Introduction to Practical PCB Design

or

"Shadow of the Colossus."

2AM Edition!

P Pounds

12 March 2018

University of Queensland

But first…

Some house keeping

Calendar at a glance

You are here →

| Week	Dates	Lecture	Reviews	Demos	Assessment submissions
1	19/2 – 24/2	Introduction			
2	26/2 – 2/3	Principles of Mechatronic Systems design			Problem analysis
3	5/3 – 9/3	Professional Engineering Topics			
4	12/3 – 16/3	PCB design tips	Progress review 1		
5	19/3 – 23/3	Your soldering is (probably) terrible			
6	26/3 – 29/3	Introduction to firmware design			
Break	30/4 – 13/4				
7	16/4 – 20/4		Progress seminar	25% demo	
8	23/4 – 27/4				
9	30/4 – 4/5			50% demo	
10	8/5 – 11/5	No lecture	Progress review		
11	14/5 – 18/5			75% demo	Preliminary report
12	21/5 – 25/5				
13	28/5 – 1/6	Closing lecture		Final testing	Final report and reflection

3

Progress Seminars

- Currently underway… one day to go

- There has been PAIN, as promised
  - Show up prepared
  - Show up with evidence
  - Show up coherent

- Most groups have survived so far
  - But not everybody…

Problem Analysis (from 2018)

Results available on Blackboard after the ~~lecture~~

Once I upload them!

5
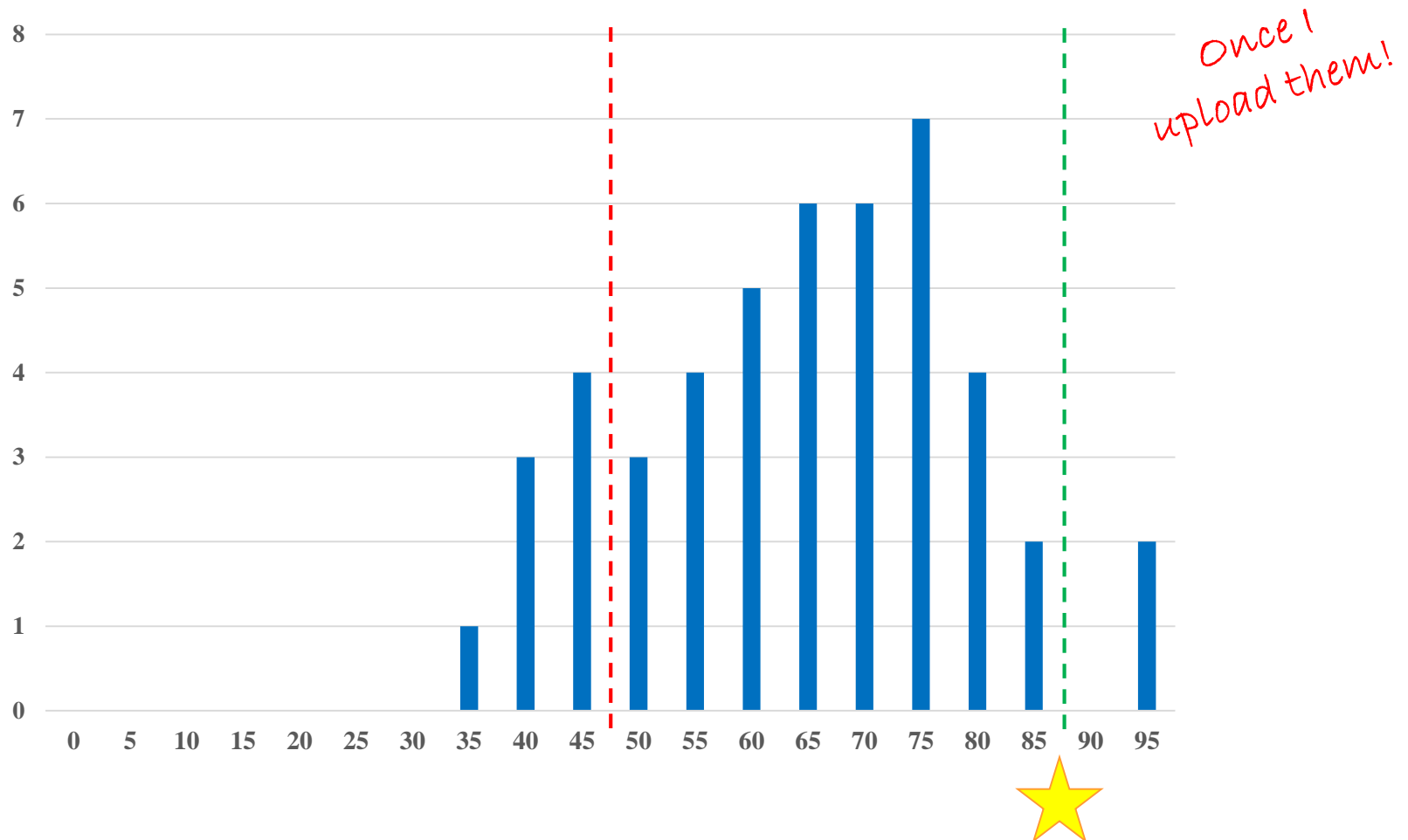
Lab access

- Some people still don't have lab access
  - Pro tip: If your swipe card doesn't work, you shouldn't be in there…

- If your card doesn't work, check with Jonathan Read: j.read2@uq.edu.au
  - He will let you know what induction tasks you have still outstanding

- Please note, I have no control over lab access, except to have it revoked

FAQ Roundup

- **How big will the placard text be?**
  - This will be outlined in the revised spec, which will go up *soon*. Example placards will be available shortly.

- **Can we attach the DSN module to the spacecraft?**
  - No. It has to be on the ground and be removable from your design.

- **Can we please have CAD files of the gimbal?**
  - Yes; they are up on Blackboard

- **When will the gimbal be ready?**
  - I have been promised three gimbals by this Friday.

- **Dude, where's my ETSG budget?**
  - It's up now – I promise!

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

Through-hole pads for pin headers

Inter-layer vias

Date and revision

SMD pads for surface mount components

Cutout

Cutout

White silkscreen printing

Power and ground pours

Top view

Green soldermask

Bottom view

Gold immersion finish
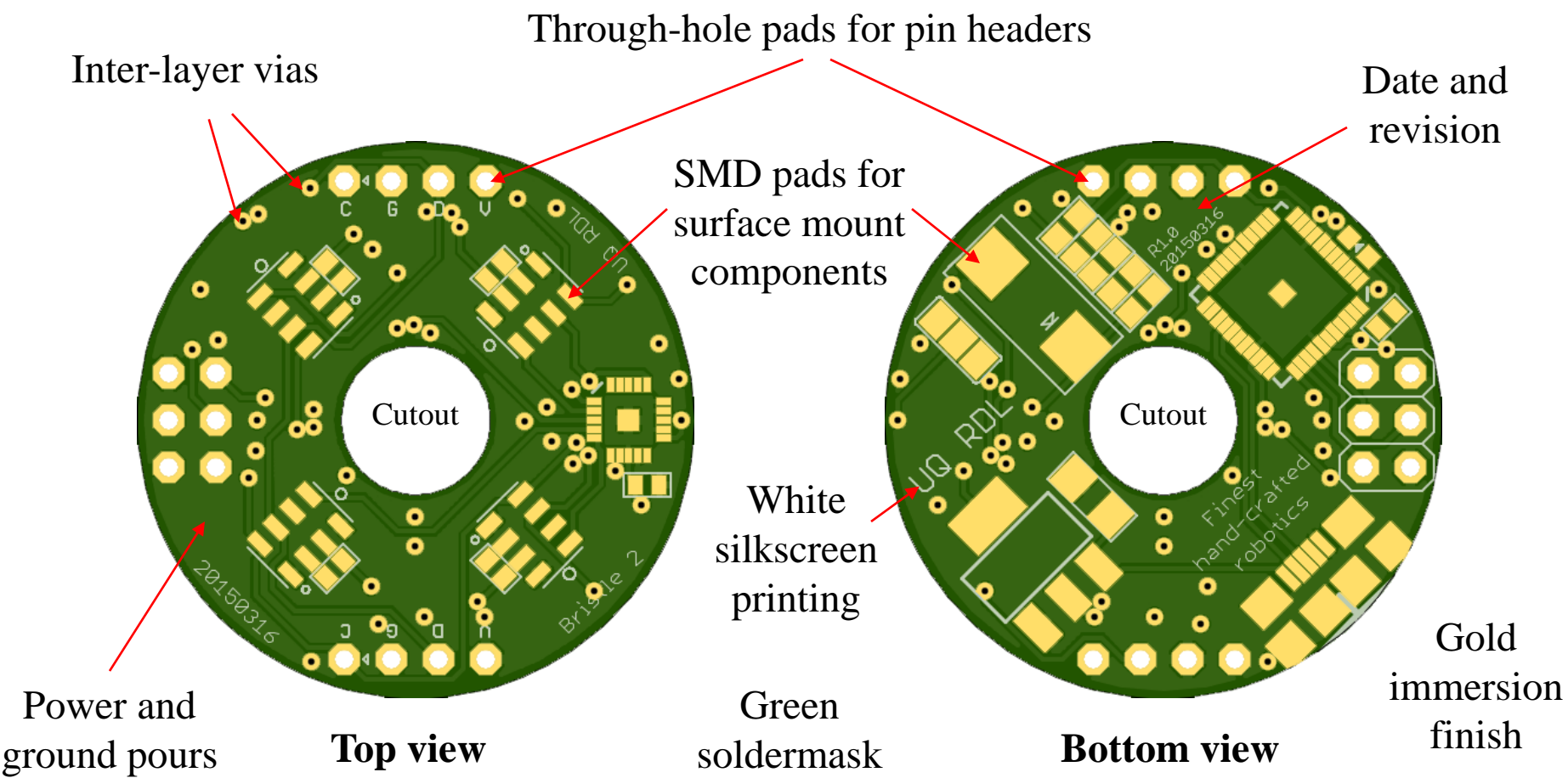
PCB structure

- Stack of conductive and insulating material
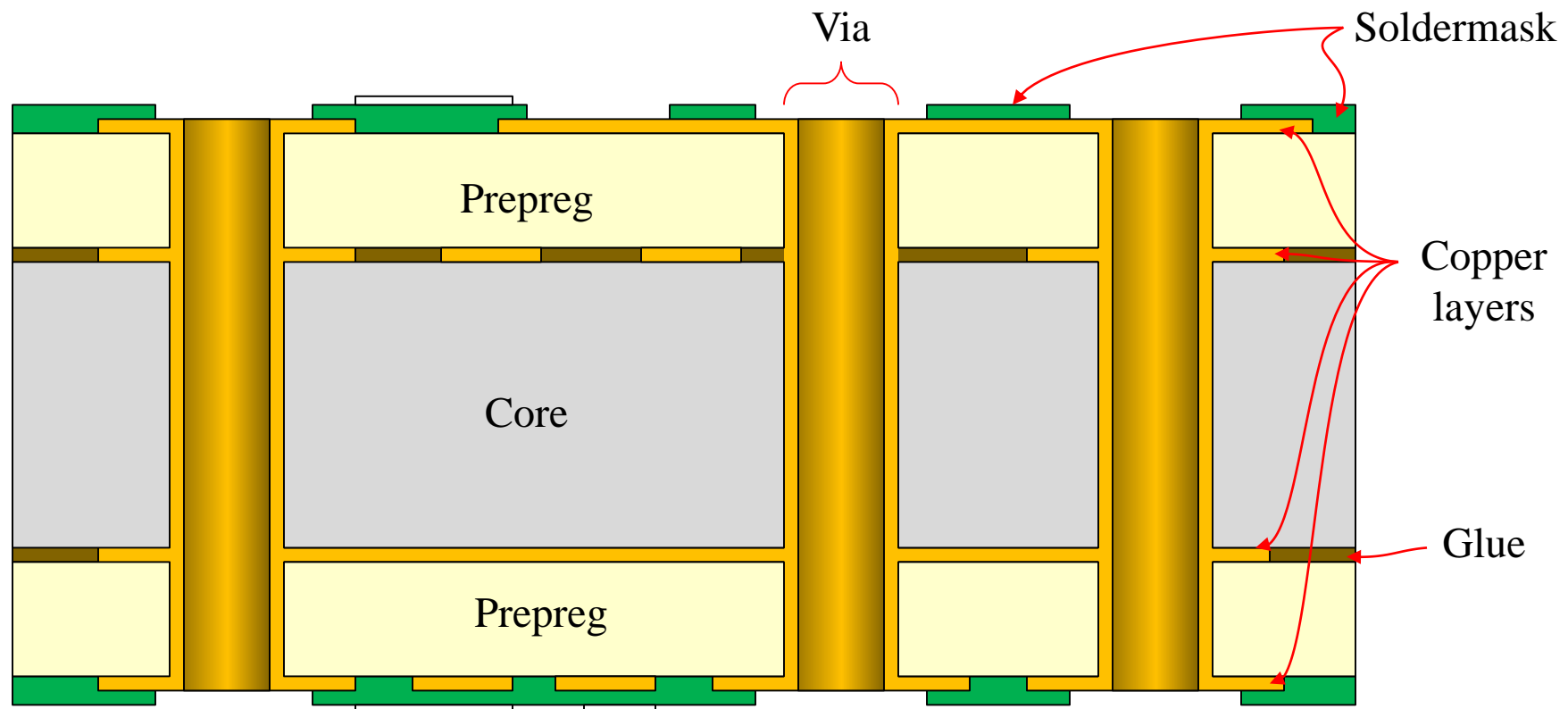
Handy summary of terms

- 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!
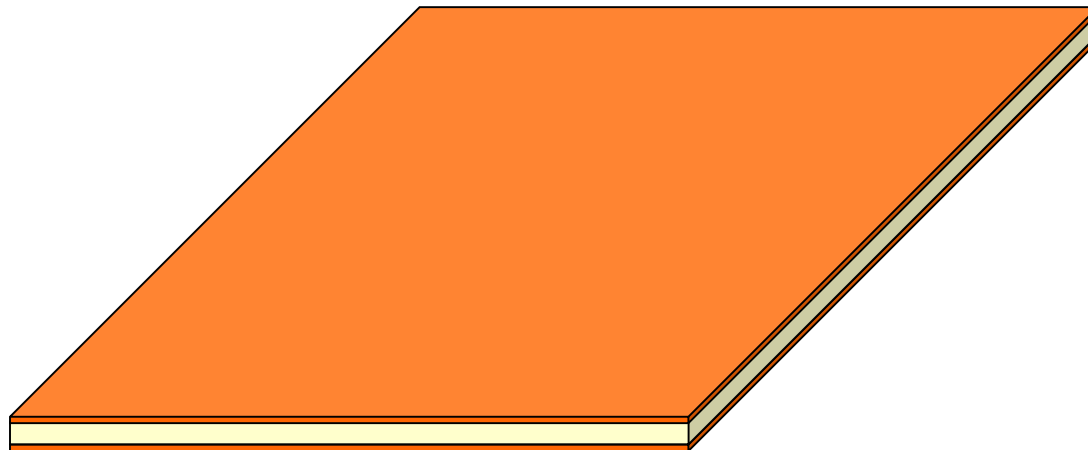
How a PCB is made

- Drill where holes will go
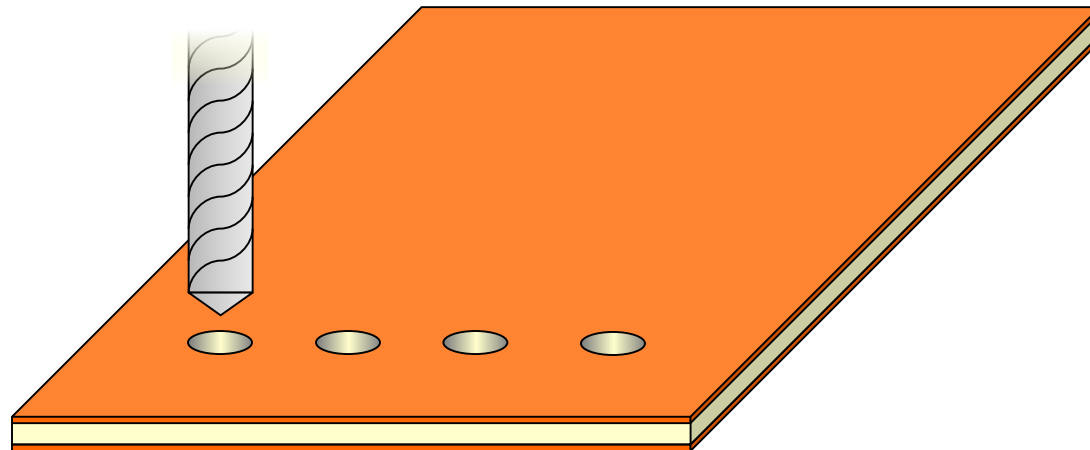
"Drilling"

How a PCB is made

- Wash the board with an electrochemical activator solution

Activator solution

"Activation"
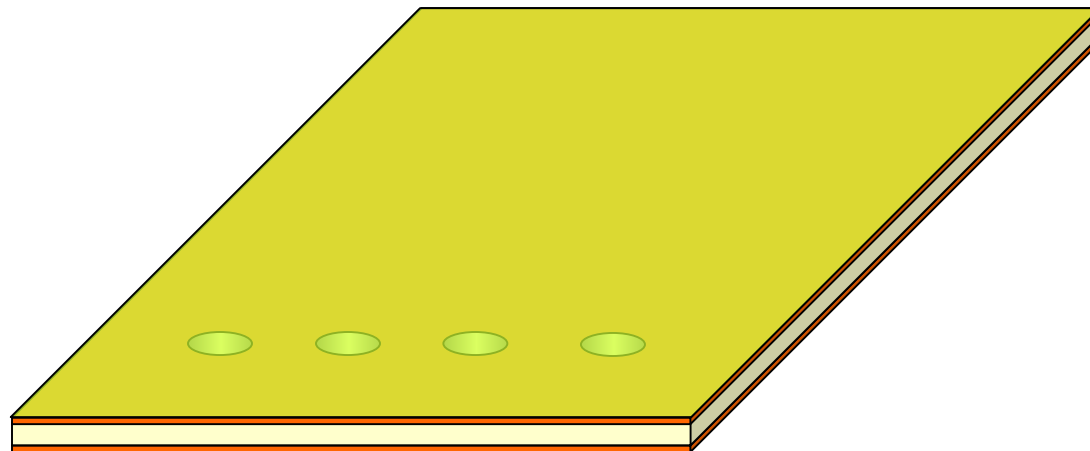
How a PCB is made

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

"Plating"
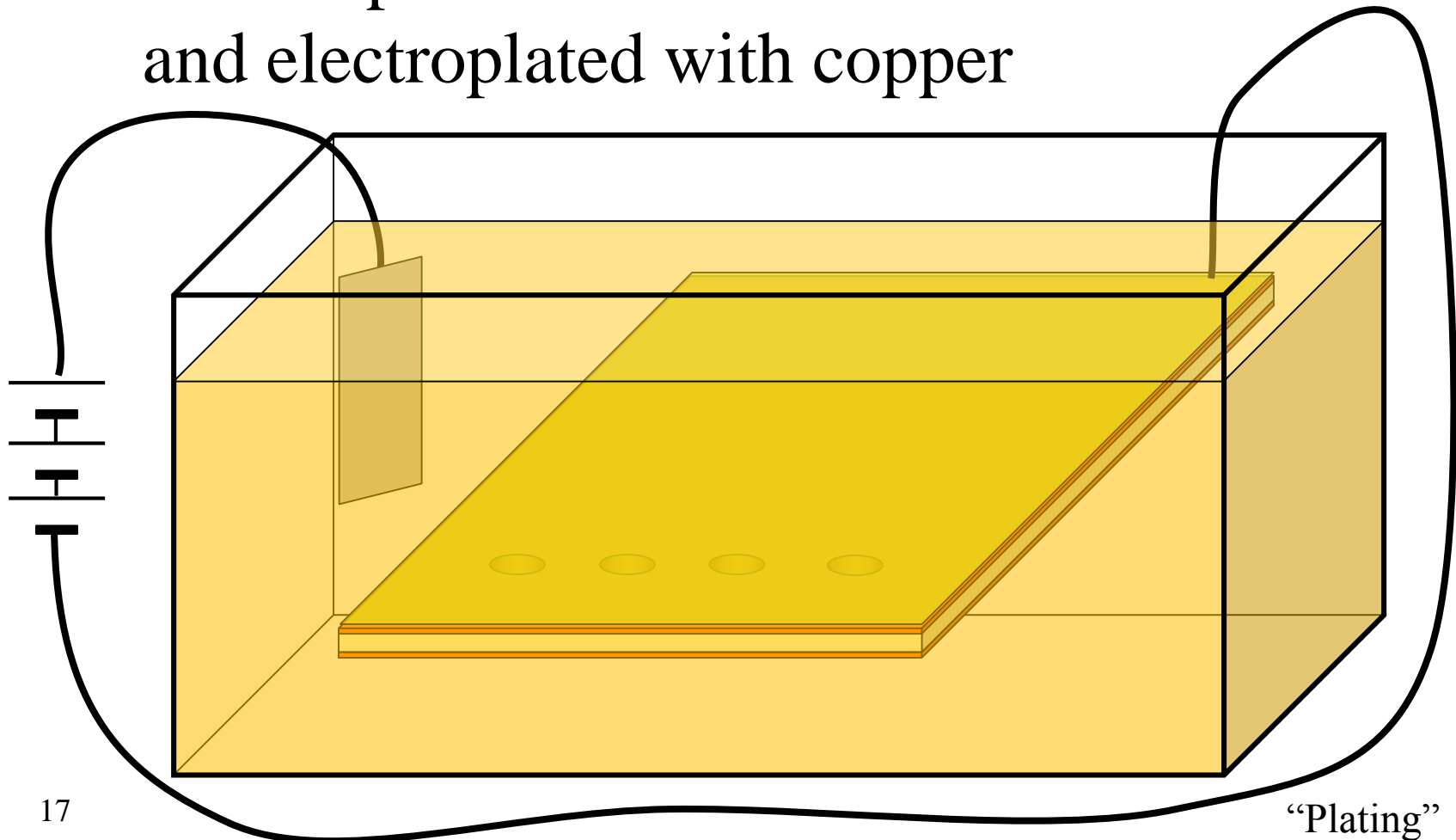
How a PCB is made
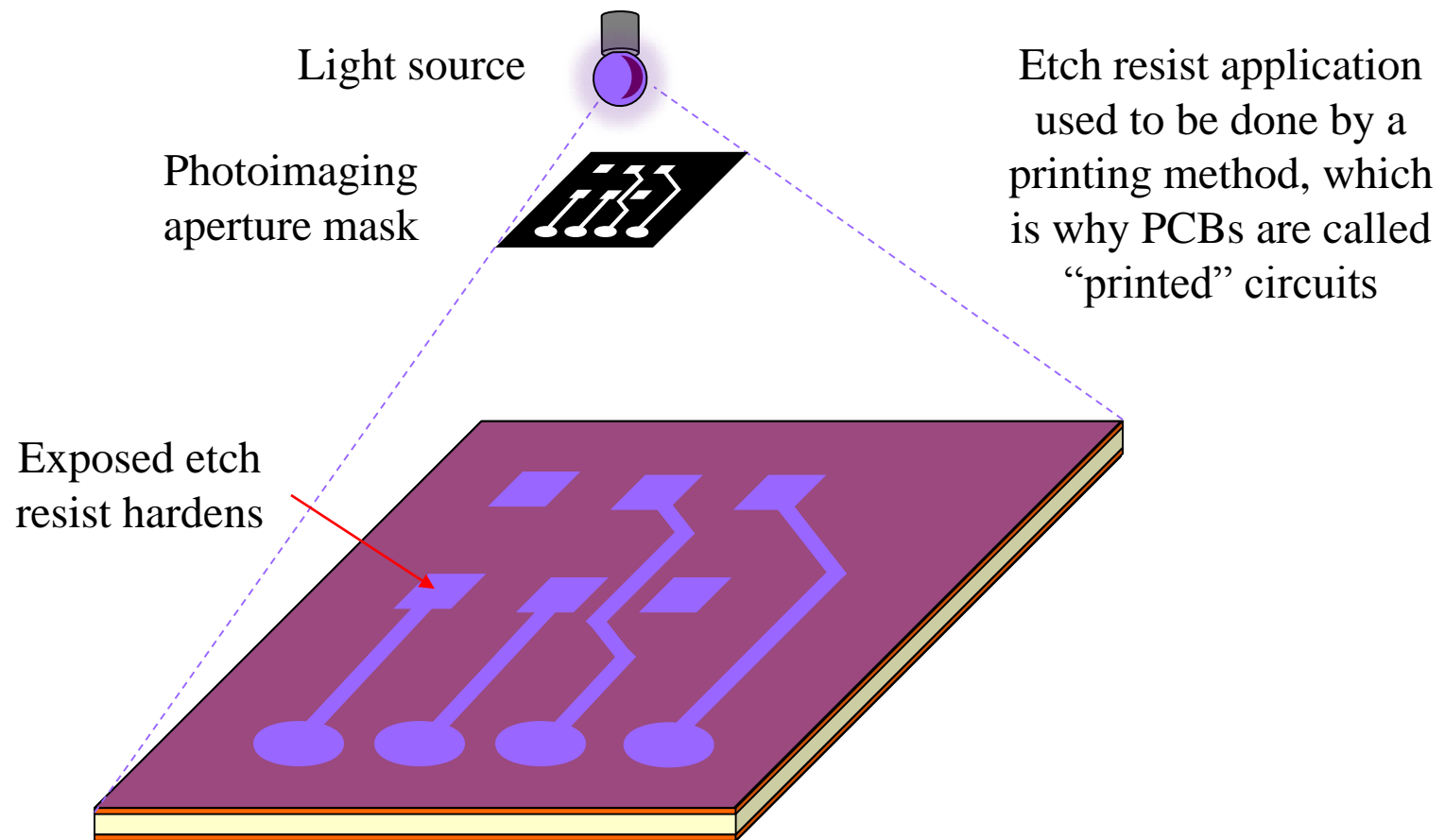
- PCB is removed from tank and washed

