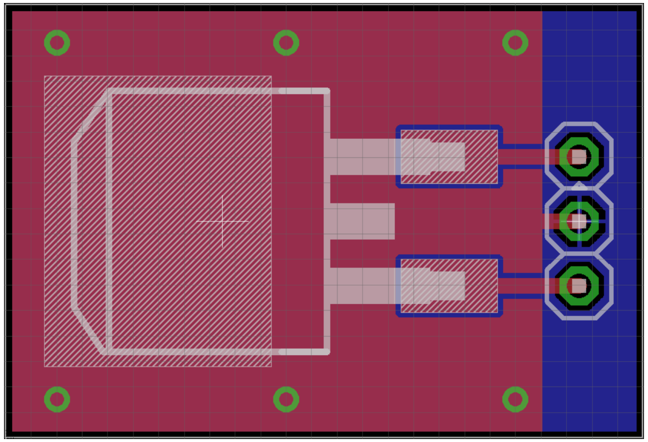
Two different pour nets

Pours and ground layers

- Ground planes are very common under microprocessors
  - Sometimes required and strictly specified!
- Also very common for power supply thermal and noise management



IC mounted over ground plane

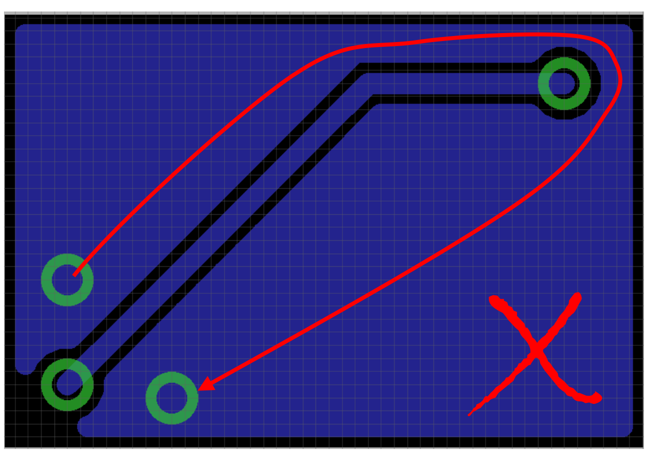


Regulator with thermal pads

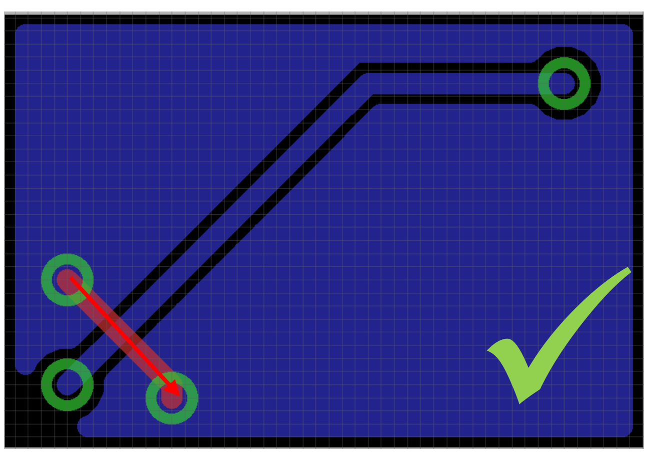
Pours and ground layers

Things to look out for:

- Big areas of copper absorb heat, so use thermal reliefs to make soldering easier
- Add cross-flow "bridges" to prevent large circuitous detours in power/ground planes



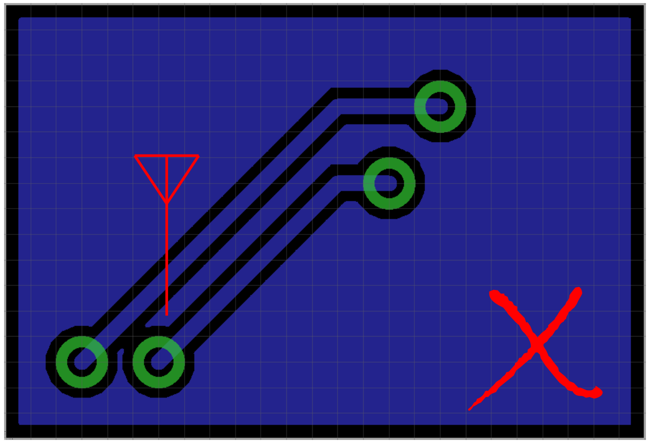
Long current detour "ground loop"