Water, or some
other solvent

Holes now
plated through

"Washing"
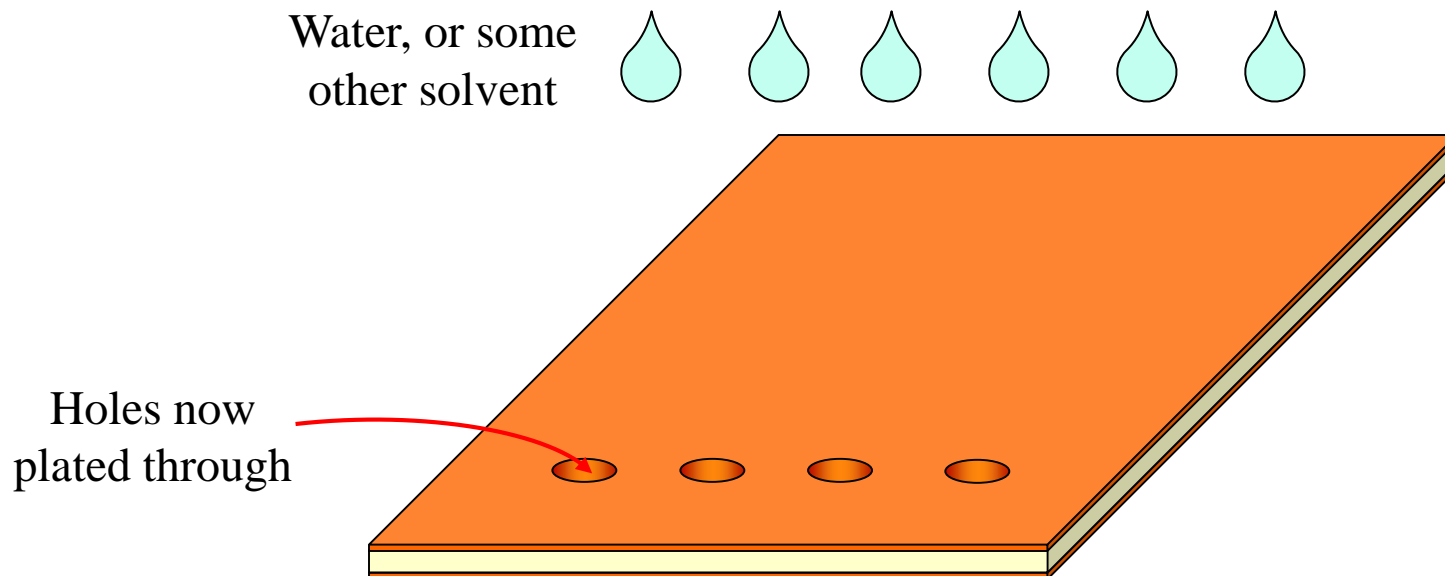
How a PCB is made

- Coat with photo-curing etch resist

Etch resist fluid

"Resist coating"

How a PCB is made

- Selectively expose to light (one or both sides)

Light source

Photoimaging
aperture mask

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

Exposed etch
resist hardens

20

"Photoimaging"
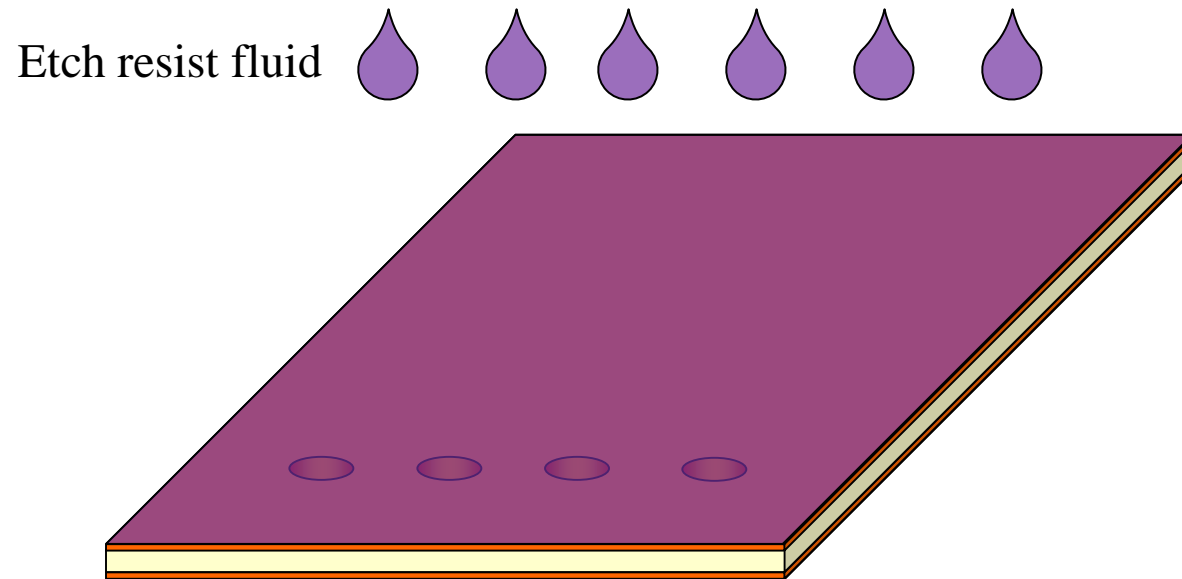
How a PCB is made

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

Water, or some other solvent

"Cleaning"
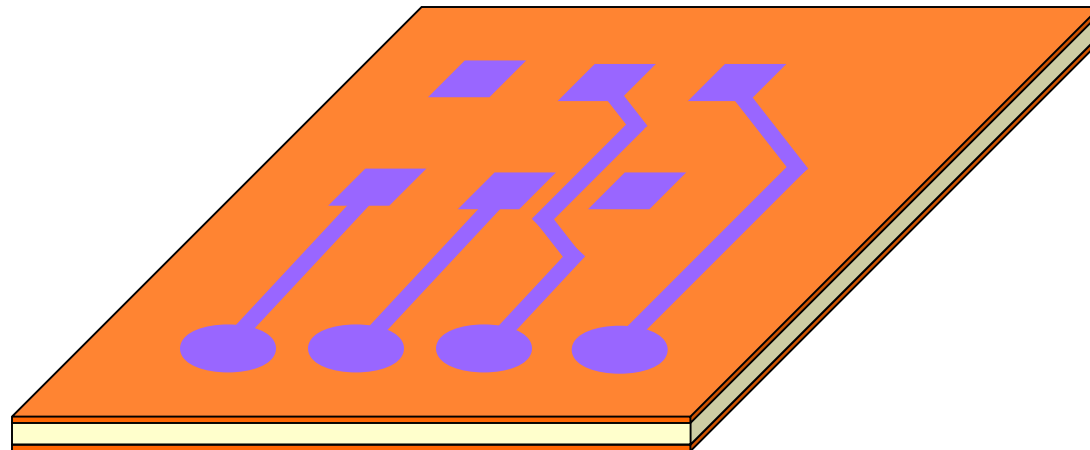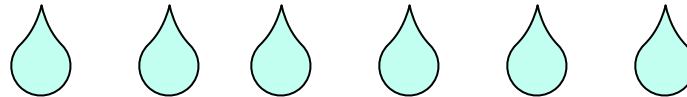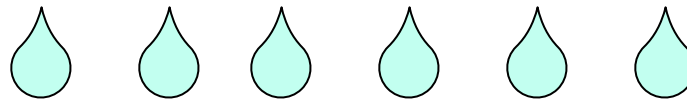
How a PCB is made

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

$FeCl_2 + CuCl_2$

$FeCl_3$ solution

Etchant (dangerous chemicals!)

"Etching"
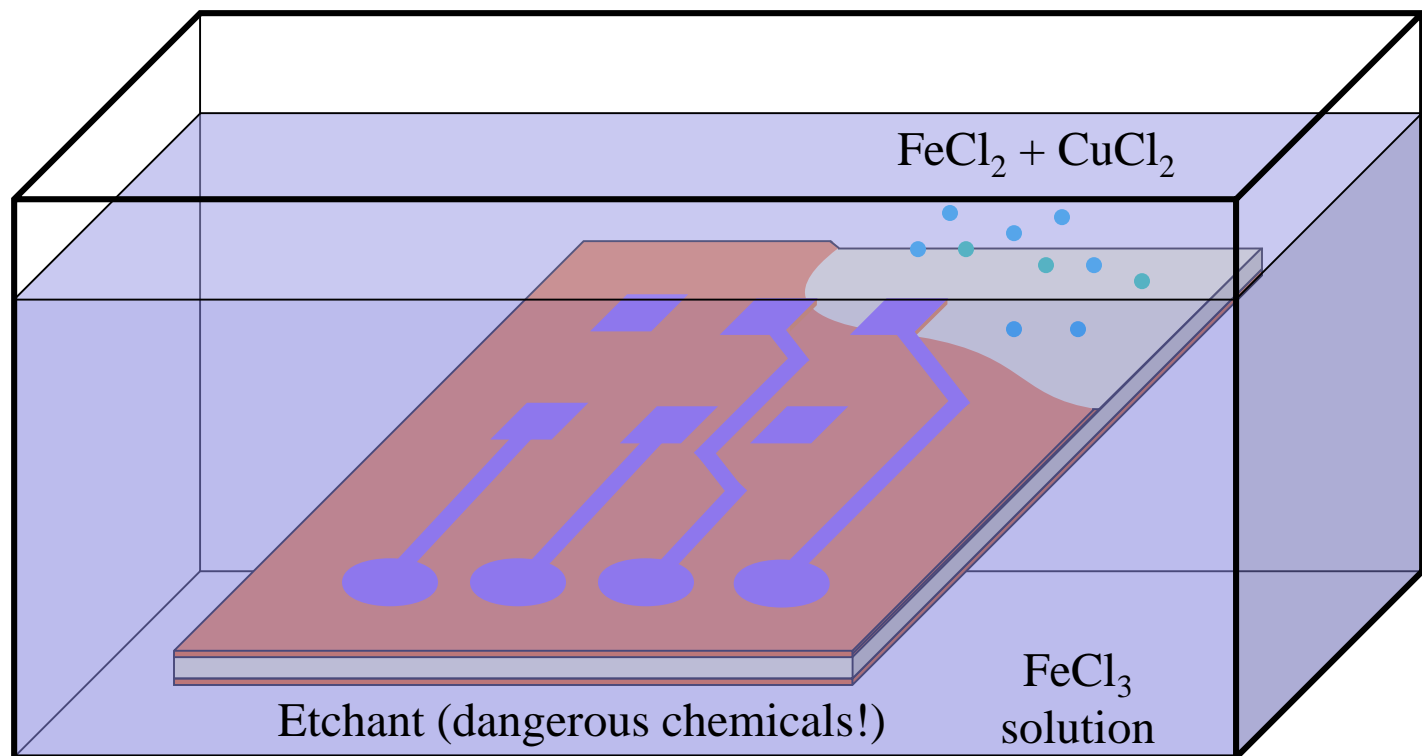
How a PCB is made

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

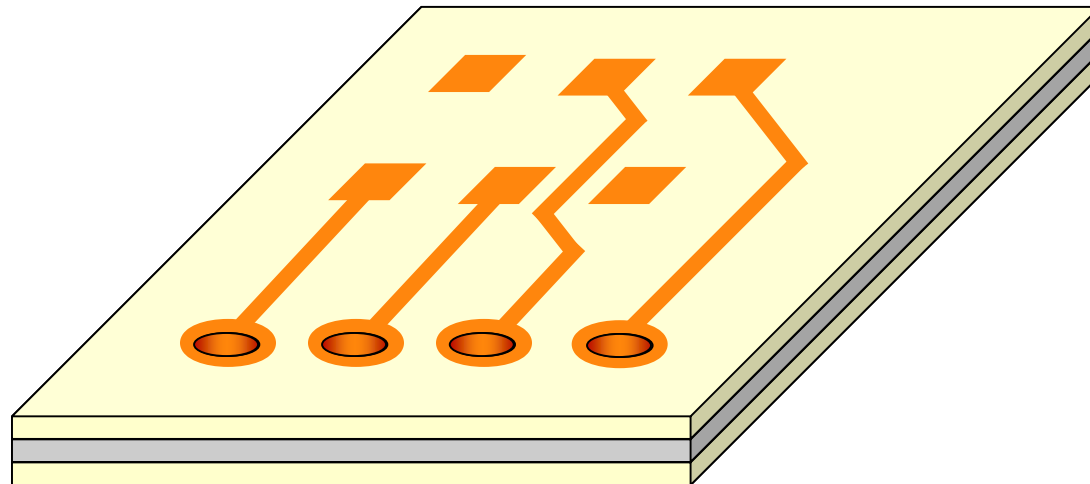Water, or some other solvent
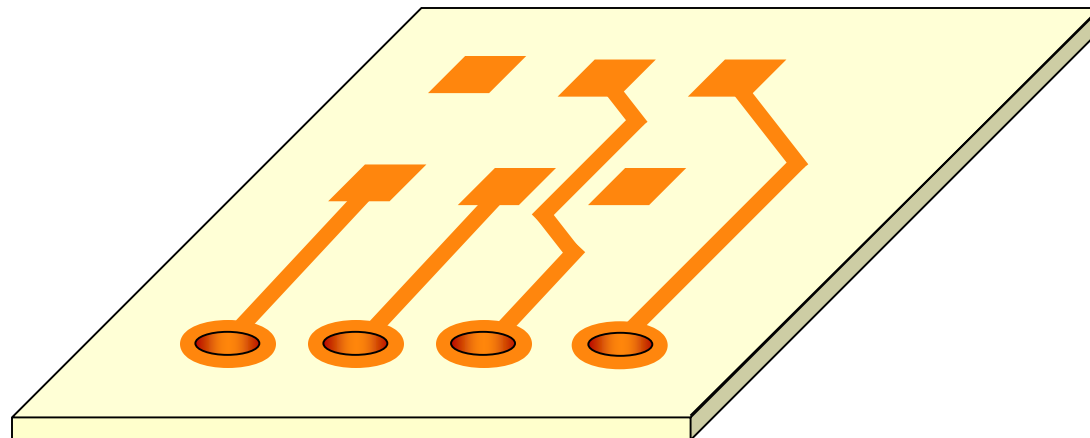
Simple single-layer boards could stop here

"Cleaning"

How a PCB is made

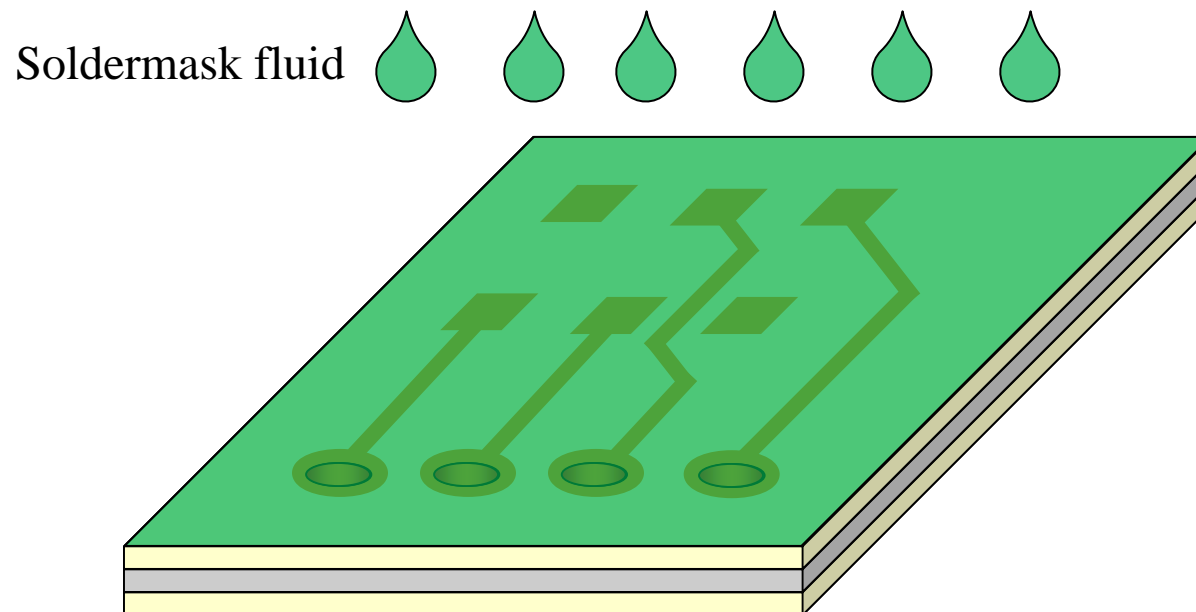- Multilayer board layers are coated with glue, aligned and then pressed together

Pressed with tons of force

Adhesive layer

Unclad core

"Laminating"
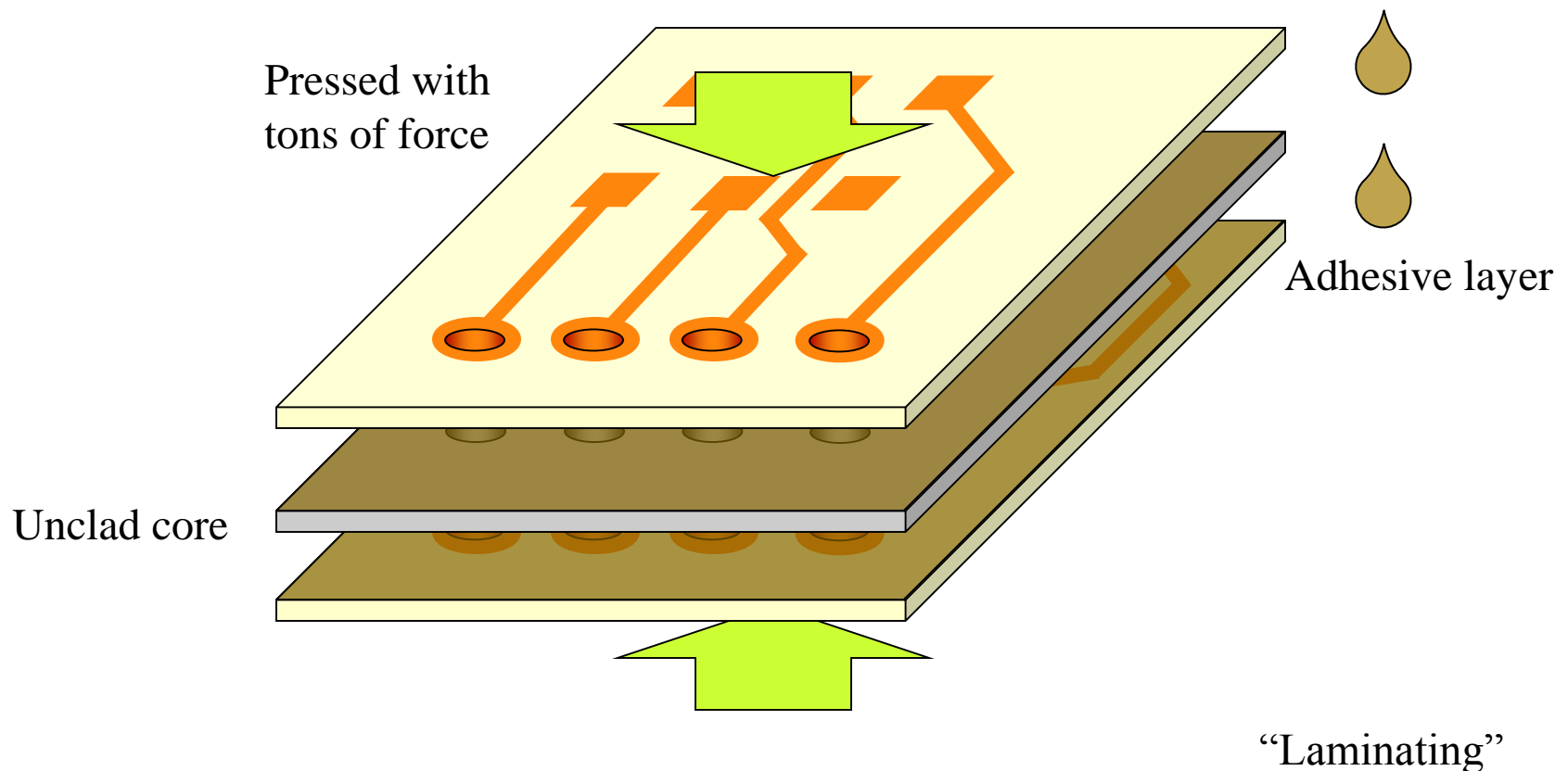
How a PCB is made

- 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 fluid

"Soldermask"

How a PCB is made

- Selectively expose to light (one or both sides)

Light source

Photoimaging
aperture mask

Unexposed
soldermask
remains
uncured

27
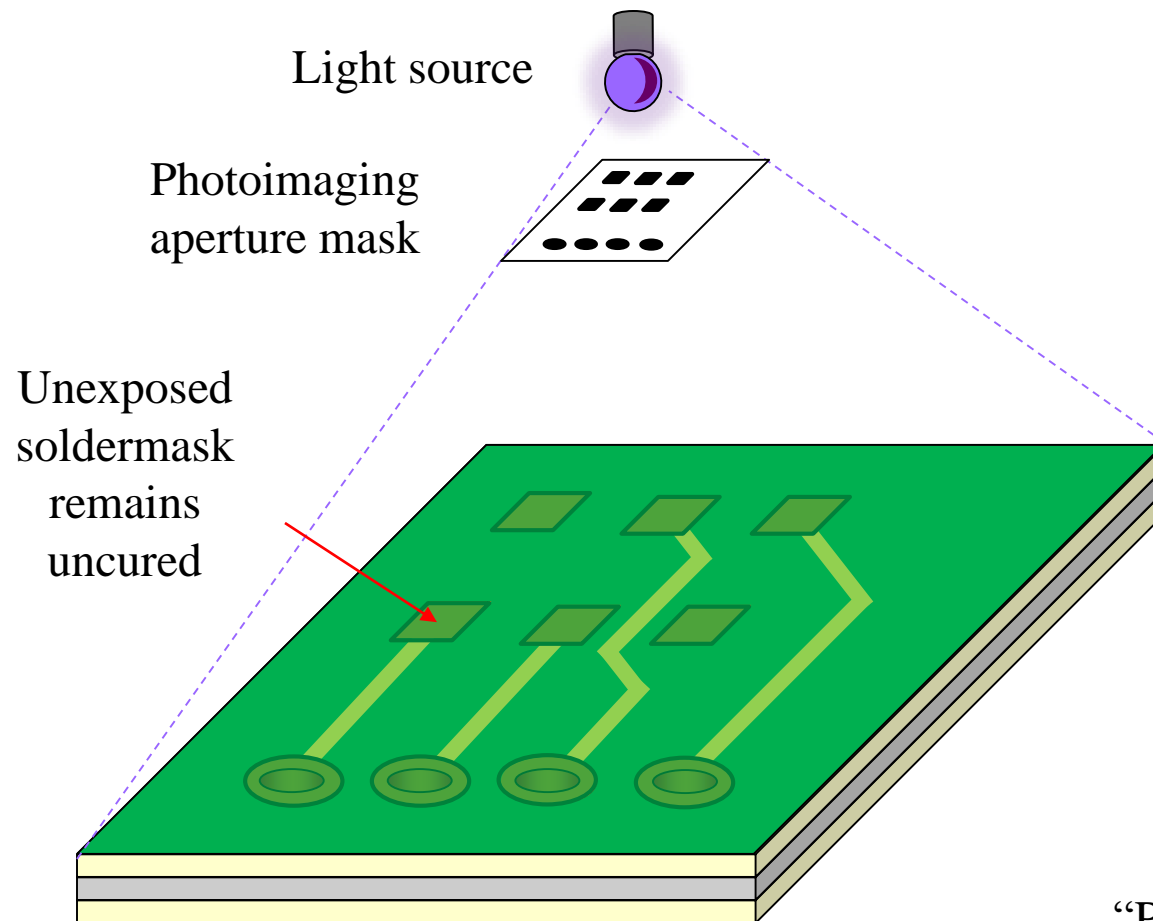
"Photoimaging"

How a PCB is made

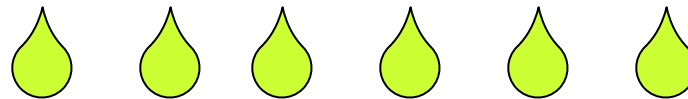- Wash uncured soldermask away, exposing copper pads, vias and holes
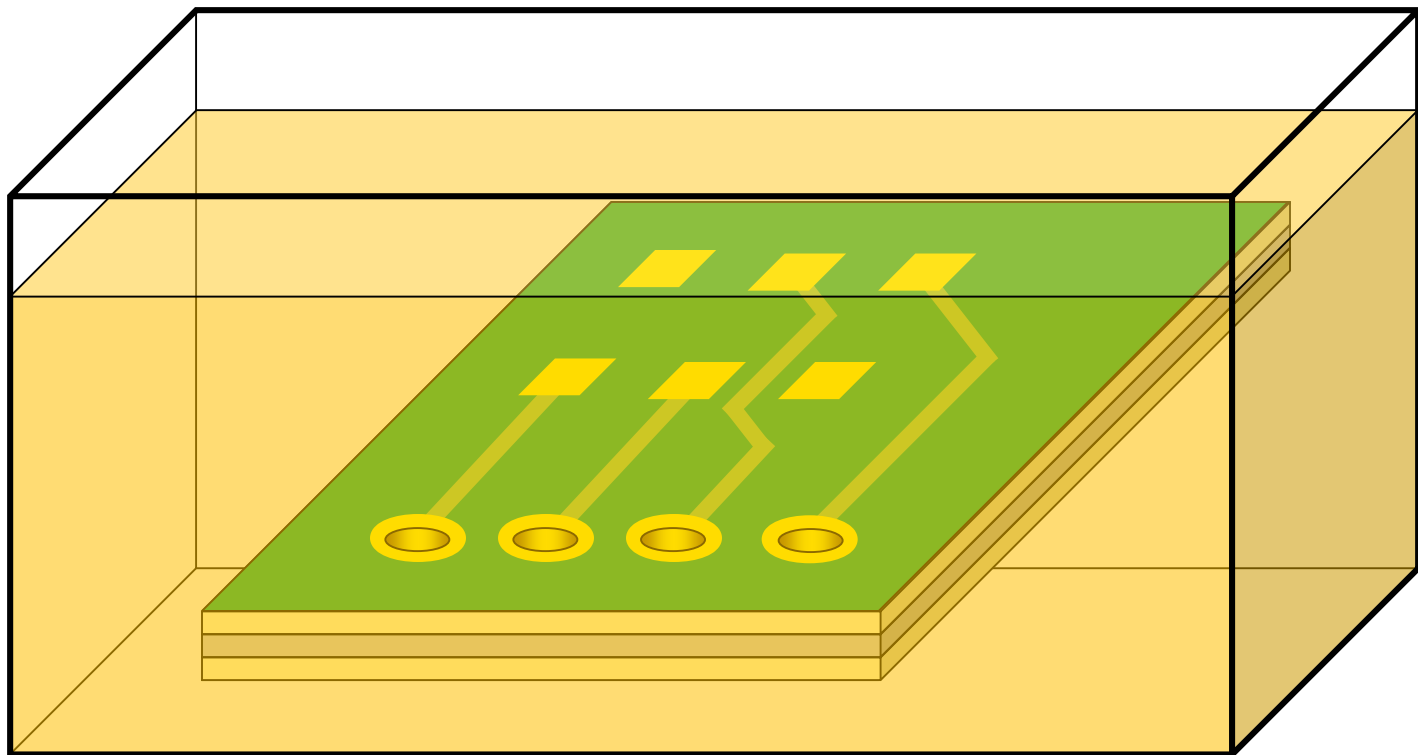
Water, or some other solvent
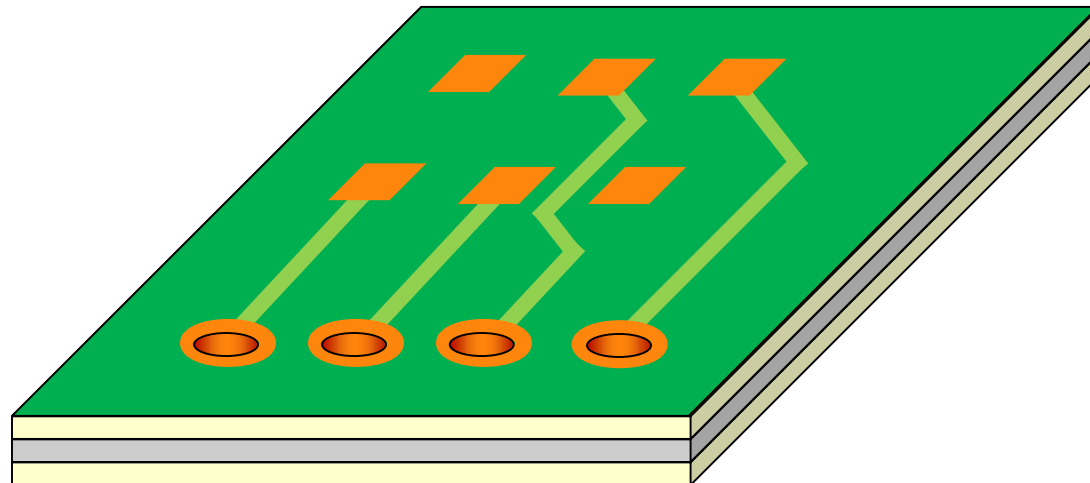
"Washing"

How a PCB is made

- Wash the board with an electrochemical activator solution
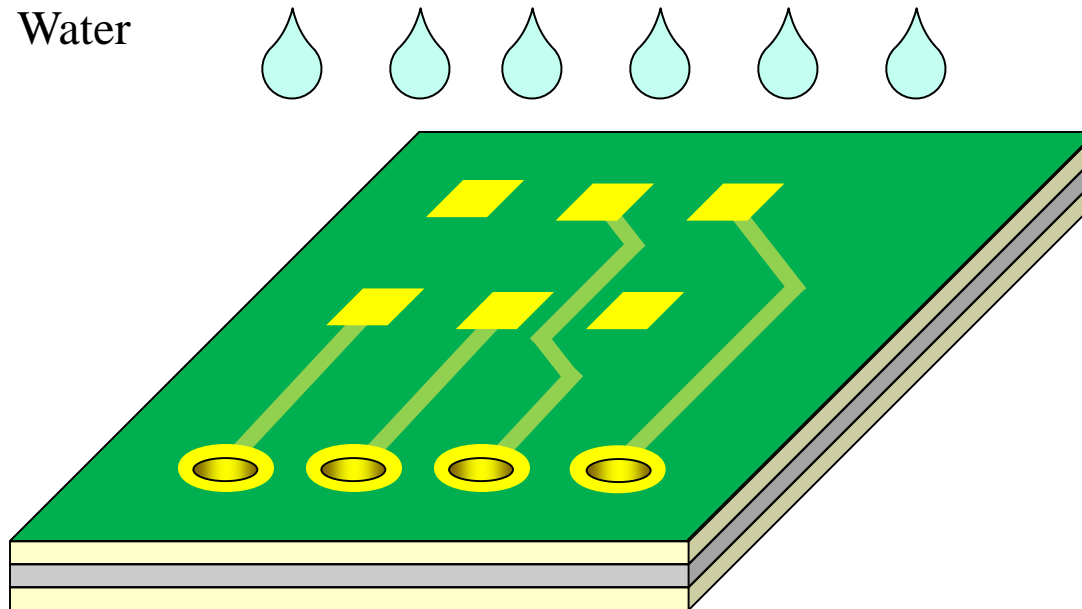
Activator solution

"Activation"
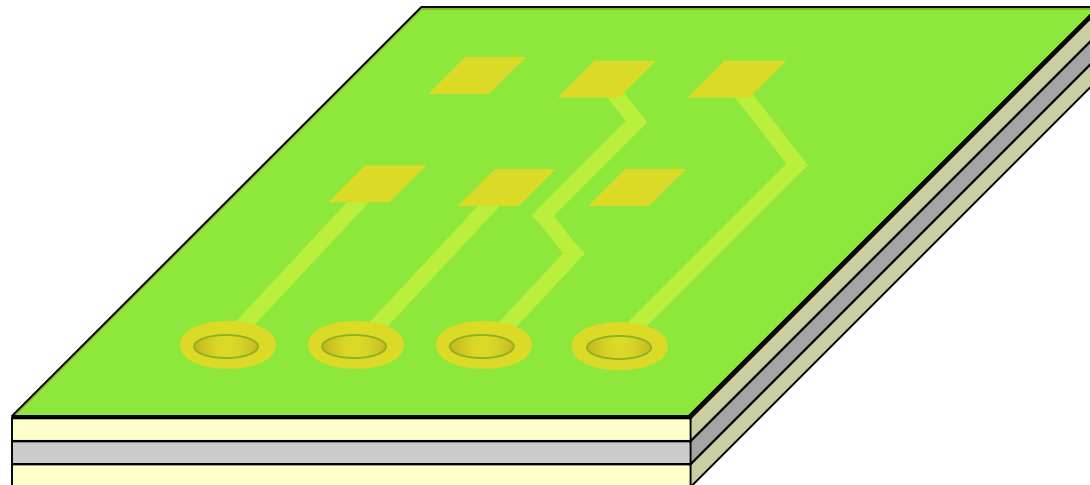
How a PCB is made

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

"Plating"
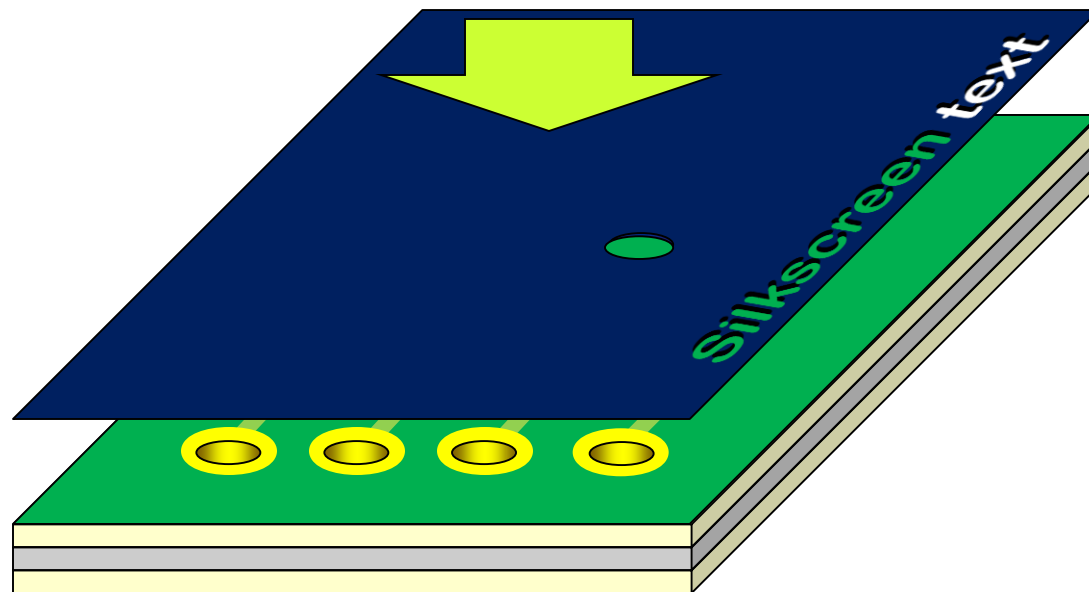
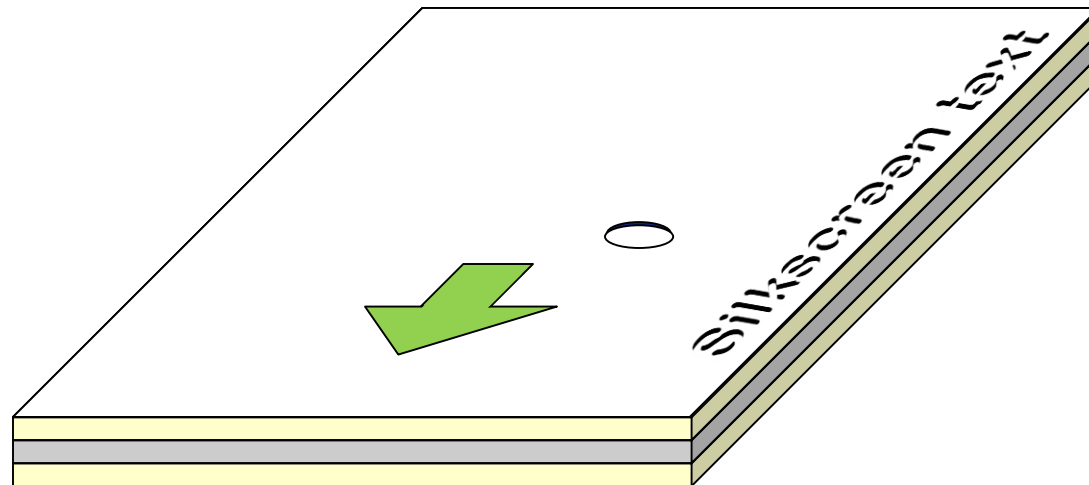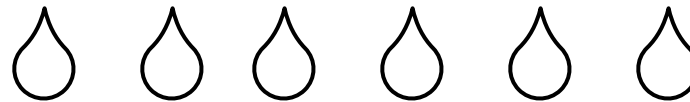# How a PCB is made

- PCB is washed clean of chemicals

Water

"Washing"

How a PCB is made

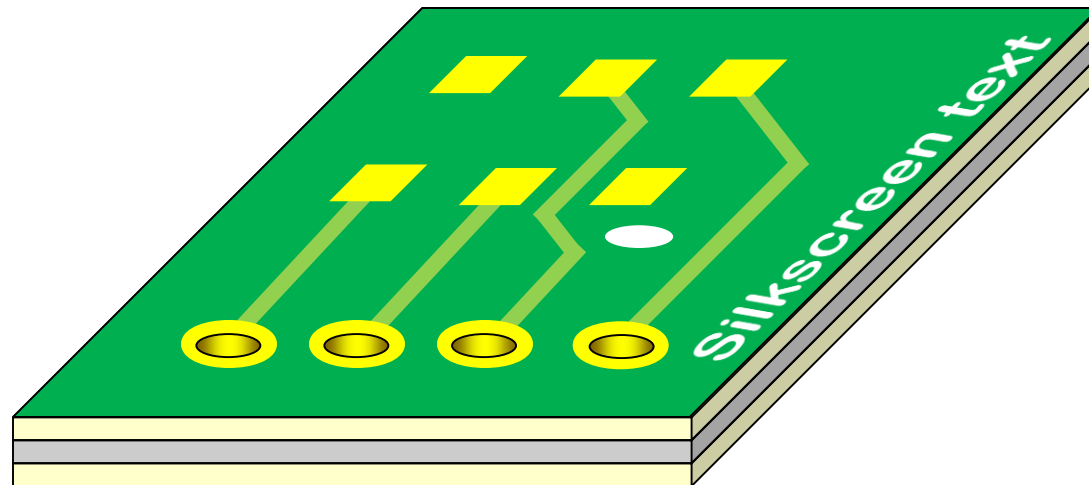- Silkscreen stencil is laid on; made using a photo-cured emulsion

"Stencilling"

How a PCB is made

- Ink is applied to the stencil, filling the voids

Silkscreen ink
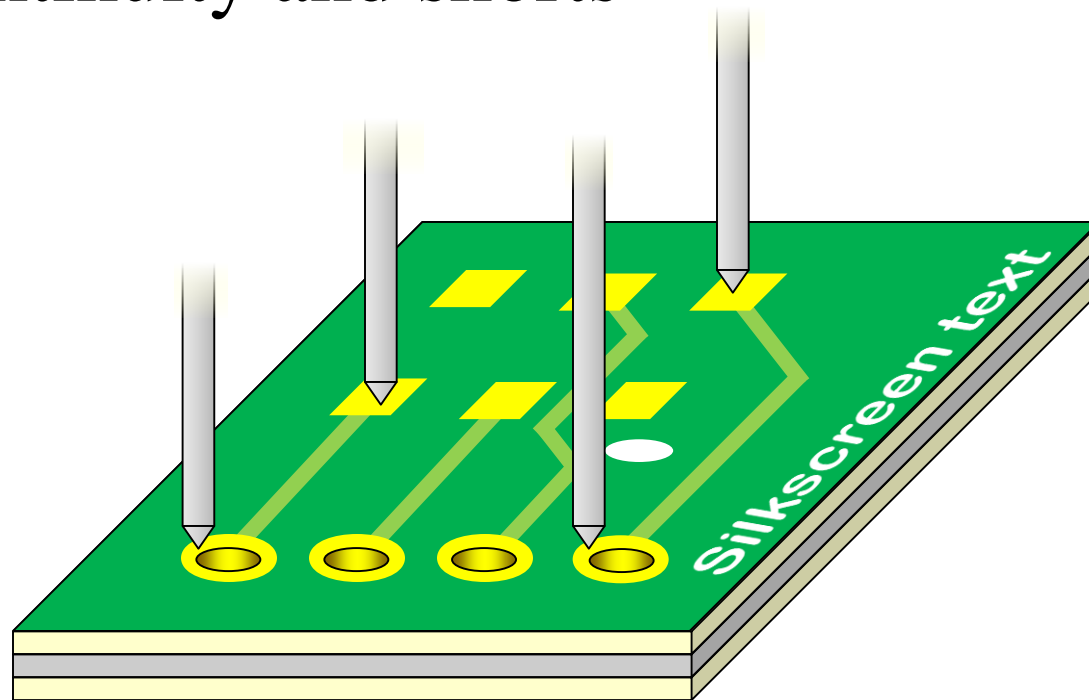rolled across surface

Silkscreen text

"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
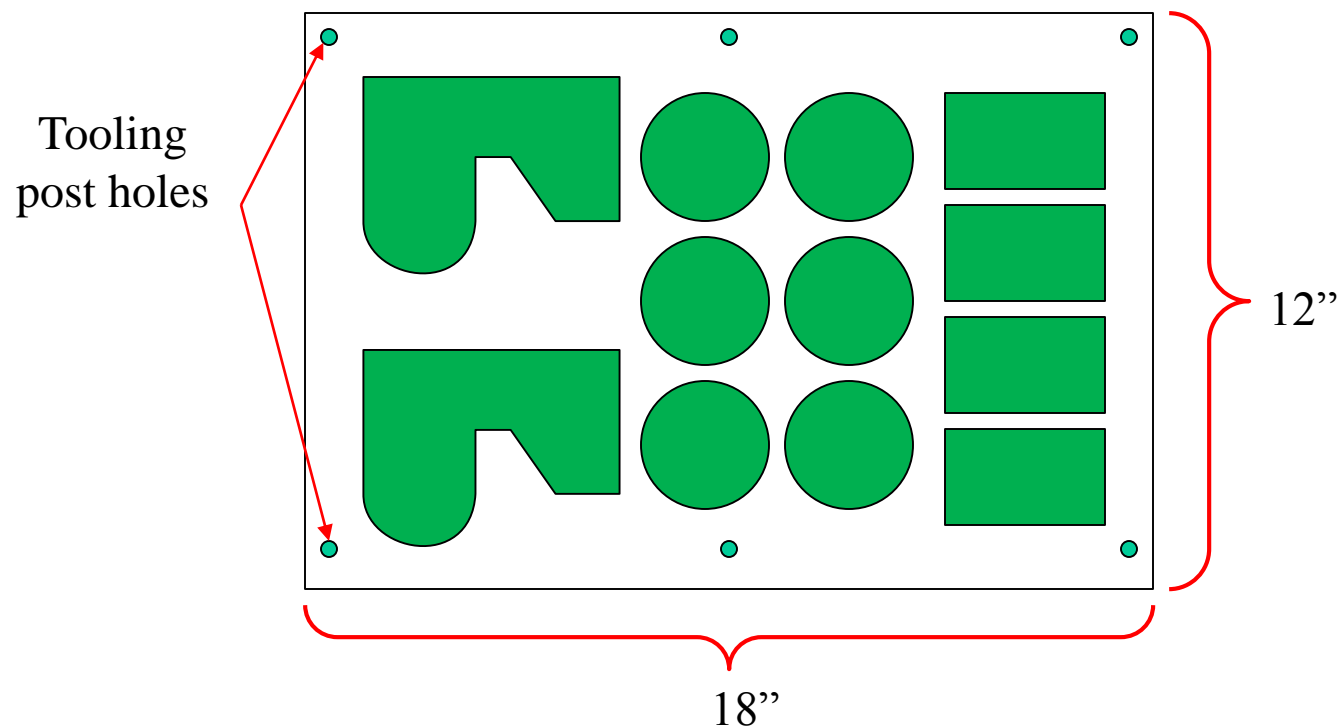
"E-test" or "Bed of nails"

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

Tooling post holes

12"

18"

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)	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.