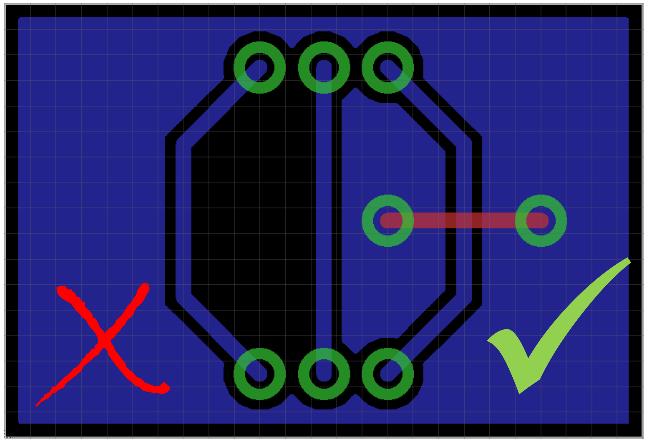
Shortcut 'bridge'

Pours and ground layers

- Avoid dead copper and stems
  - Stems act like RF antennas and inject noise into your signal – right on the ground plane!
- Bridge island fills if you can



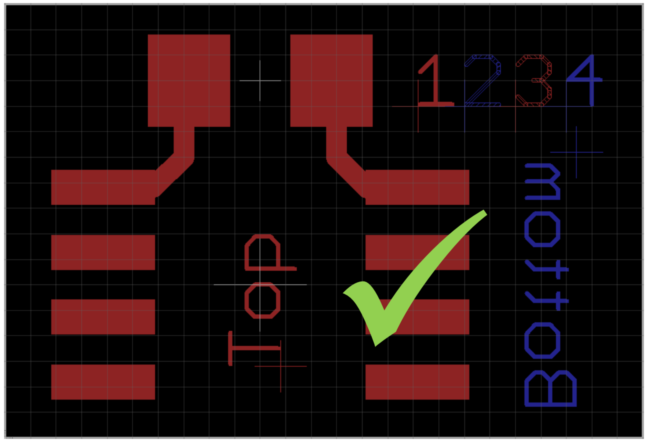
Avoid pour antennas

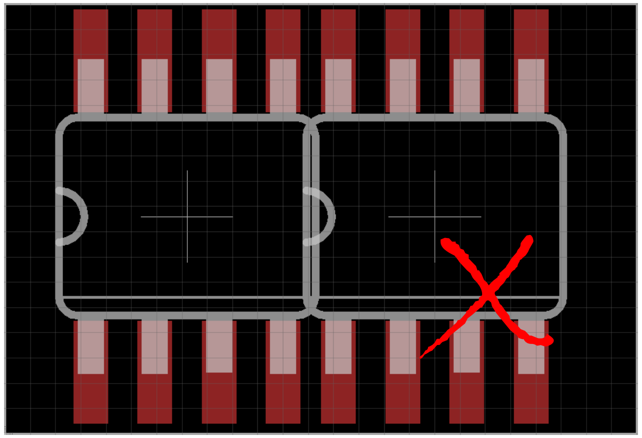


Island fills, with and without bridge

Other copper layer-y things

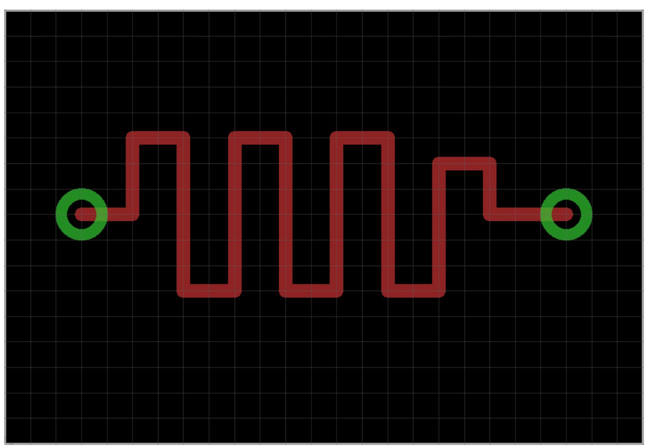
- Include top/bottom and numbered layer callouts when submitting separate layer files
- If the fab tech is unsure, they won't ask
  - They'll just give you "something"