39

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
```
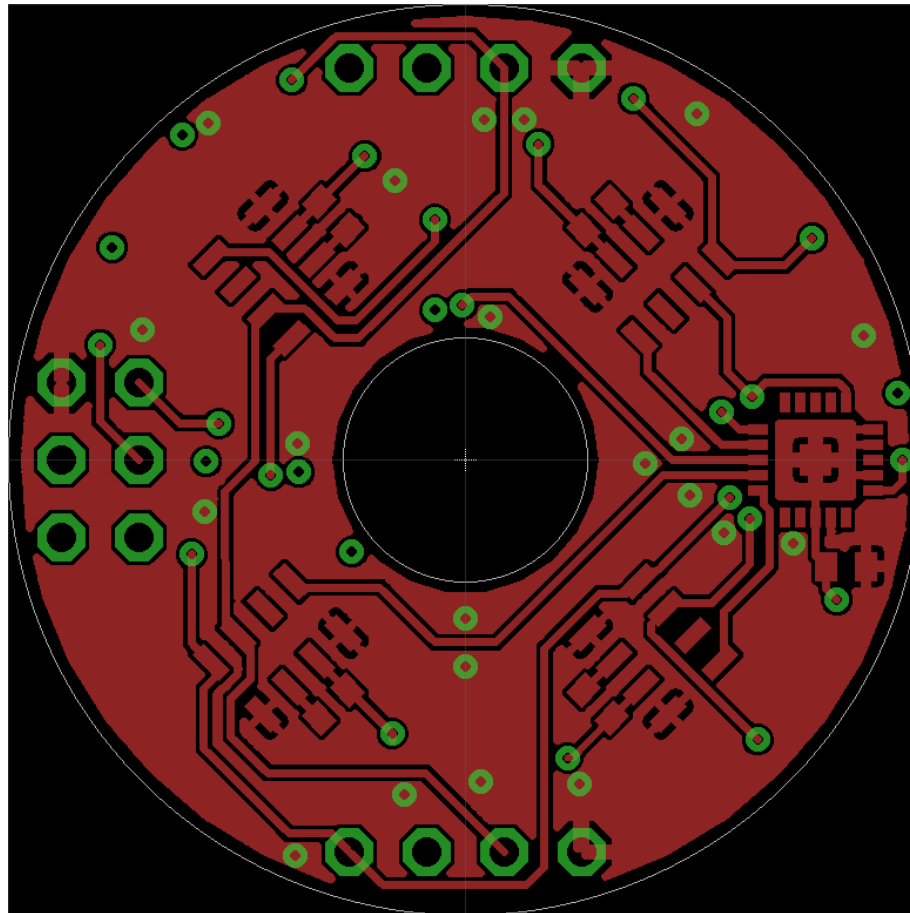
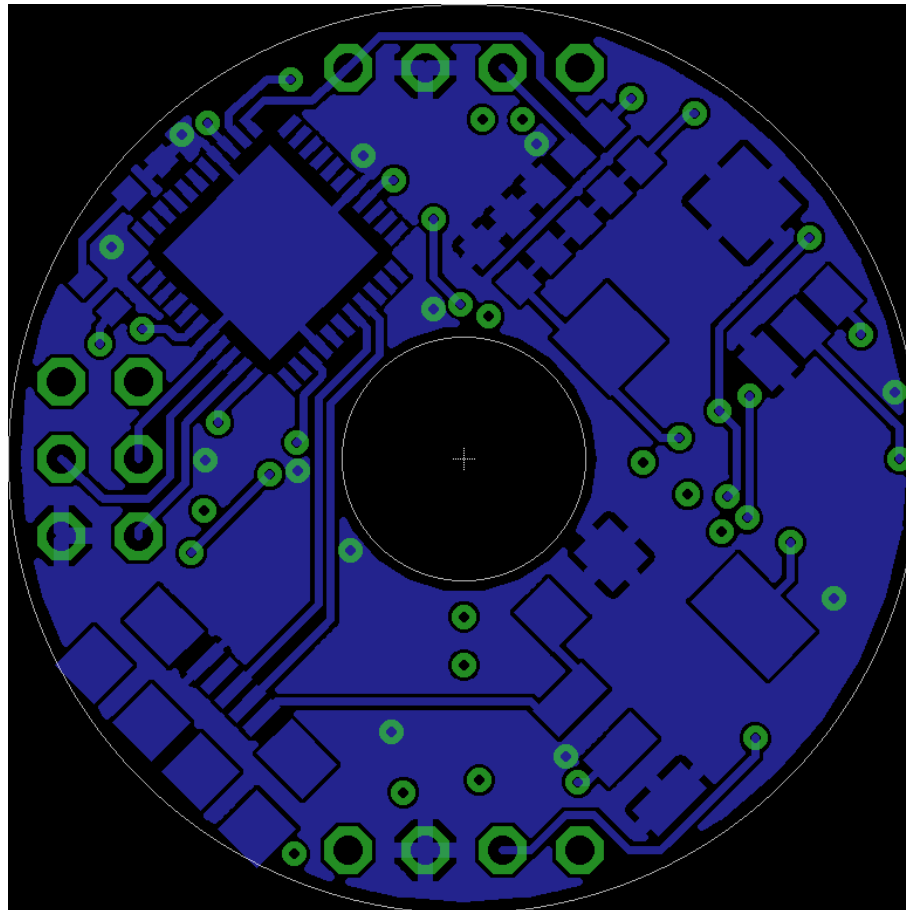# Mechanical layer

- Boring simple drawing of the board outline
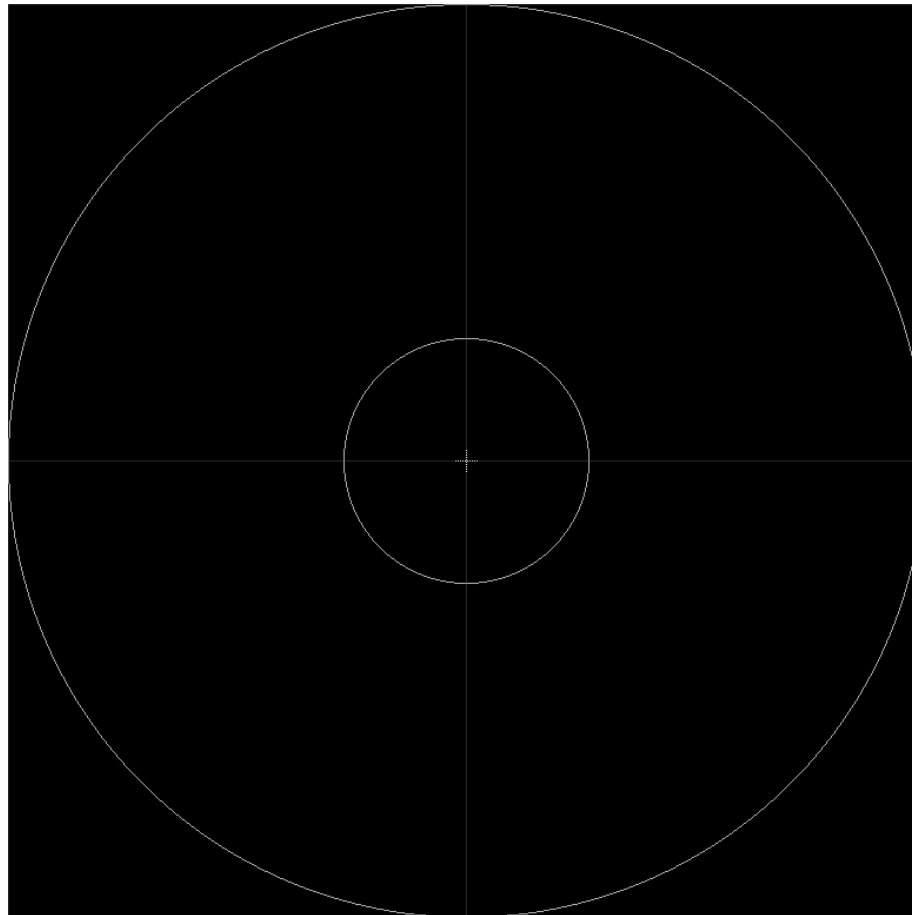
Top layer

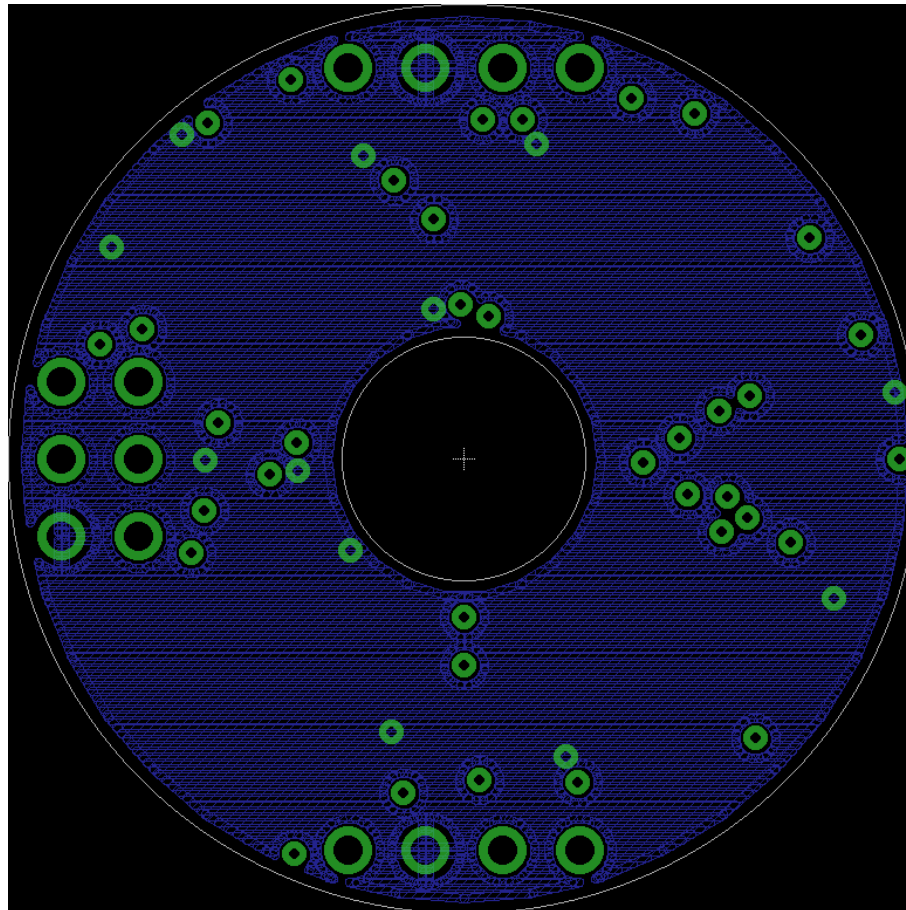- Aka "component side" – tracks plus pads, vias

Bottom layer

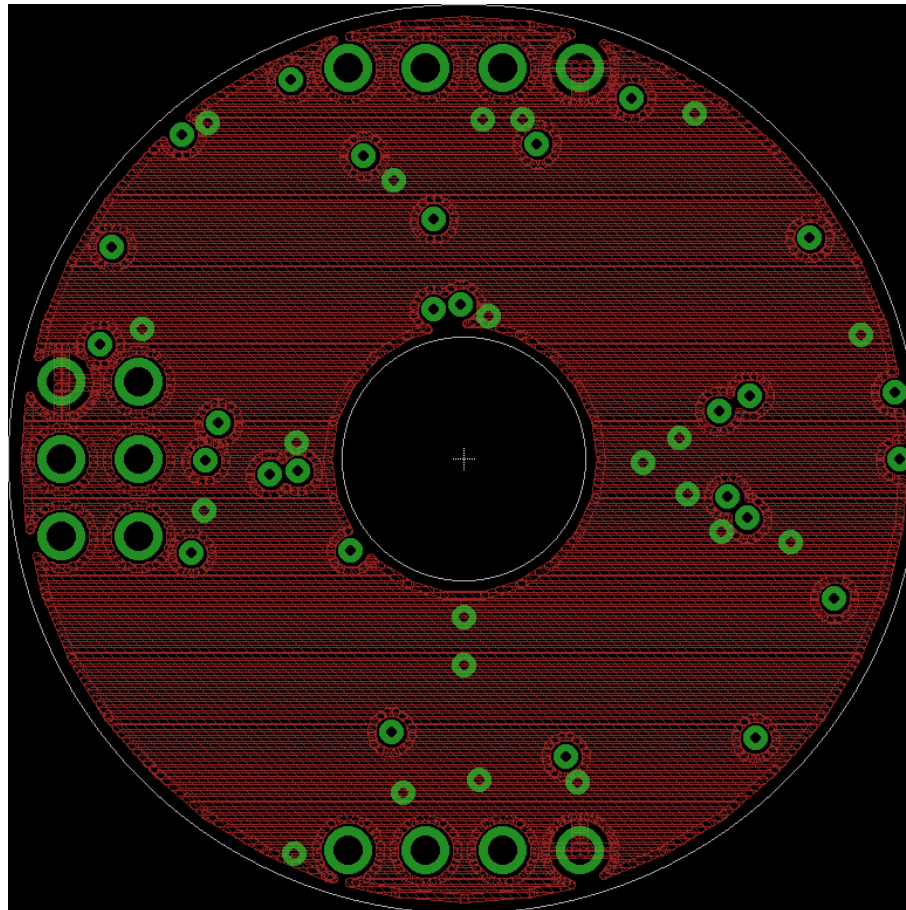- Aka "solder side" – tracks plus pads, vias
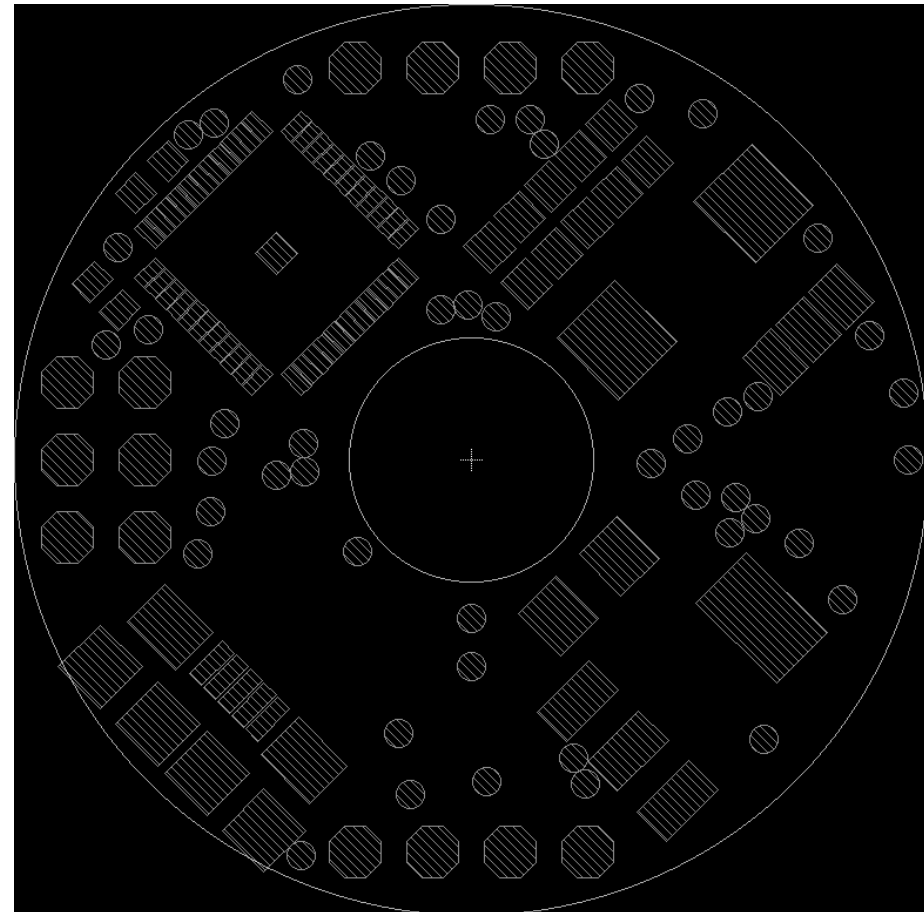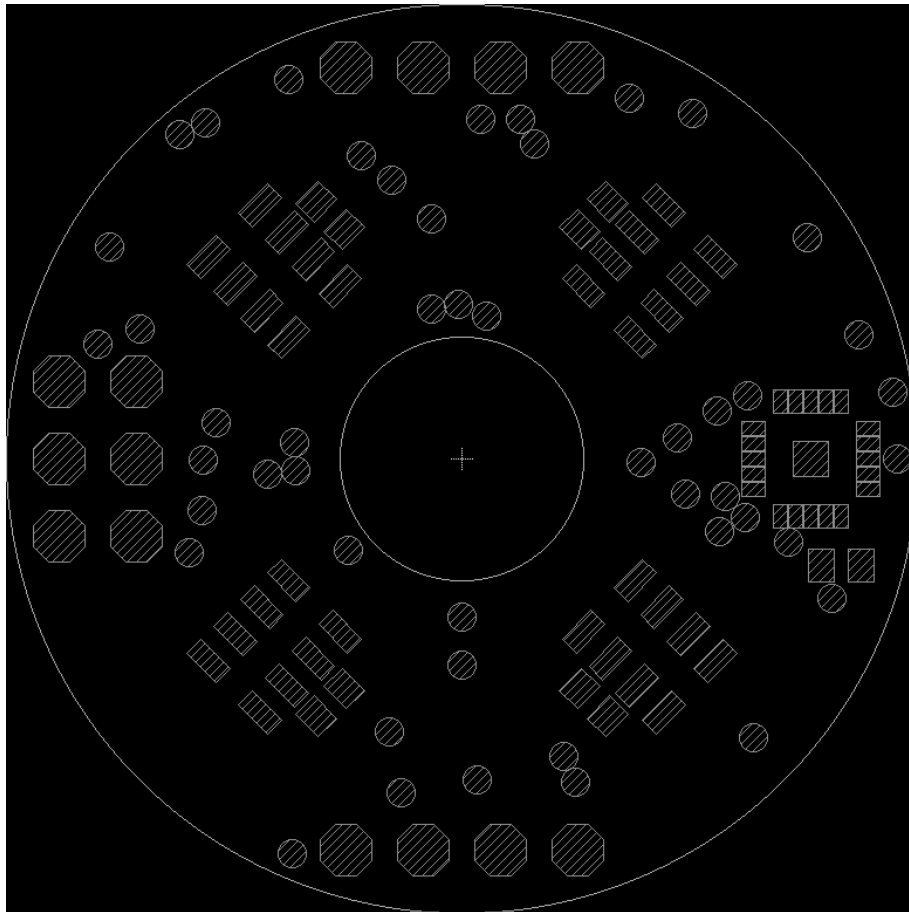
Inner layer 1

- Middle ground plane

Inner layer 2
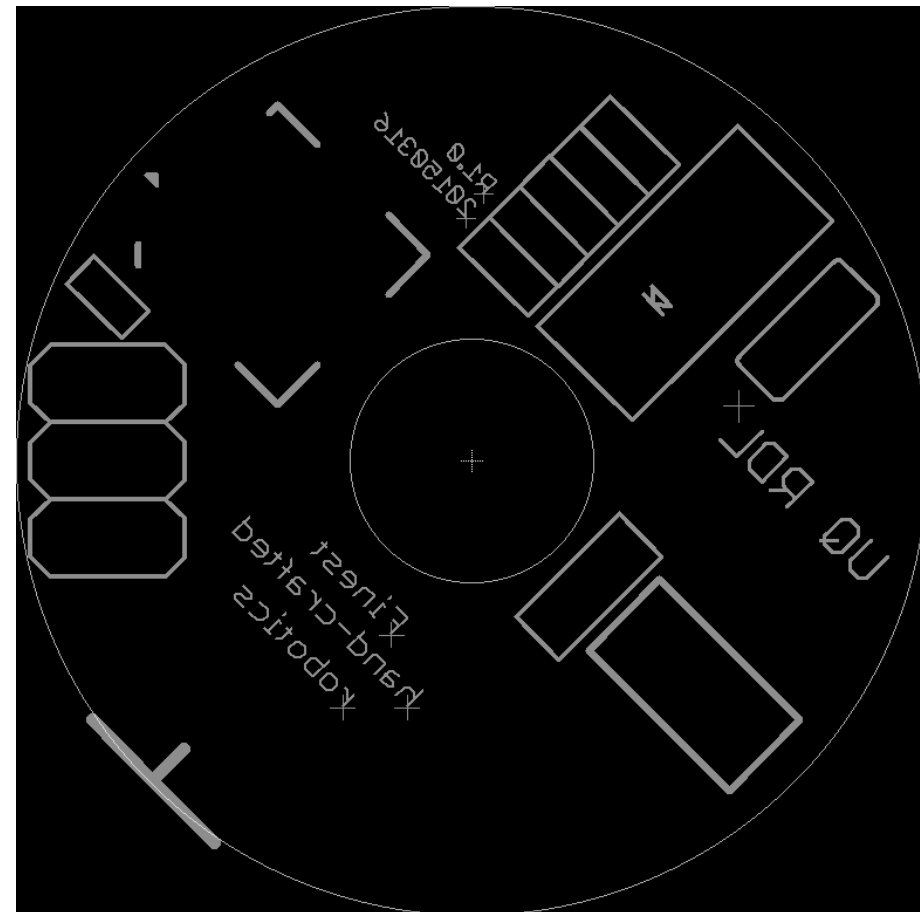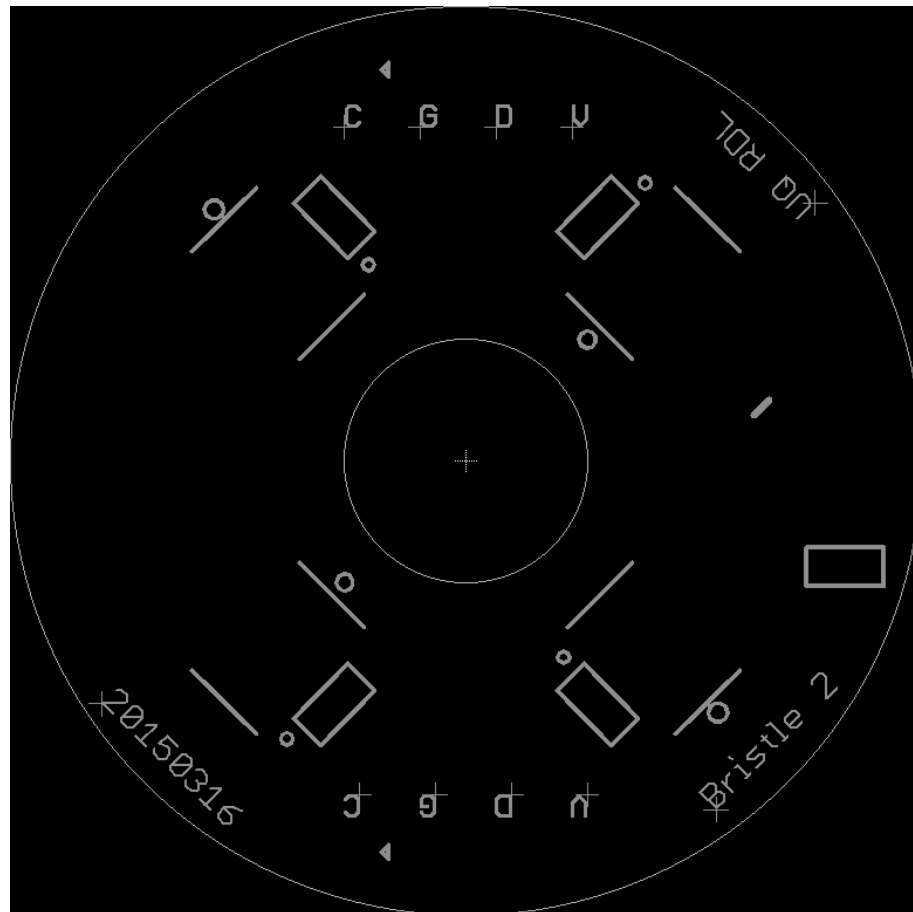
- Middle power plane

Top and bottom stop layers
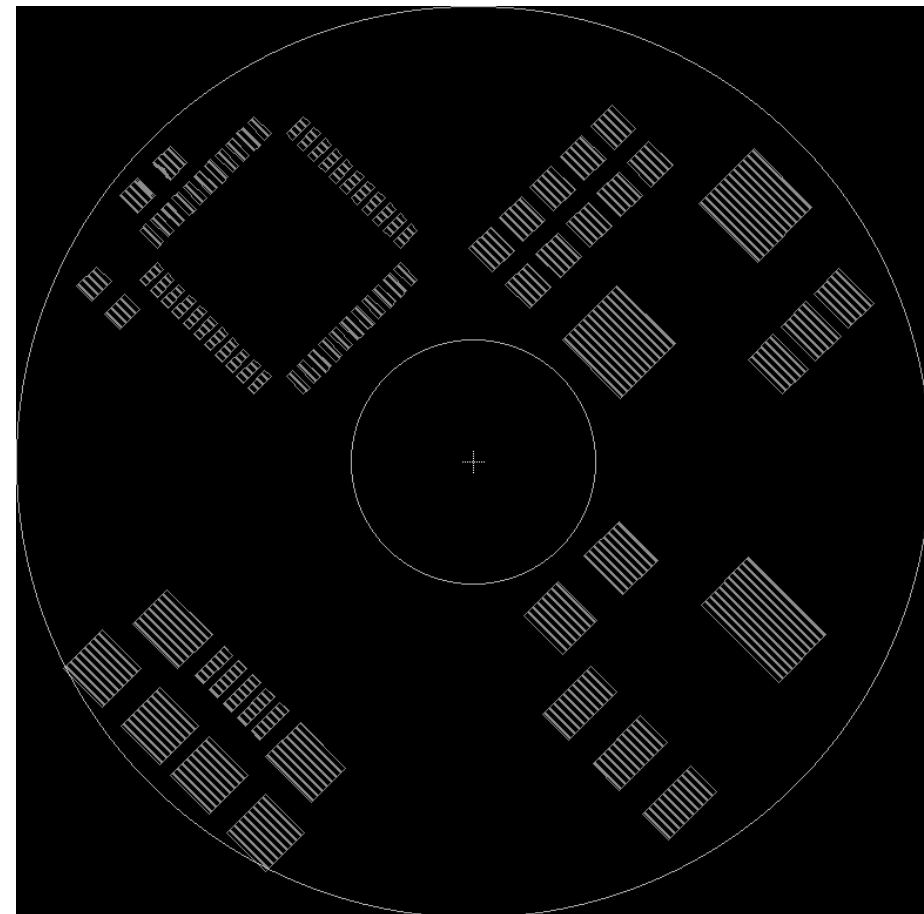
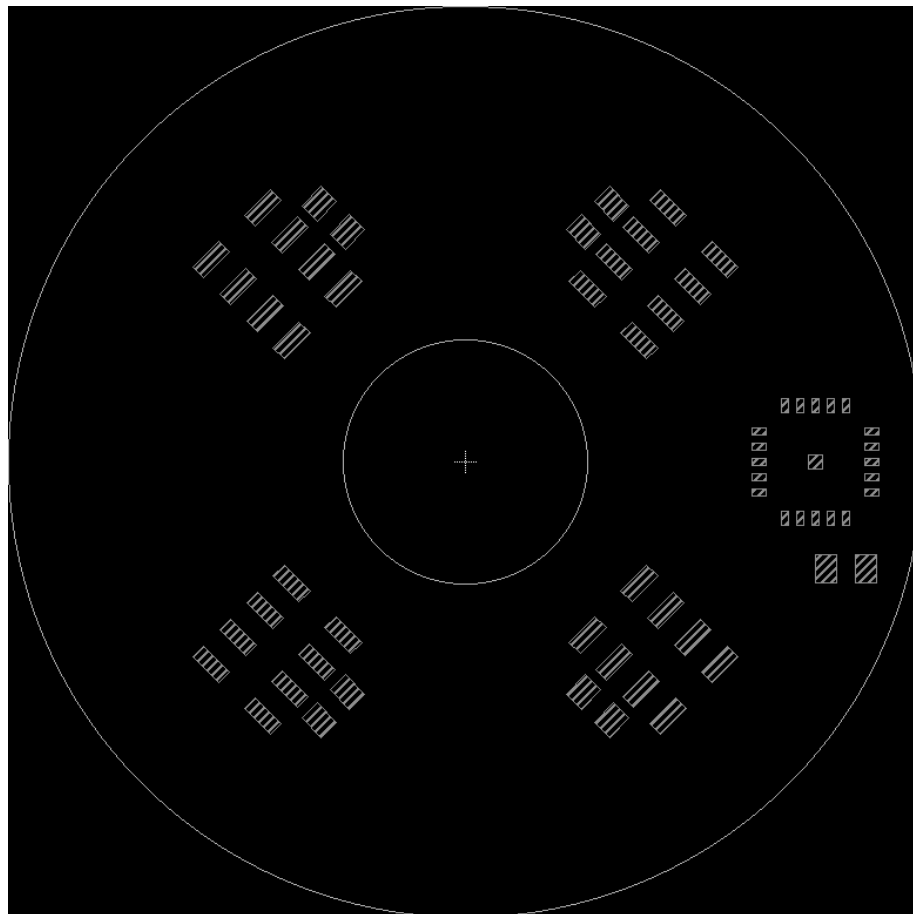- Soldermask apertures – places without mask

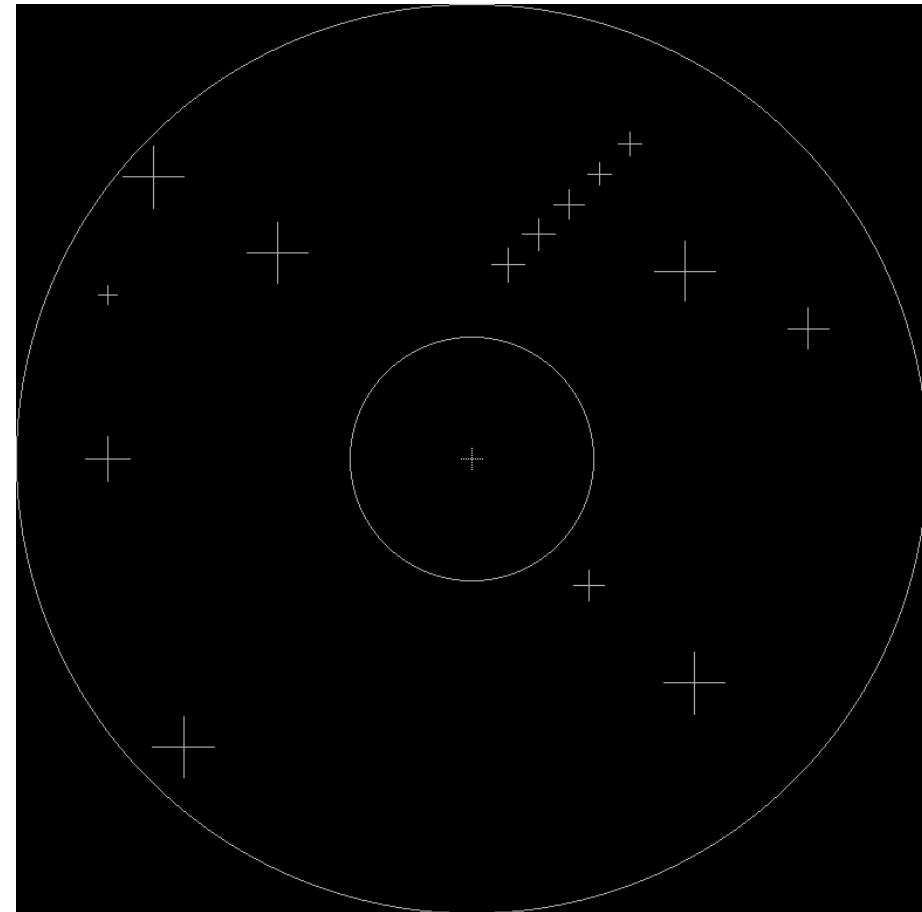# Top and bottom silkscreens
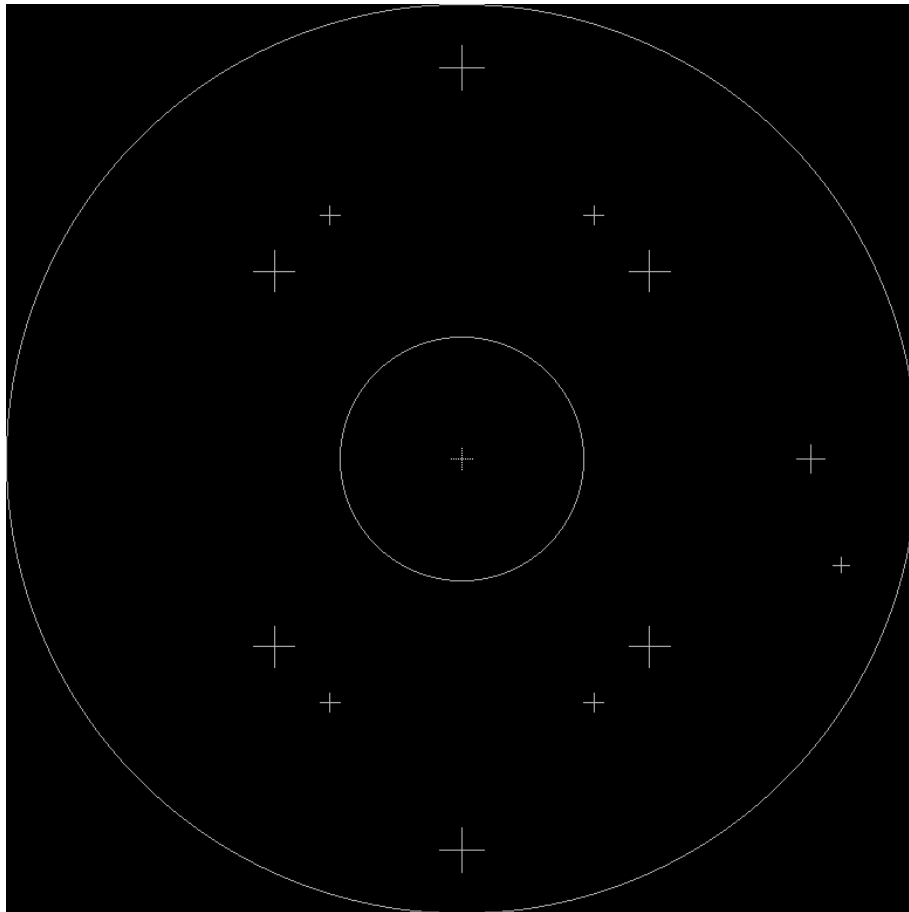
- Text and annotations (reversed on bottom)

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…
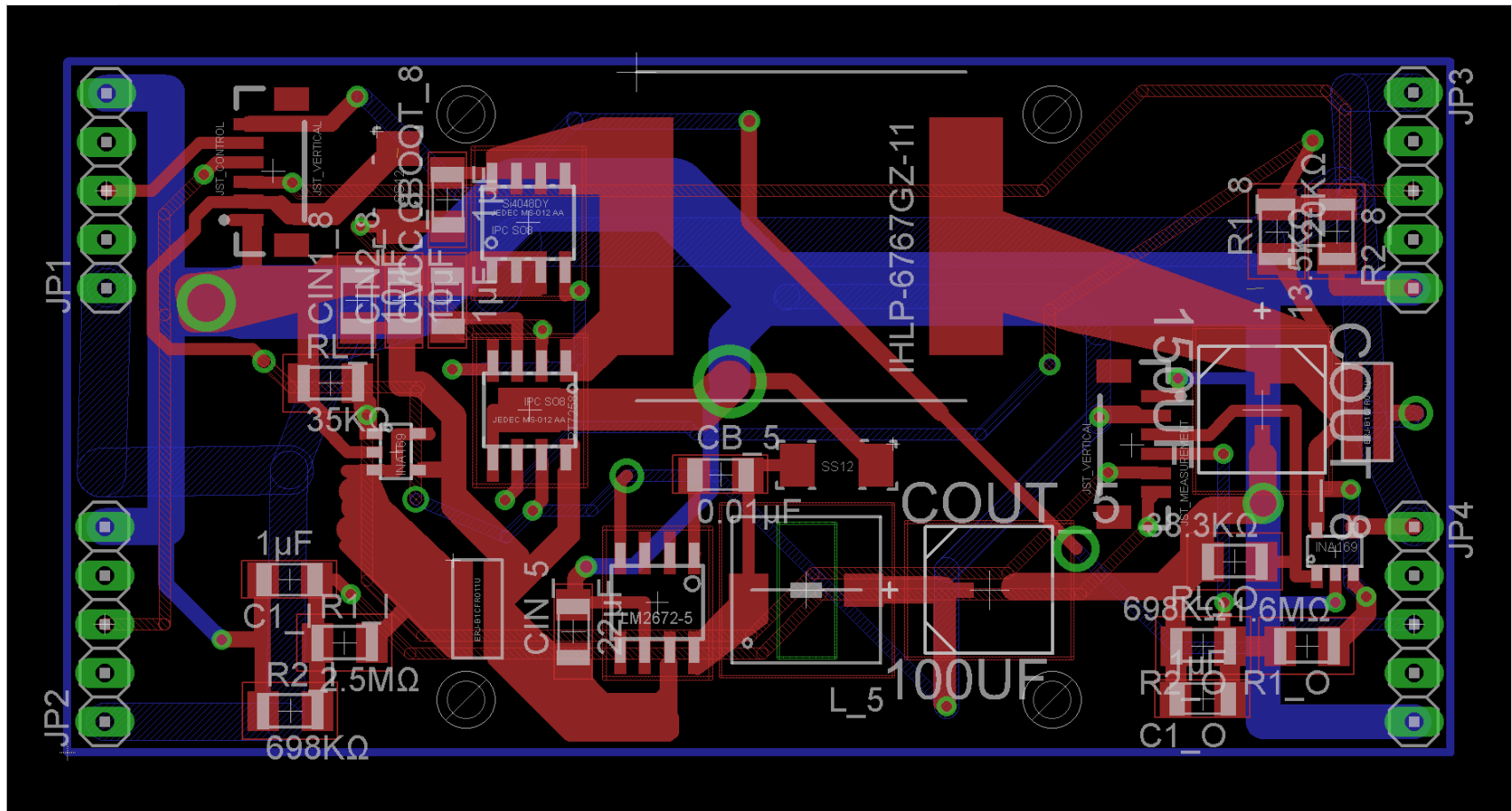
Lots of technical specification in a PCB!

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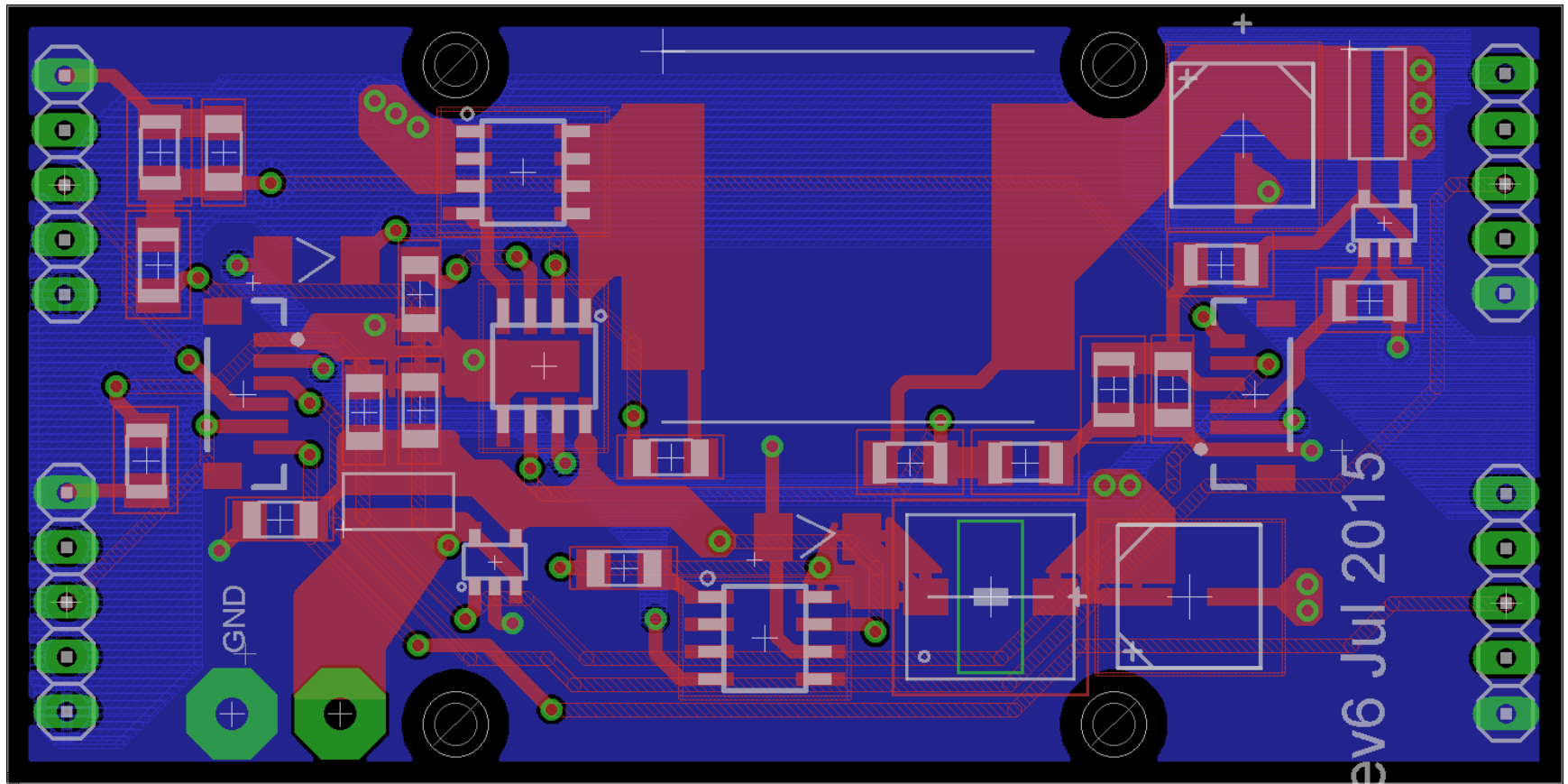board probably won't work.

Hey, that's great…

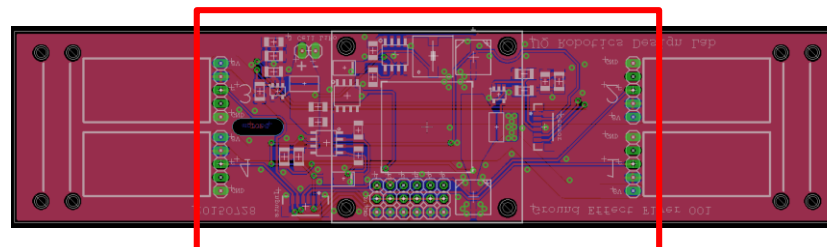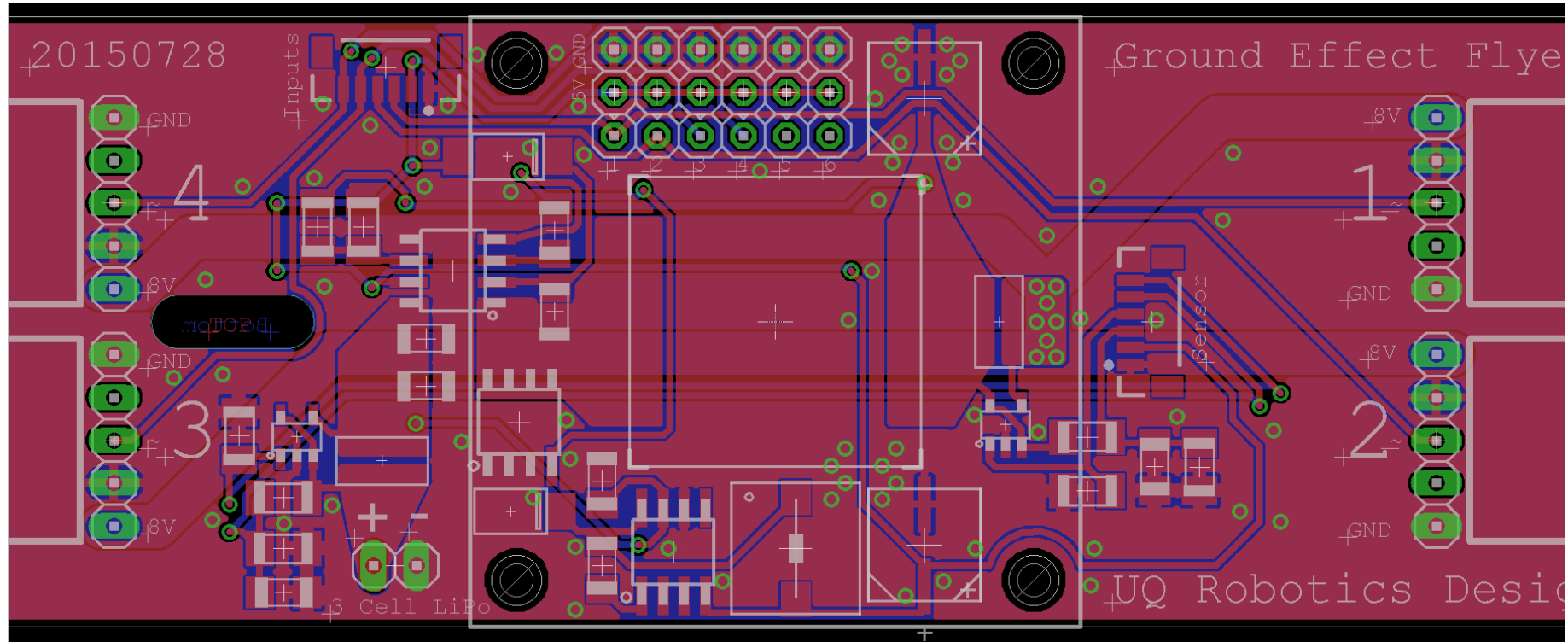So what about actually designing a PCB?

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
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
  - It is 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!
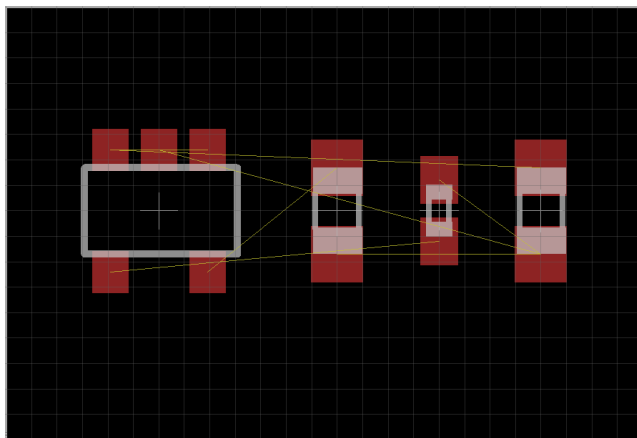
Default part layout – not helpful!   Oh hey, that's where that goes…
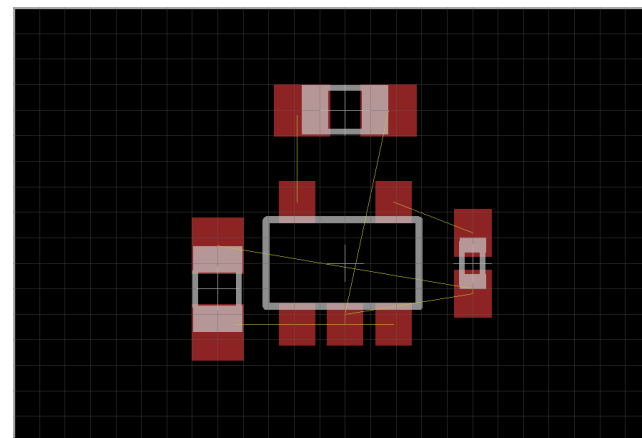
Routing tracks

- 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…
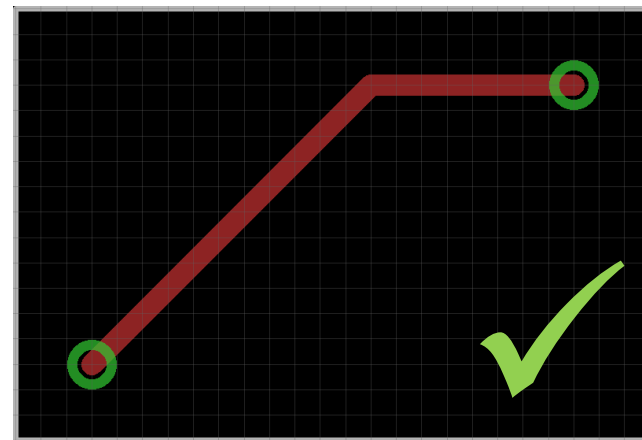
(The majority of my corrections to student PCBs consist of the following violations)

Routing tracks

- ## Adhere to the 45º 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
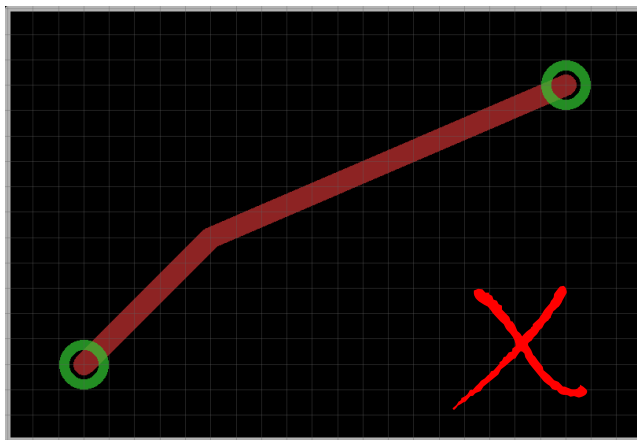
Looks sloppy and lazy   Looks like it's meant to be this way

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
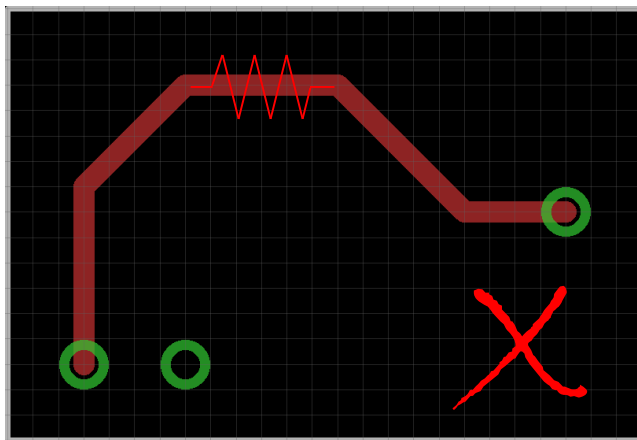
Circuitous circuit　　　　　　　　Short track better
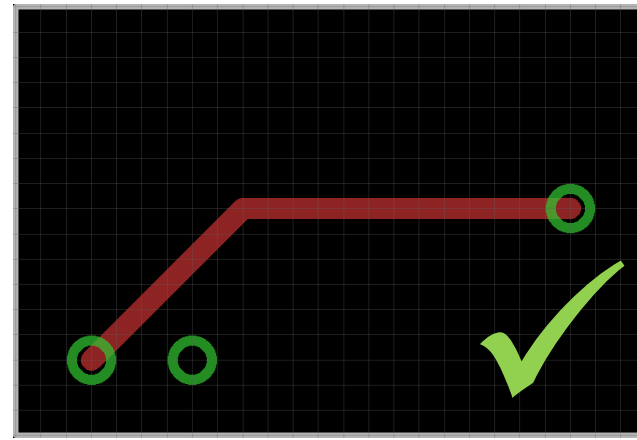
Routing tracks

- 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
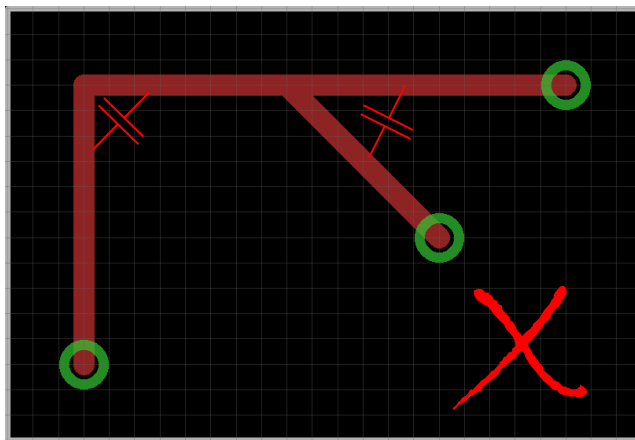
Right angle and acute corners   Obtuse corners best
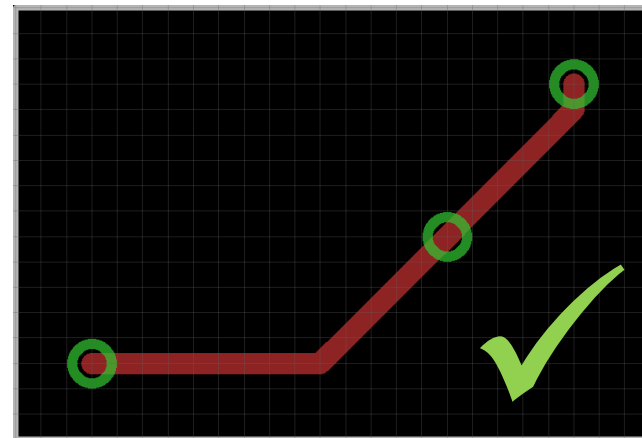
Routing tracks

- 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
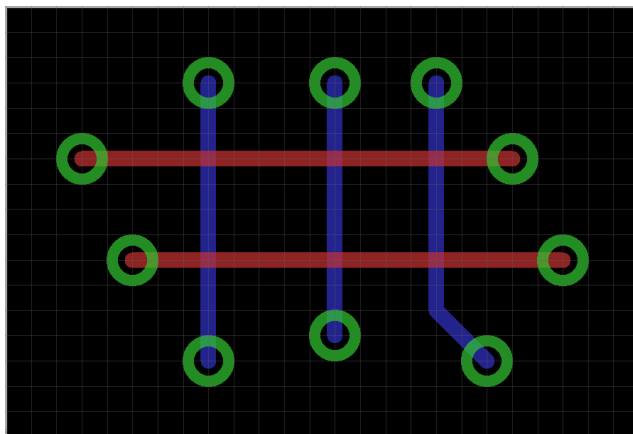
Different layer track directions   90° high-freq crossings
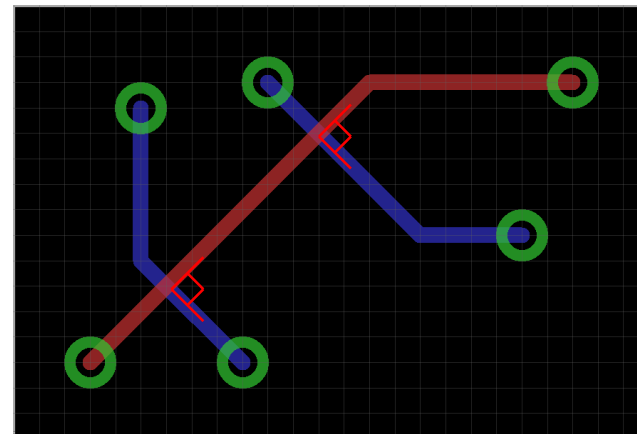
Routing tracks

- Enter surface mount pads cleanly
  - Right angles on edges or 45° through corners
  - Avoid acute break-outs from pad edges
  - Can't always be achieved in practice, though…
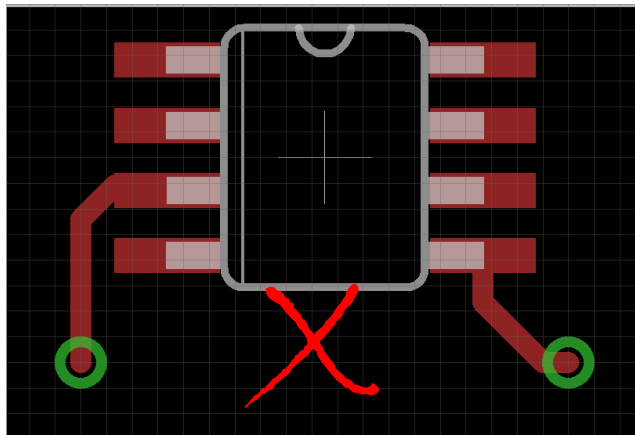
Crappy SMD pad exits   Nice SMD pad exits
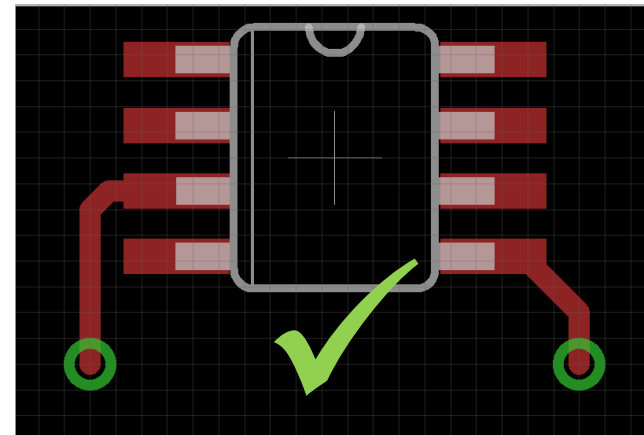
Routing tracks

- 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?
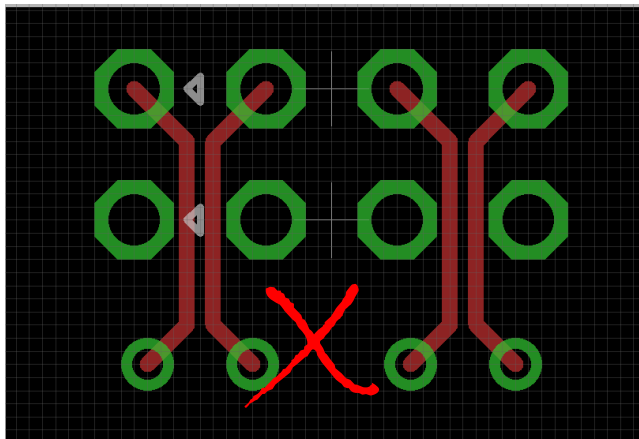
Avoid cramped buses Clearance is your friend
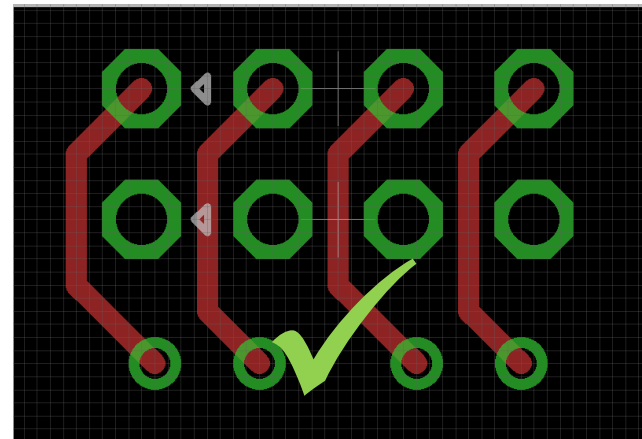
Routing tracks

- 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

- Veee-ahs or Vye-ahs?

- 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

Via sizes

- Don't use dozens of different drill sizes
  - Use the same size as often as possible
  - Economy of scale: "Two of one thing is cheaper than one of two things"
  - Tool changes cost money
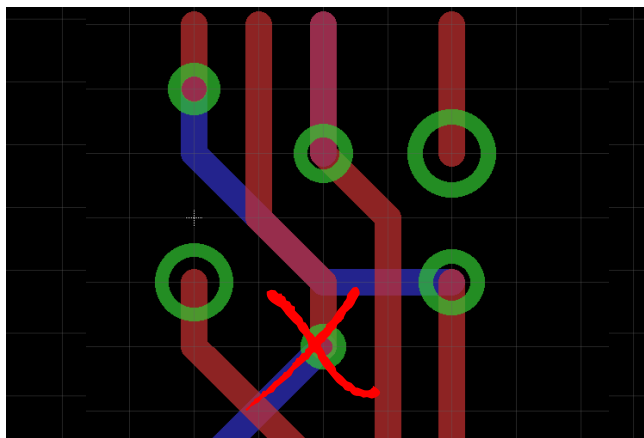
A million different sizes   One drill, no tool changes
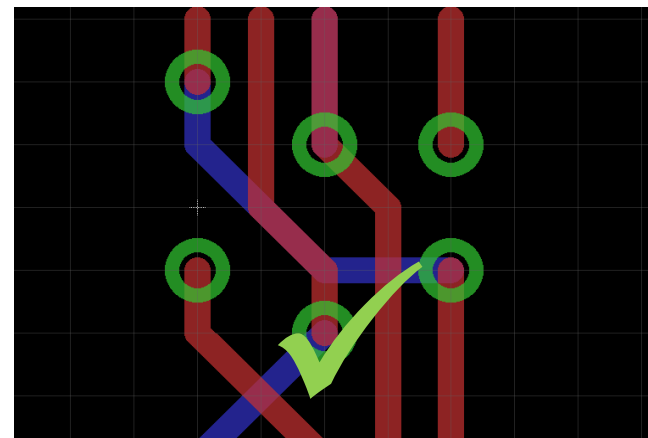
Via sizes

- Make the via restring proportionally sized
  - Restring: annulus of copper around the drill hole
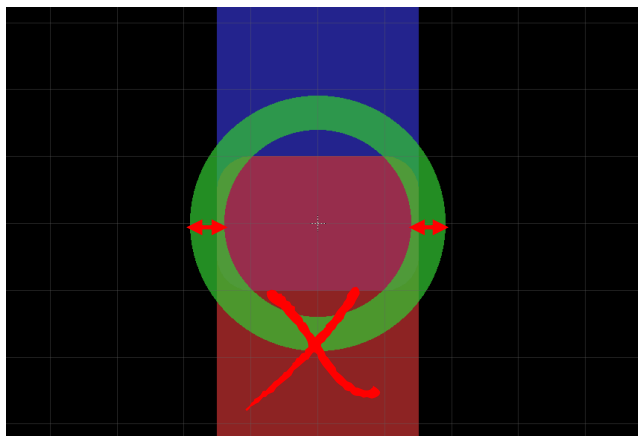- Allows misaligned drills to still have copper contact with plated area
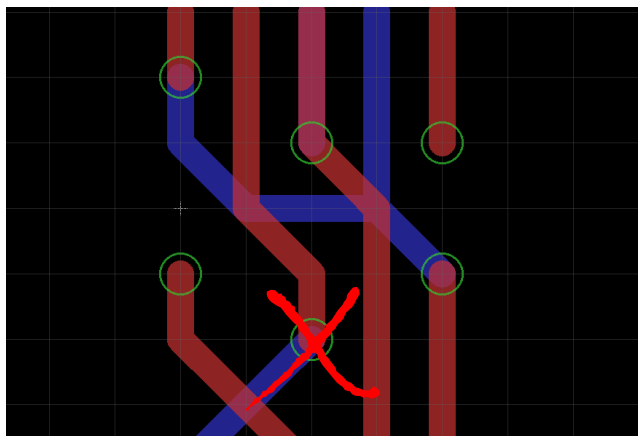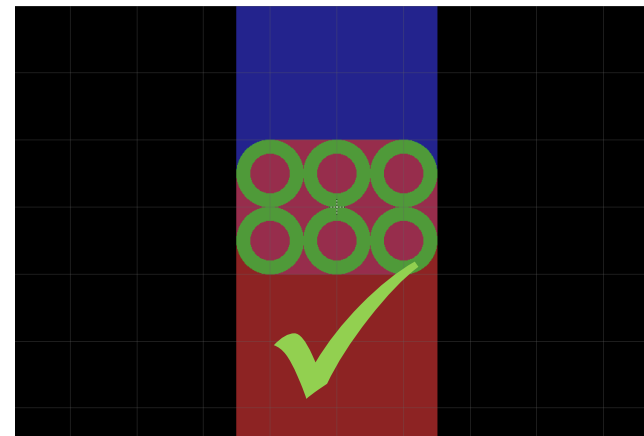

Too small: unreliable   Too large: bad thermals
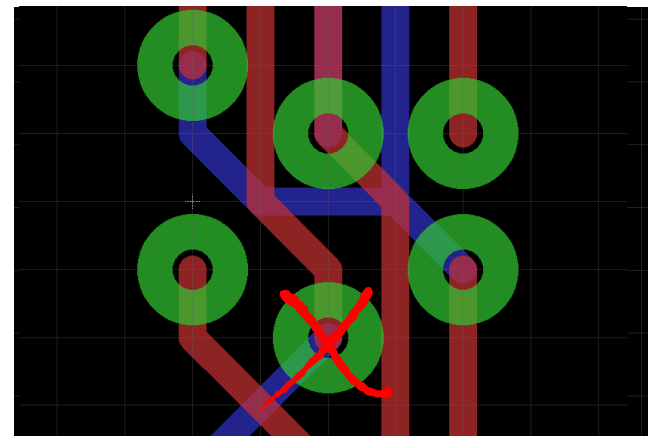
Via sizes

- Drilled out track section does not carry current across it

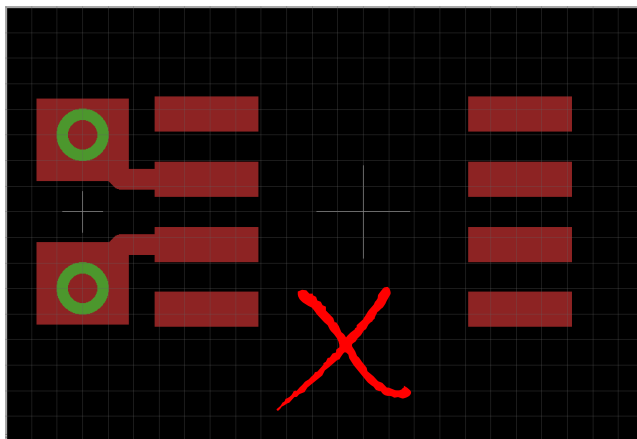- Many small vias give more copper and less resistance than one large via
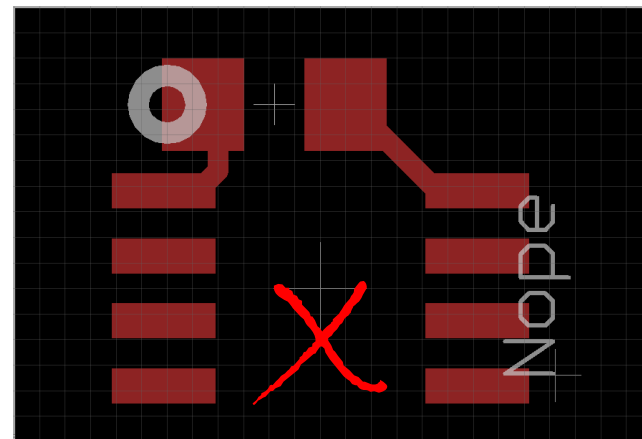
Little actual conductive surface   Lots of copper surface area

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

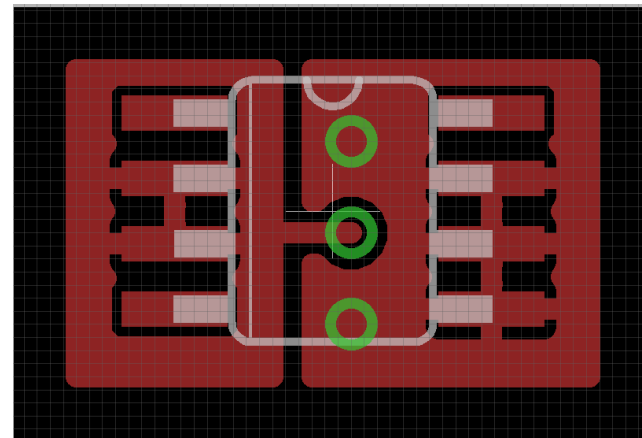# Zwischenspiel.

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
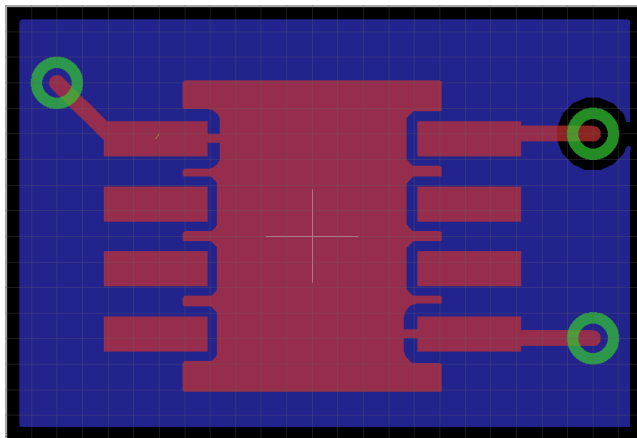 - Different signal values pours in different parts of the circuit
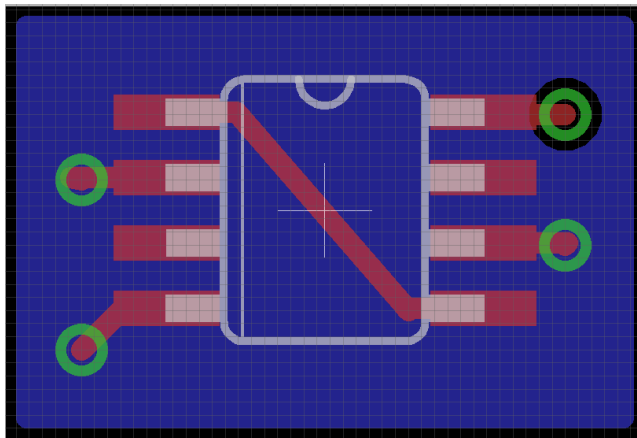
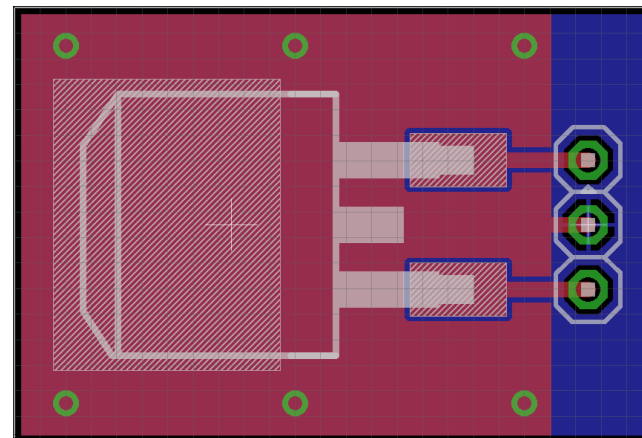Easy routing of backplane signals   Two different pour nets