  - Or worse... exactly what you asked for!



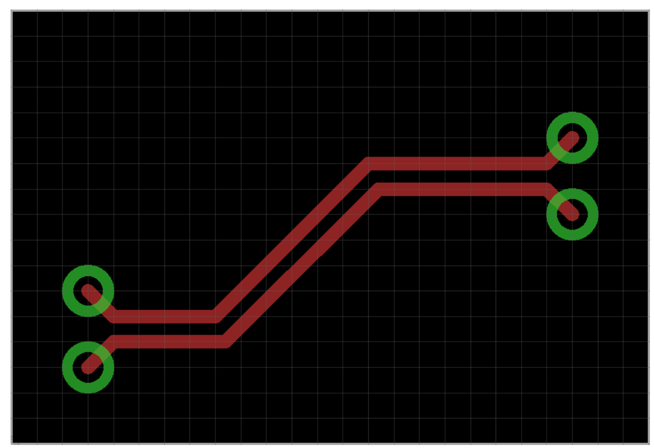


Other copper layer-y things

- Some high-performance circuits put strong demands on track length and impedance
  - Tricks like differential signal length matching, inductance wriggles… advanced stuff!



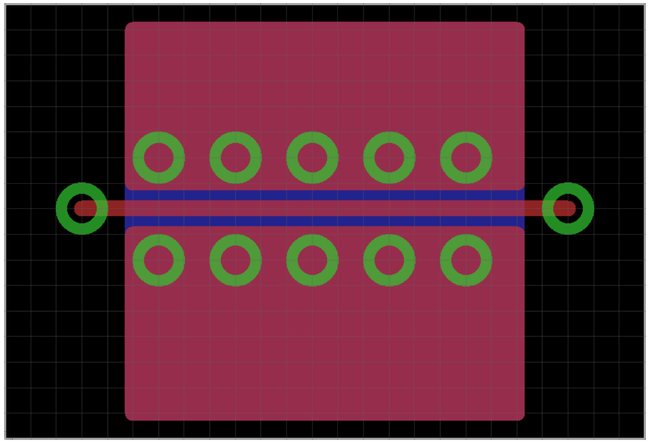
Inductance ‘wriggle’



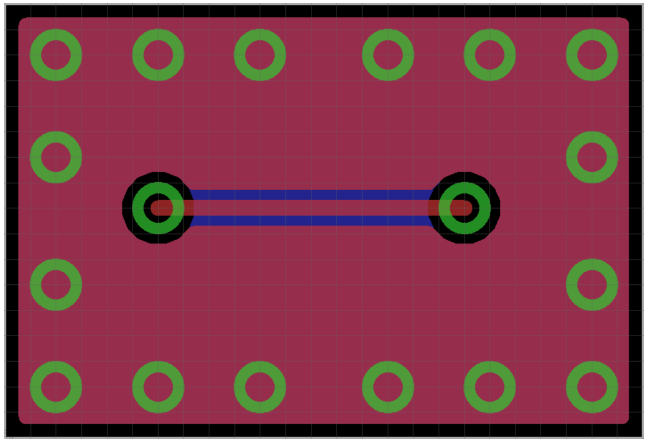
Matched-length differential tracks

Other copper layer-y things

- RF stitching is a thing: string of vias around ground fill and traces
  - Create waveguides; avoids HF ground loops
- Guard trace around periphery of the board to protect against EMI