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
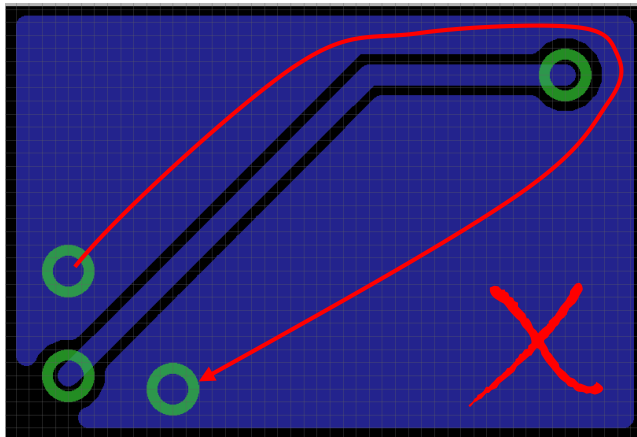
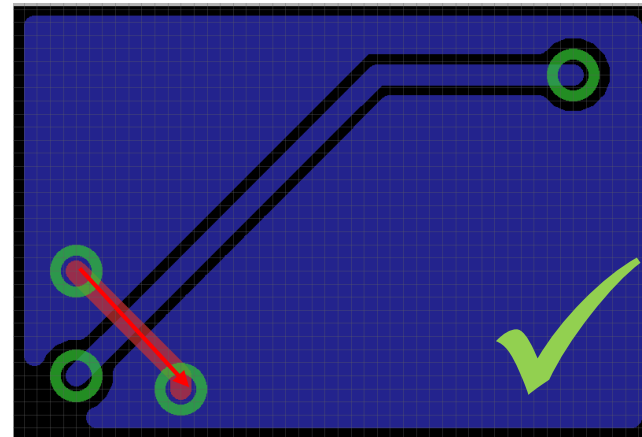IC mounted over ground plane　　　　Regulator with thermal pads

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
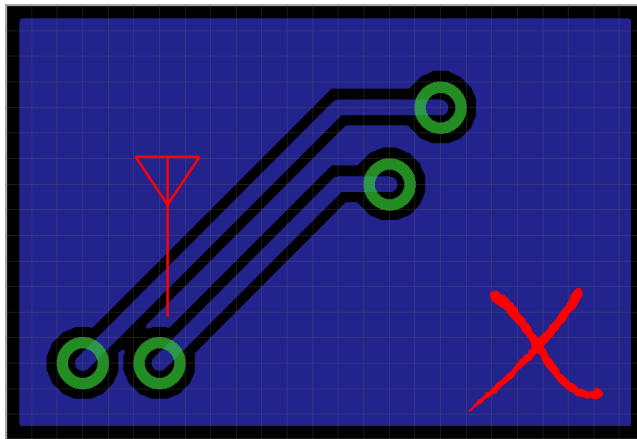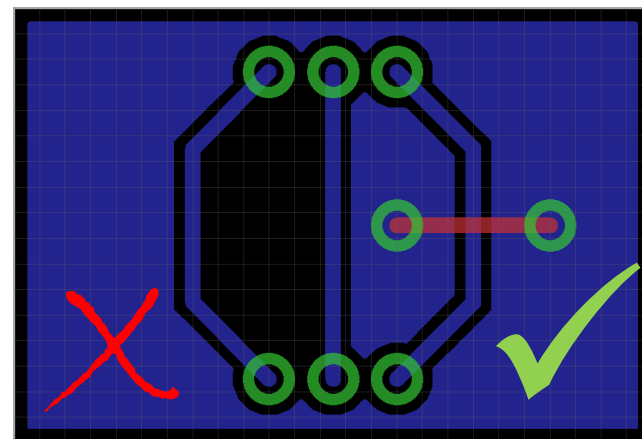
Long current detour "ground loop"   Shortcut 'bridge'

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!
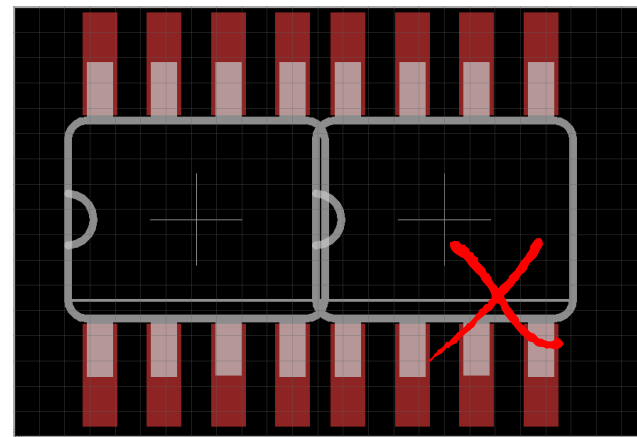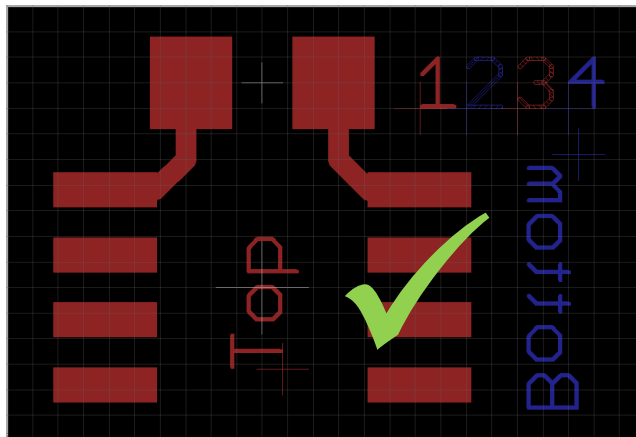- Bridge island fills if you can

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!
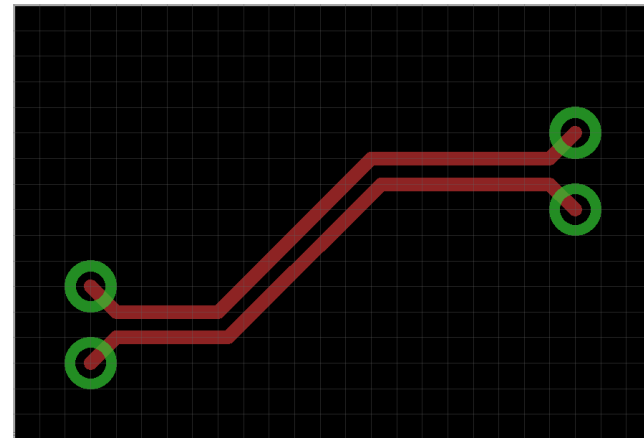
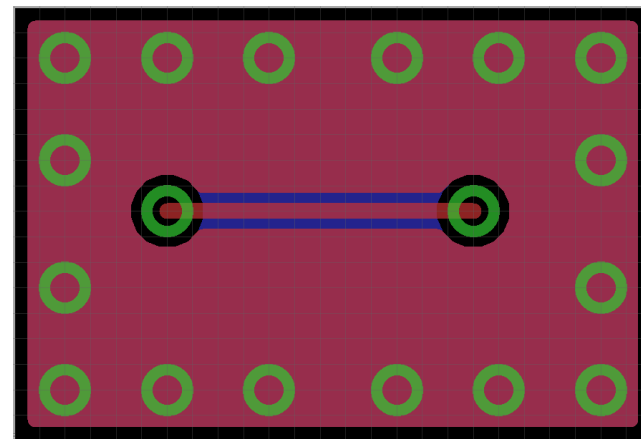Inductance 'wriggle'   Matched-length differential tracks

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
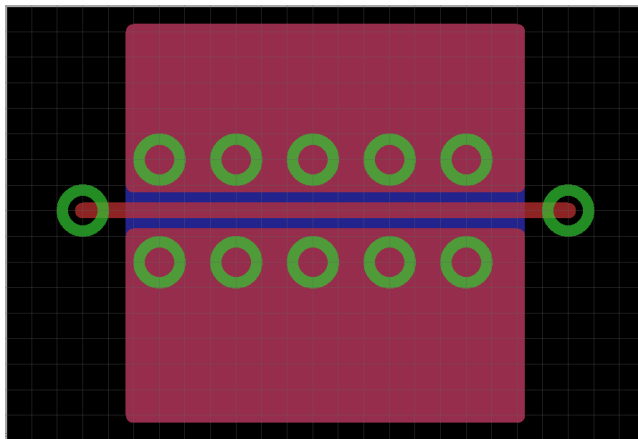
Stitched RF wave guide   EMI guard loop

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!
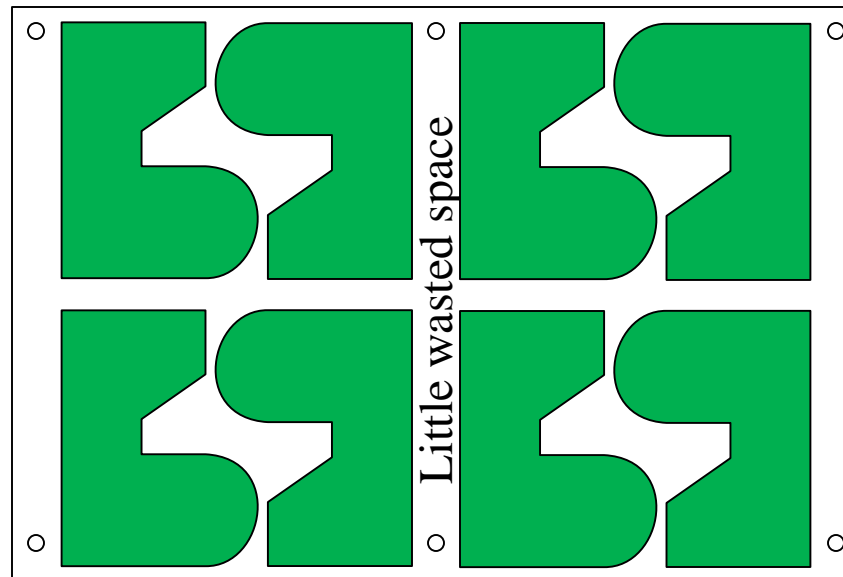 - Also, large/thin boards sag under their weight

Panelisation

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

91
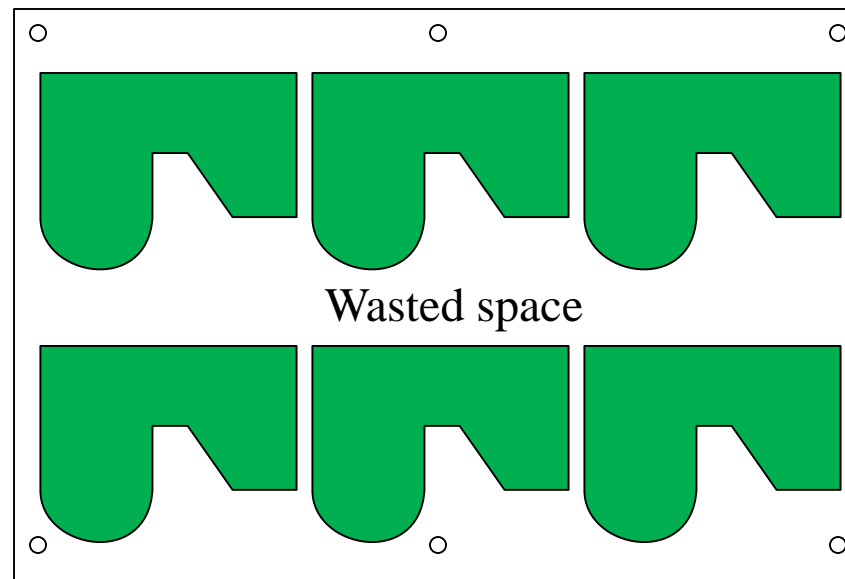
Panelisation

- 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$^{\text{o}}$ 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)
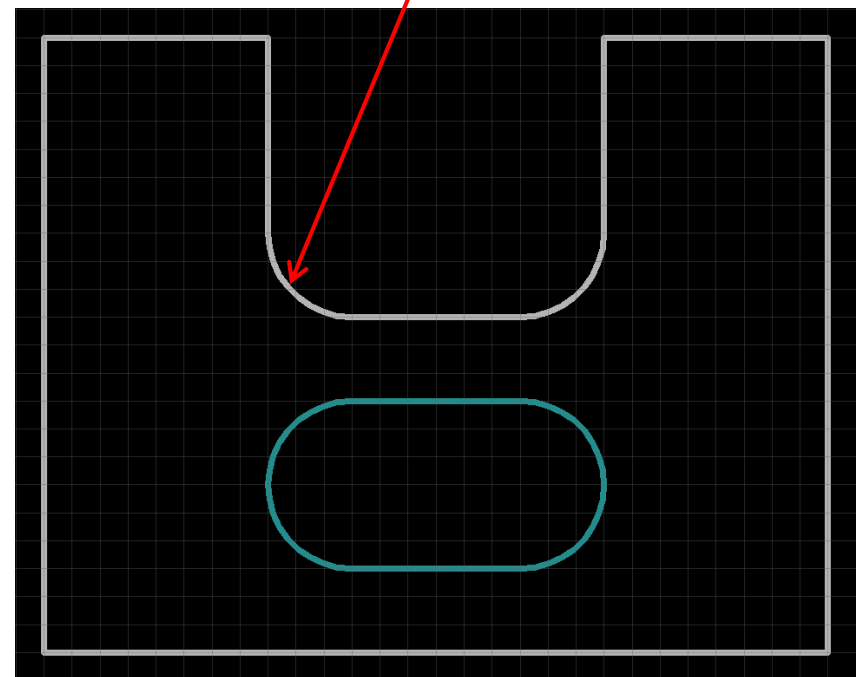
Milling

- If you do it wrong…

Convenient corner pocket method
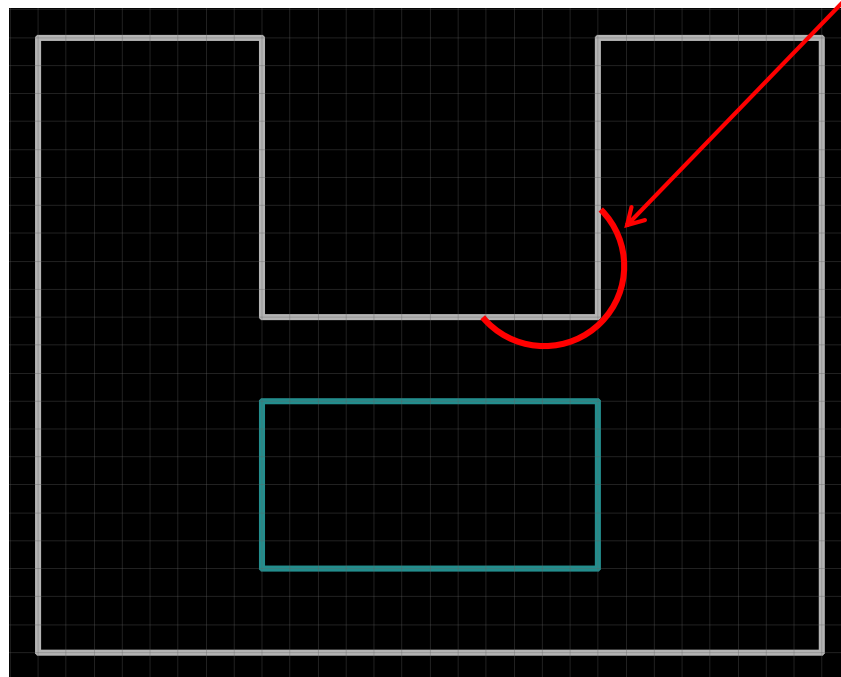
Coarse mill radius
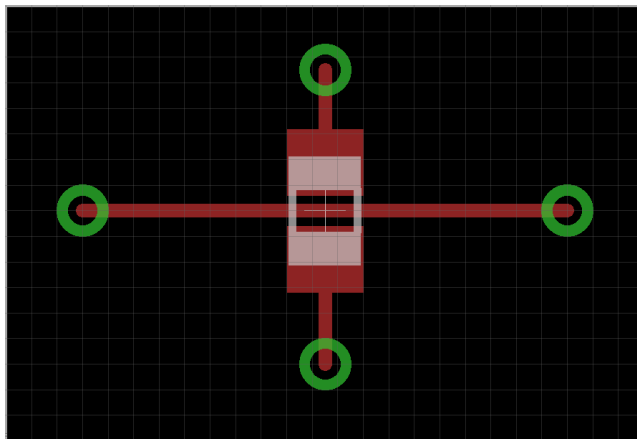
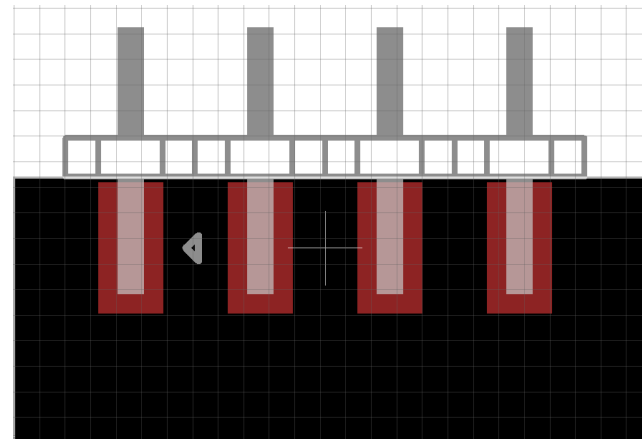What you ask for (unrealistic)   What you will actually get

Go for the biggest mill you can get away with

Cute tricks

- 0 Ohm resistor 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
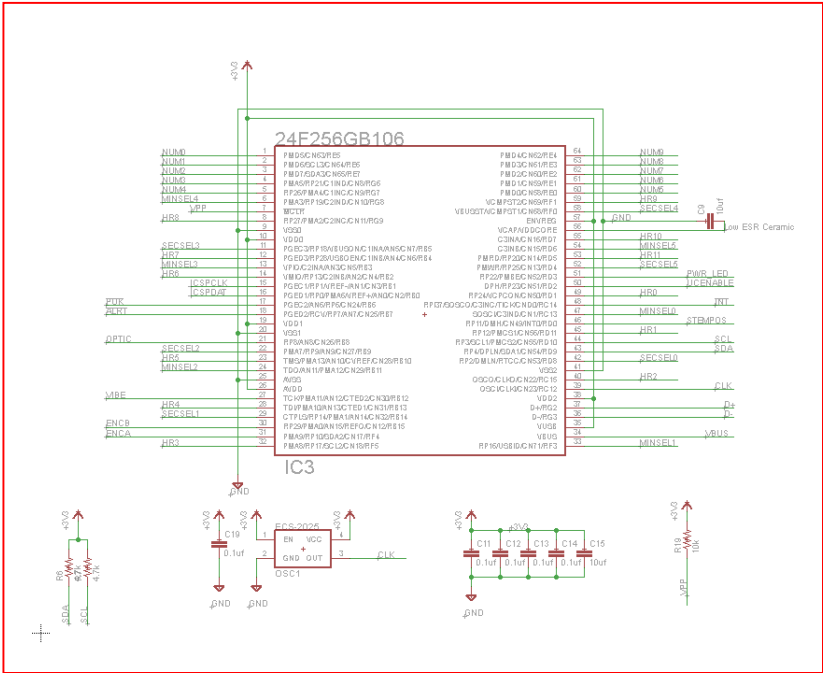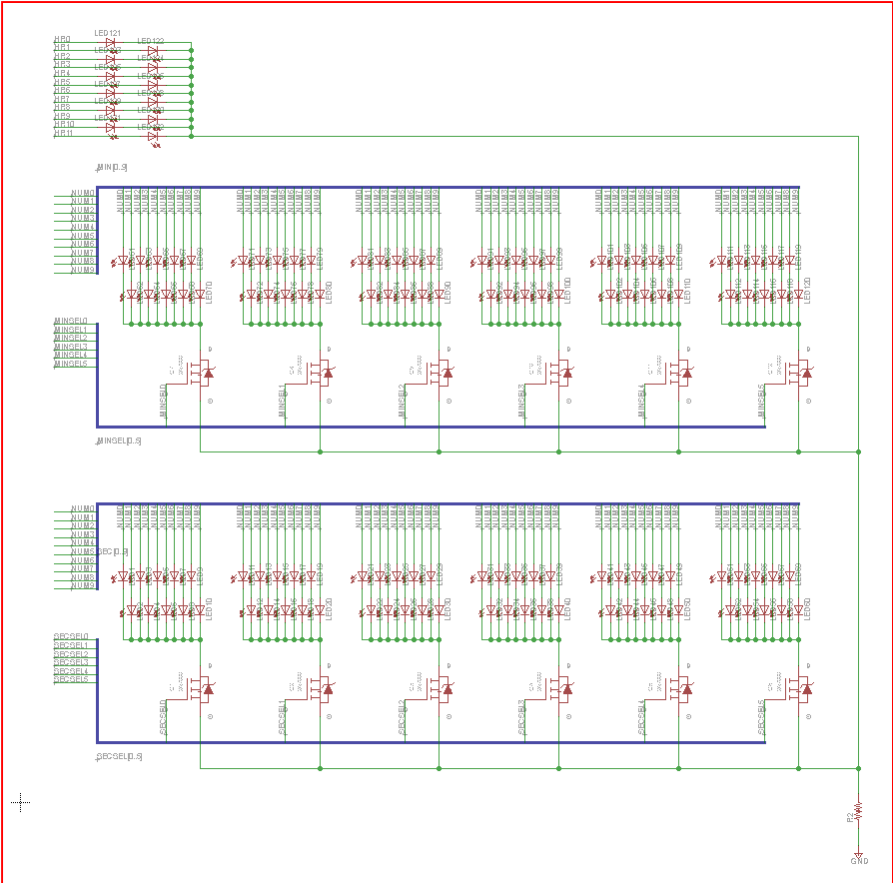 - One should complement the other

Some comments on schematics

- Logical flow left to right
- Voltages up to down

Information flow
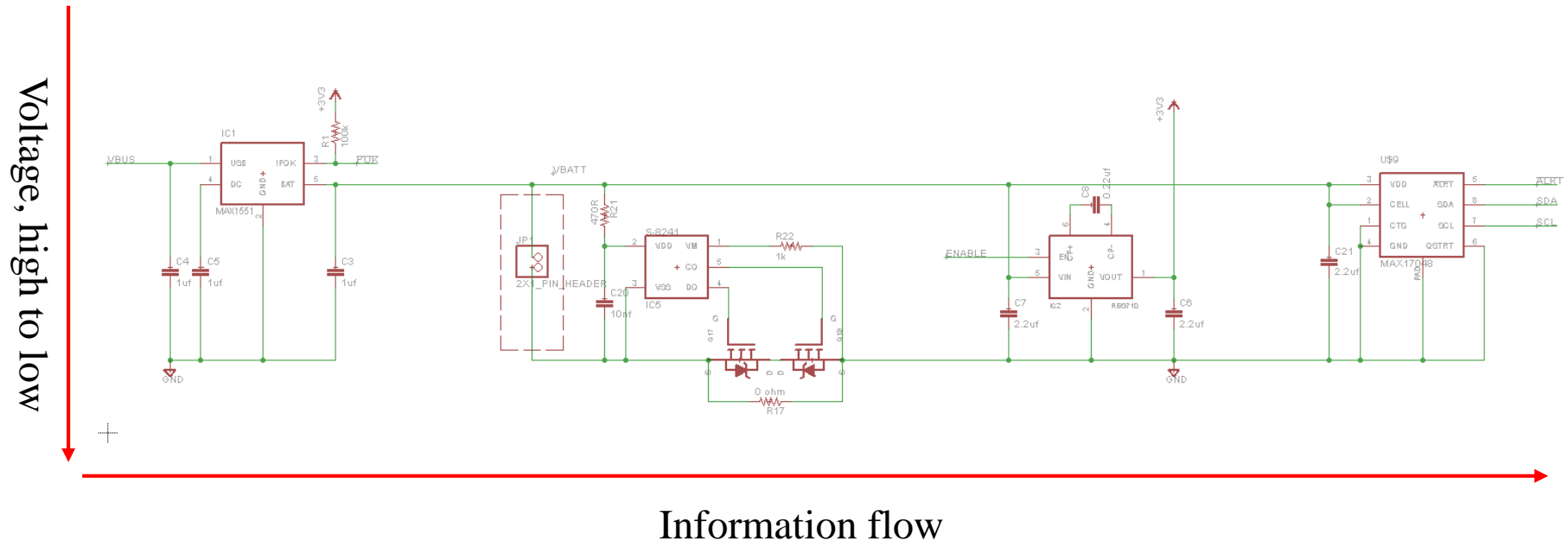
Some comments on schematics

- 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!
  - Make a list of all your mistakes over time so you don't keep repeating them!

- Open up high quality products and look at the PCBs inside – you can *see* the love

Whirlwind intro to CAD packages

- Lots of different ones
  - Most will get the job done;