Stitched RF wave guide



EMI guard loop

Specifying board dimensions

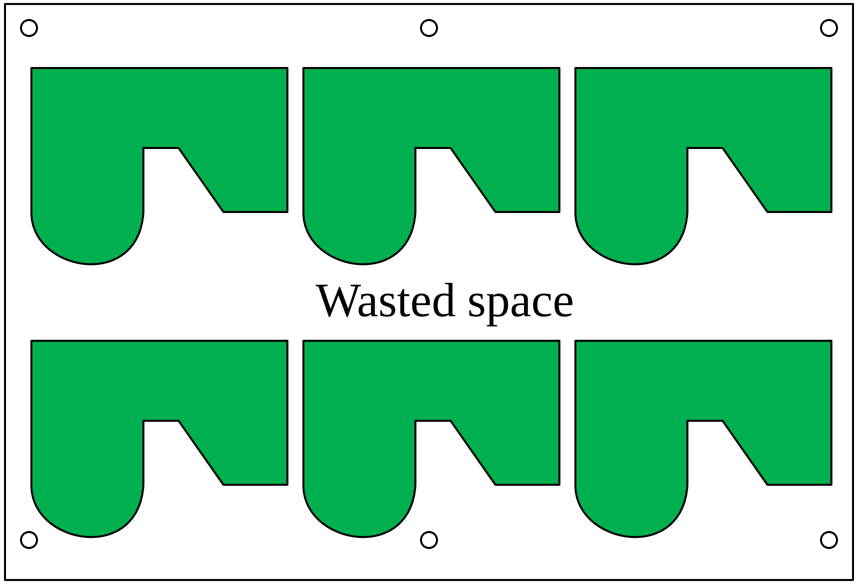
- How big should your board be, anyway?

As small as possible!

- Less copper, shorter signal traces, less parasitic resistance/capacitance/inductance, etc, and thus better performance
- Boards smaller than 10 x 10 mm or larger than 610 x 450 mm require specialist fabs
  - Largest I've ever seen of was 1000 mm long!
  - Also, large/thin boards sag under their weight

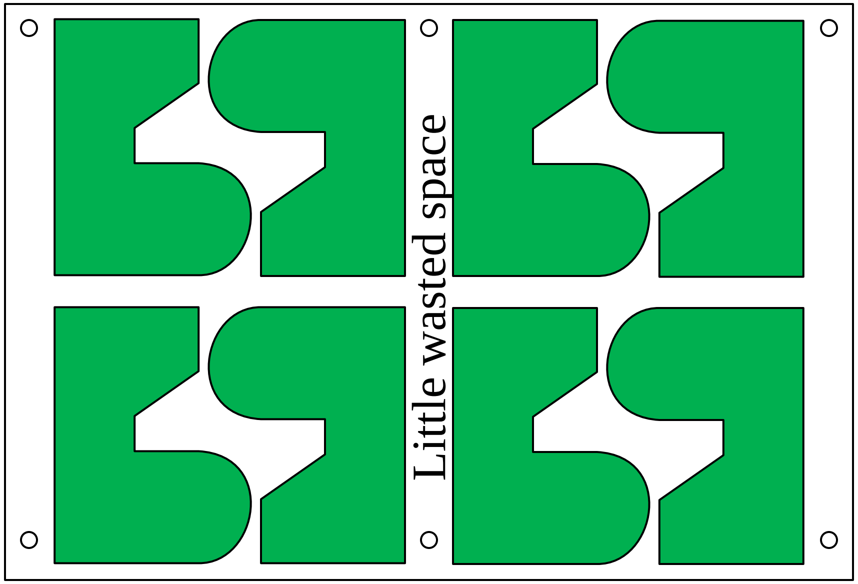
Panelisation

- Awkward outlines can make panelisation difficult and leads to wasted panel space
  - Plan for panelisation when choosing an outline



Panelisation

- Careful board design, clever panelisation and panel sharing can dramatically increase yield and reduce costs per board

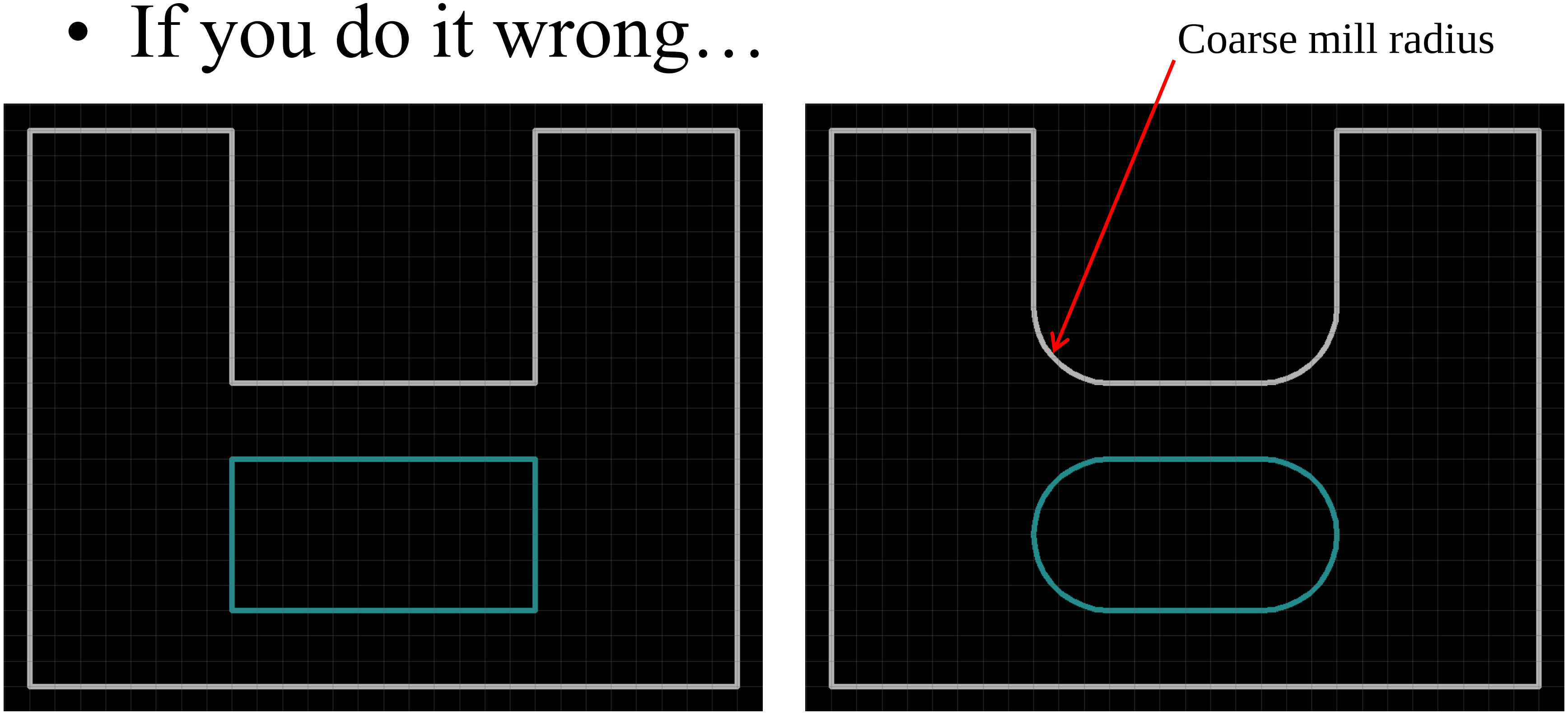


Milling

- Individual circuit boards are separated from the panel by milling
  - Milling is also used to produce slots and cutouts
- The smallest milling bit is typically about 0.2 mm diameter
  - Don't expect perfect 90^o corners
  - If you specify a realistic radius, you might get it
  - If you specify a right angle, you'll get whatever they feel like giving you (probably 1 mm)