  - But some are more useful/common than others

- At some point in your career, someone will insist that you use a CAD package you hate
  - You poor, poor soul

- Here is a quick list of common/uncommon PCB design CAD packages

Altium

- Weapon of choice for many professionals
  - Lots of powerful tools: 3D view, flexible fills, track impedance control, circuit duplication…
 - But gawdawful to learn its intricacies
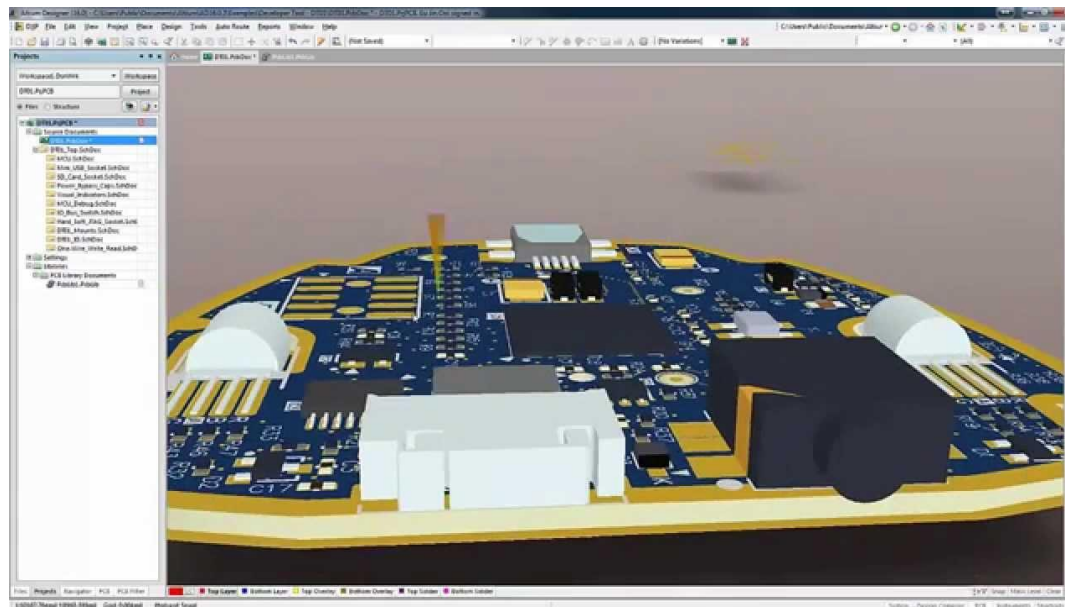
Cadsoft Eagle

- (Previously) the hobbiest's choice
  - Tons of library support and lots of references
  - Easy to learn and dive right in

    …but lacks ~~many~~ some of Altium's advanced features
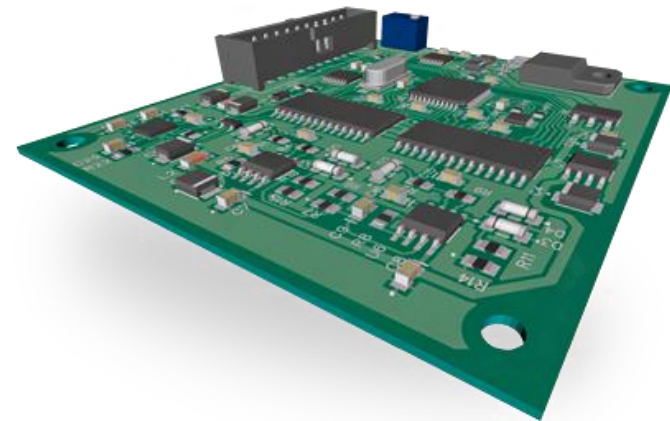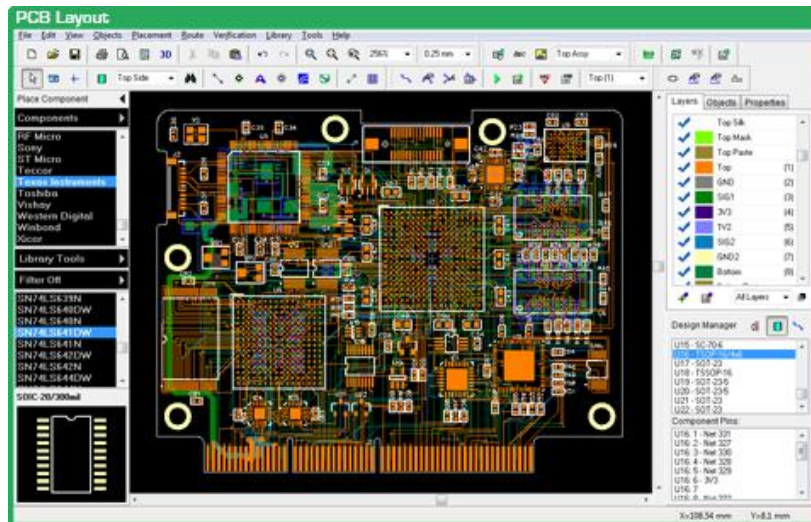 - Owned (ruined?) by Autocad; licensing sucks
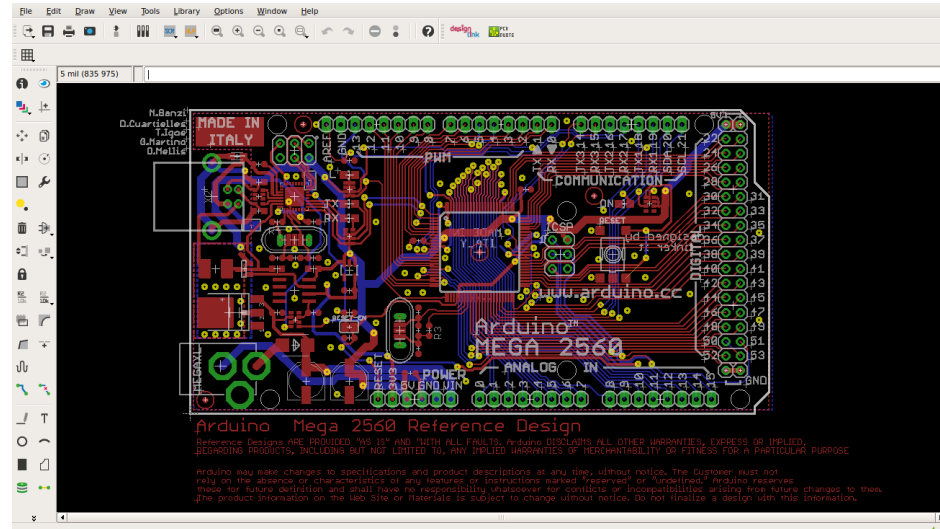
Kicad

- New contender for the hobbiest throne
  - Harder to use than Eagle but Free^{TM}
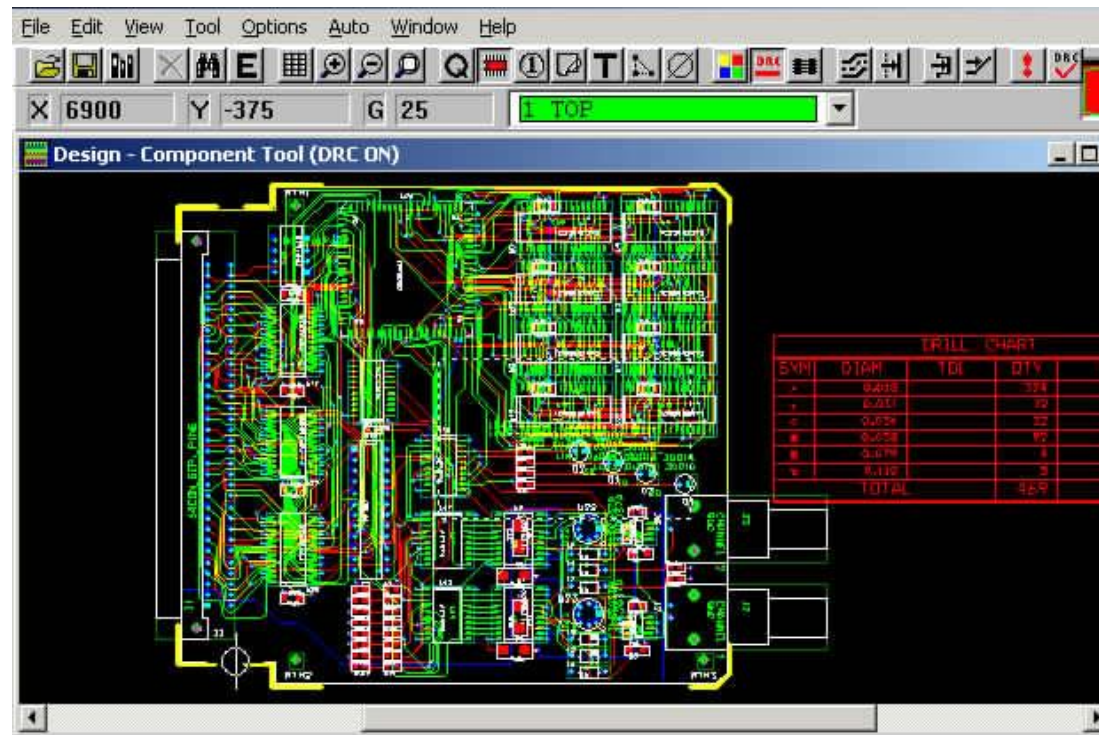 - Functional alternative – watch this space
  - Rhymes with "eye sad" ☹
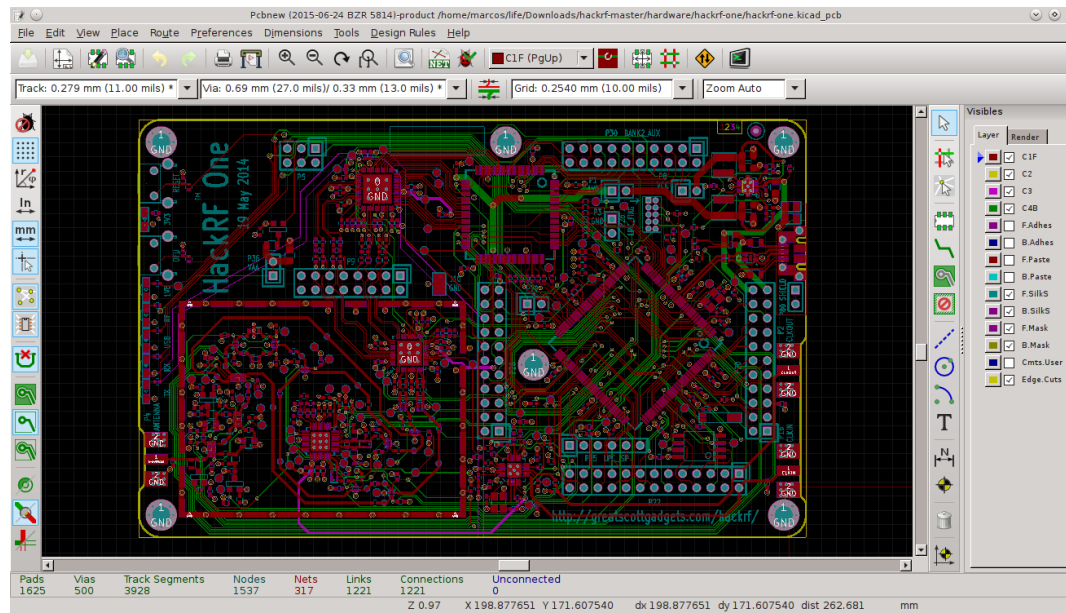
Diptrace

- Not as popular as the others, but strong support by its adherents
  - Rapidly growing new advanced features…
  - Maybe a replacement for Eagle?

OrCAD

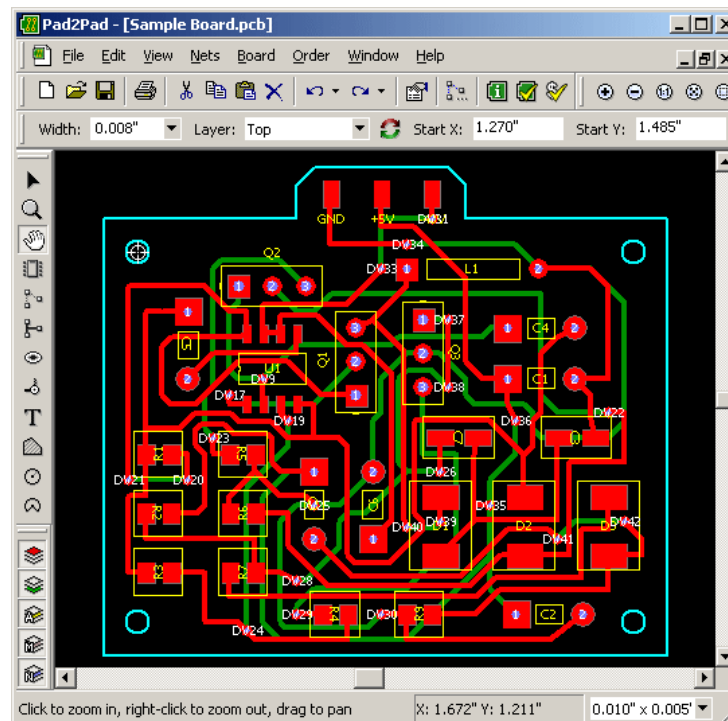- Somewhat old now
  - Not a lot of userbase, but still exists

105

Pad2pad

- Online integrated vendor-specific software
  - A lot like eMachineShop, if you know what it is
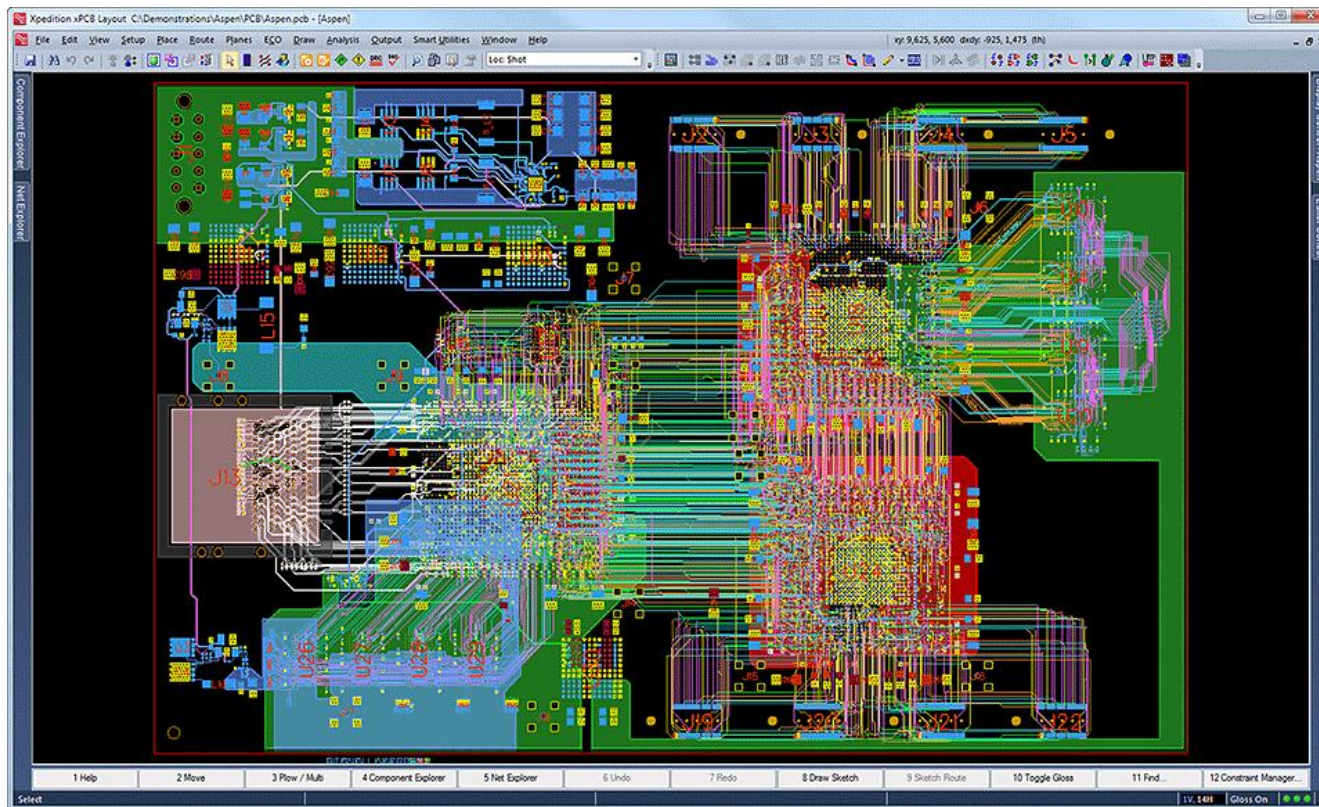 - Ehh…. Ok – but vendor lock-in sucks

Protel

- ## What eventually became Altium
  - ### Your grand dad's PCB CAD tool…
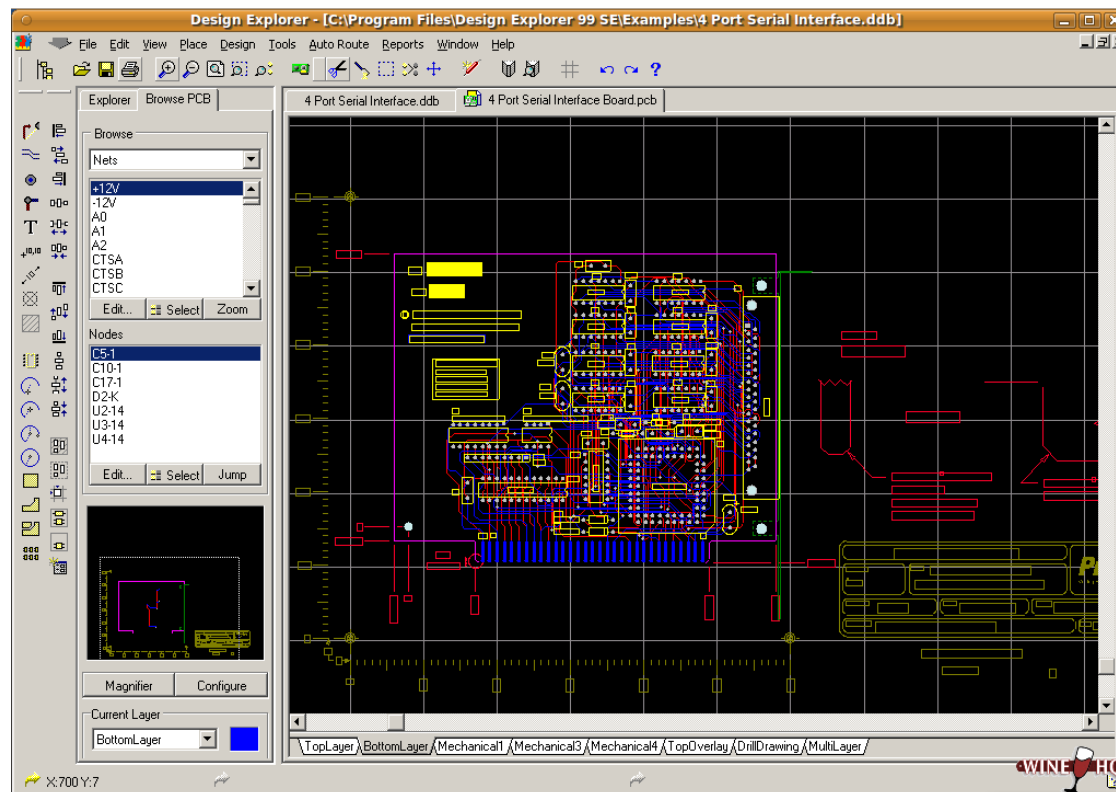
Mentor Graphics

- Ancient, but very capable package
  - What they used to make motherboards with

gSchem, gEDA/PCB

- ## The classic Linux packages
  - – Nobody without a neckbeard uses either of these anymore
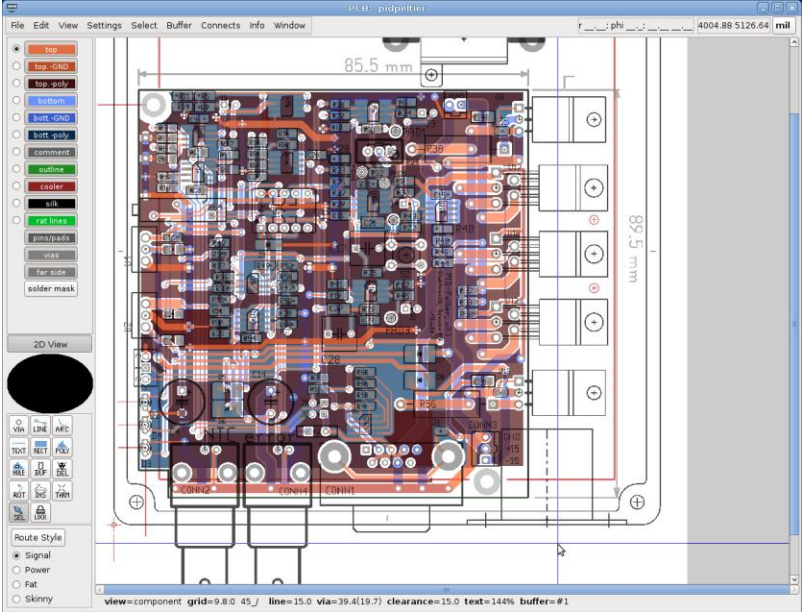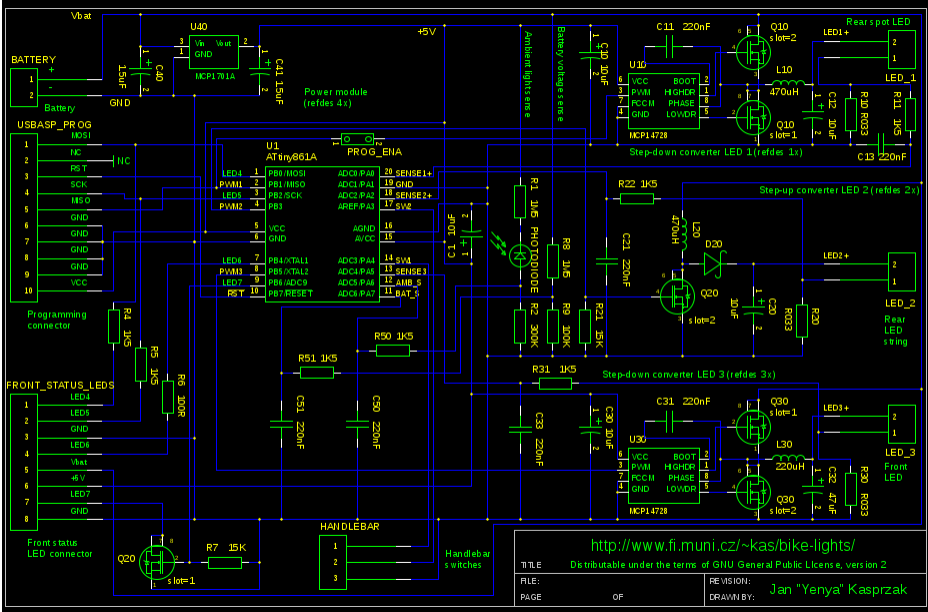
And others besides!

- Seems a new CAD program comes out every month
  - Many are terrible, some less so
- No one sits on the PCB throne as supreme reigning god-emperor of circuit CAD
  - Crippled by irrational business decisions, free software malaise, or geriatric irrelevancy.

Circuit Fab Houses

- Fabrication houses are a bit like artisans
  - Huge range from hacks to master craftspeople
  - The same files and specs sent to a dozen fab houses will produce a dozen different outputs of varying quality