Milling

- If you do it wrong…



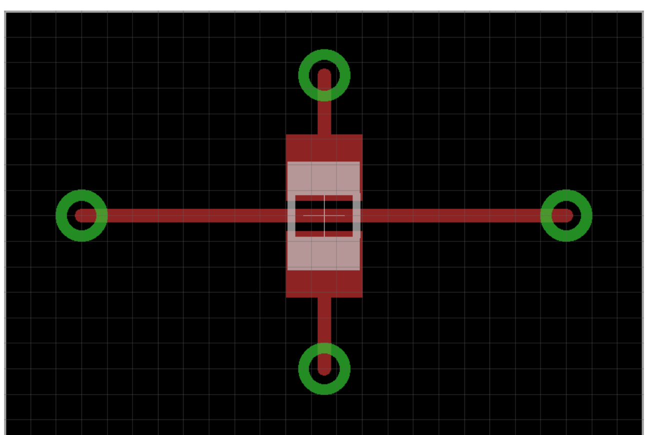
What you ask for (unrealistic)

What you will actually get

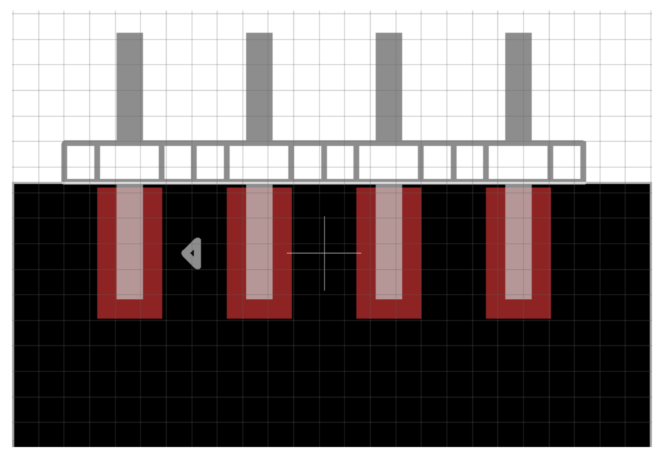
Go for the biggest mill you can get away with

Cute tricks

- 0 Ohm resister bridge
  - Super compact way of adding a jumper
- Side-mount a pin header for a low-profile, low board footprint edge connector



0 Ohm jumper



Pin header connector on edge

Some comments on schematics

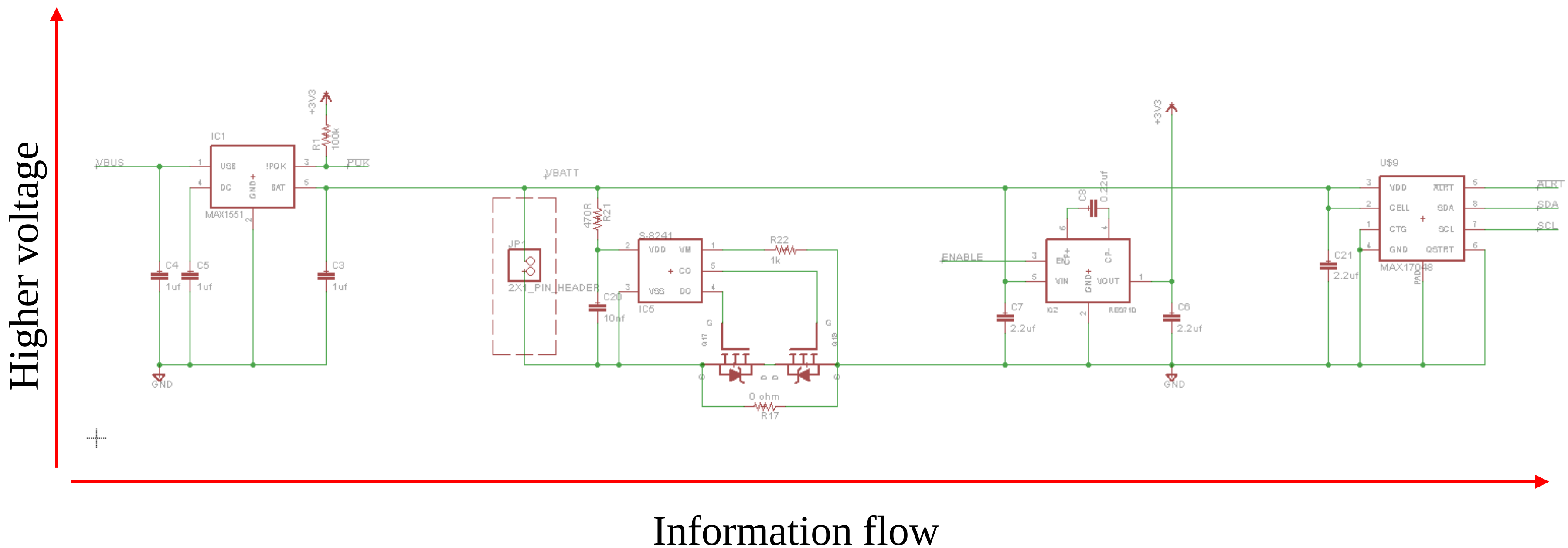
Dishonourable mention:

- Terrible layout of circuit diagrams and schematics is almost as much trouble as poorly laid out boards

- Your circuit schematic and its associated board are tightly coupled
  – One should complement the other

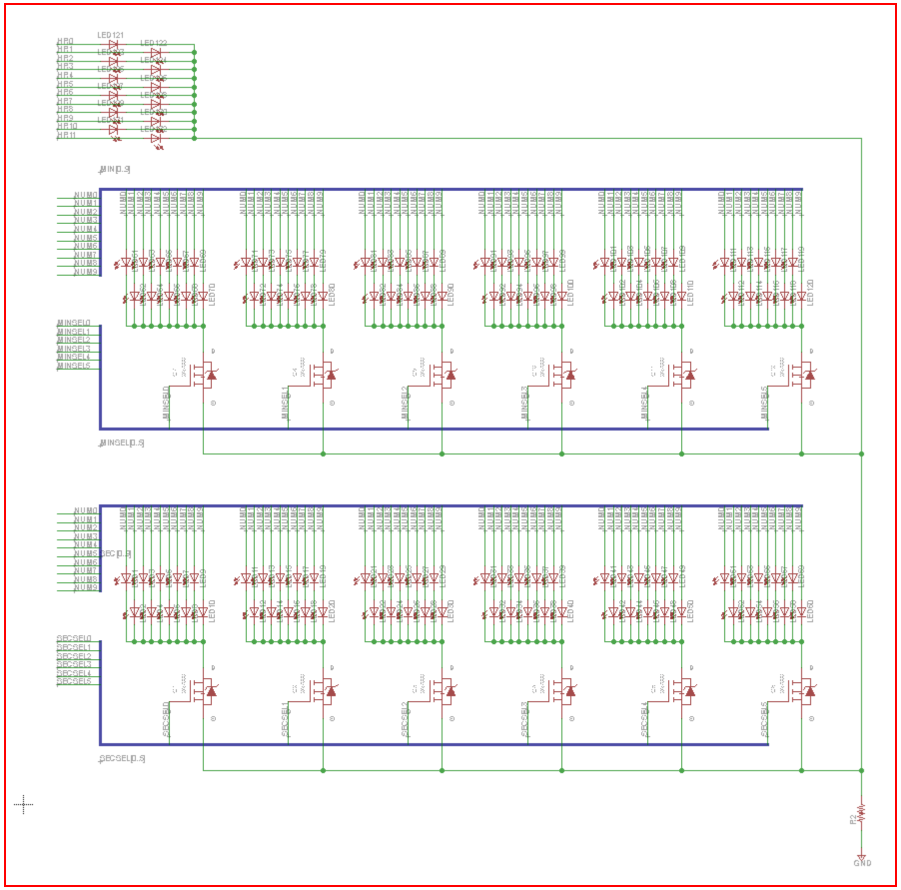
Some comments on schematics

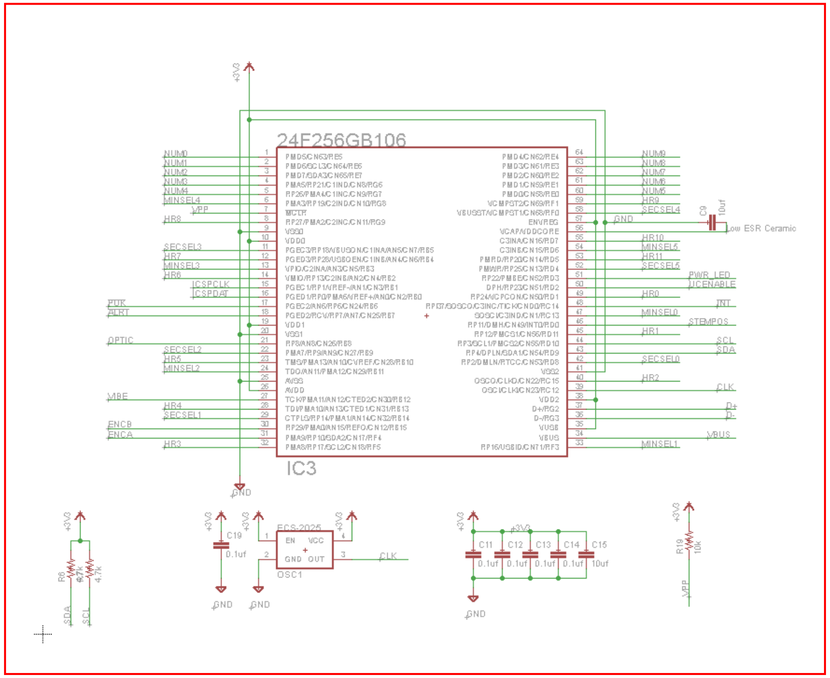
- Logical flow left-right
- Voltages up-down



Some comments on schematics

- Split into modules; divide by sheet

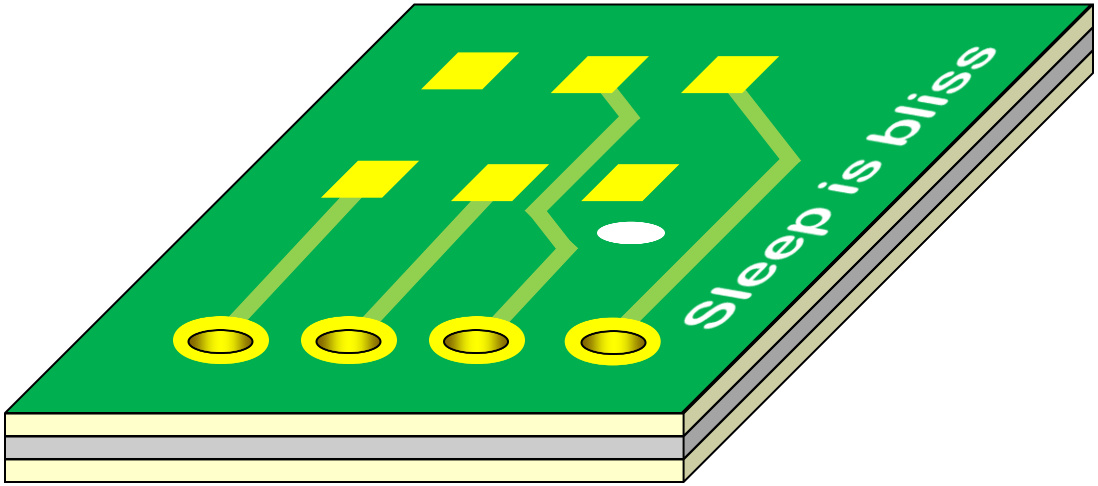




Tip of the iceberg

- Again… PCB design is one of those things you can spend a career perfecting
- I’ve made scores of PCBs and I *still* find new and innovative ways of screwing it up!
- Open up high quality products and look at the PCBs inside – you can *see* the love

Questions?



Tune-in next time for…

Questions and Answers Vol. 2

or

“We’ll get to Q&A Vol. 2 eventually... Right?”

Fun fact: 98% of all humans have parasites.