- Finding a manufacturer you –trust– is very important to your career as a PCB designer

Top of the heap

- The high-tech houses
  - *VERY* expensive, but results are always excellent
  - Often cater to defence/aerospace/spooks clients
  - Do ALL their own fabrication in-house and never, ever outsource

- Examples:
  - Lintek (Queanbeyan/Canberra)
  - Minco

Quality Professionals

- Have a real 'Name' that they defend
  - Usually do their own stuff (often assemble too!) but sometimes outsource to China
  - *But*, they care about quality and confidentiality, so usually very reliable and litigatable.
  - Range from quite small to HUGEMONGOUS

- Examples
  - Entech (Melbourne), BEC (Brisbane)
  - Foxconn

Lower tier professionals

- "Average" quality and reliability
  - Liberal in interpreting your specs to save money
  - More variability and cost-cutting than you'd really like – cheaper materials, finishes
  - Very often outsource to cheaper fabs overseas
  - Confidentiality problems not unknown – see your design on the streets of Hong Kong! ☺
- Examples
  - None I'll mention in print!

Low-end hobbyist grade

- Focus on cheap and quick
  - Typically not the highest precision/performance, but can't beat the price!
  - Often excellent web-portal integration
  - Sometimes eccentric design limitations?

- Examples
  - Circuit Labs
  - OSH Park (nice online preview tools!)
  - Many others

Integrated fab/assembly houses

- Upload your design plus BOM and have it fabbed and populated all at the same place!
  - And at an affordable price! OMG
  - Only limited range of capabilities for now, but this is going to change the industry

- Examples
  - Macrofab – seriously, check these guys out!
   (Depressingly, their recent UX interface upgrade sucks)

Useful references

- "The Circuit Designer's Companion", Tim Williams

- "Practical Electronics for Inventors", Scherz and Monk

- Also check out:
  - sparkfun.com
  - pololu.com
  - adafruit.com
  - hackaday.com
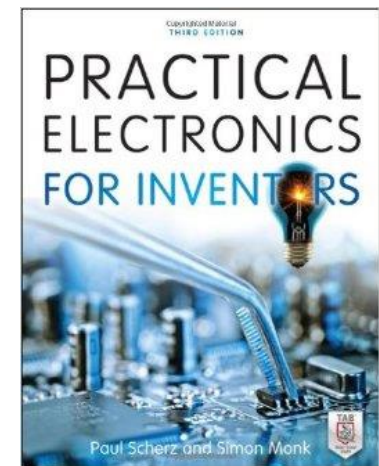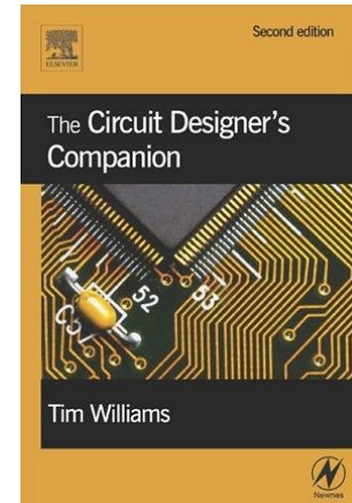
Moral of the story

- There is SO much to learn about being a good electronics designer/engineer

- The best way to learn is to do it, and it's never been cheaper!
  - $740 in 2007 vs $15 in 2017!

- Learn the tools and give it a shot
  - This is what I do for fun! ☺

Questions?
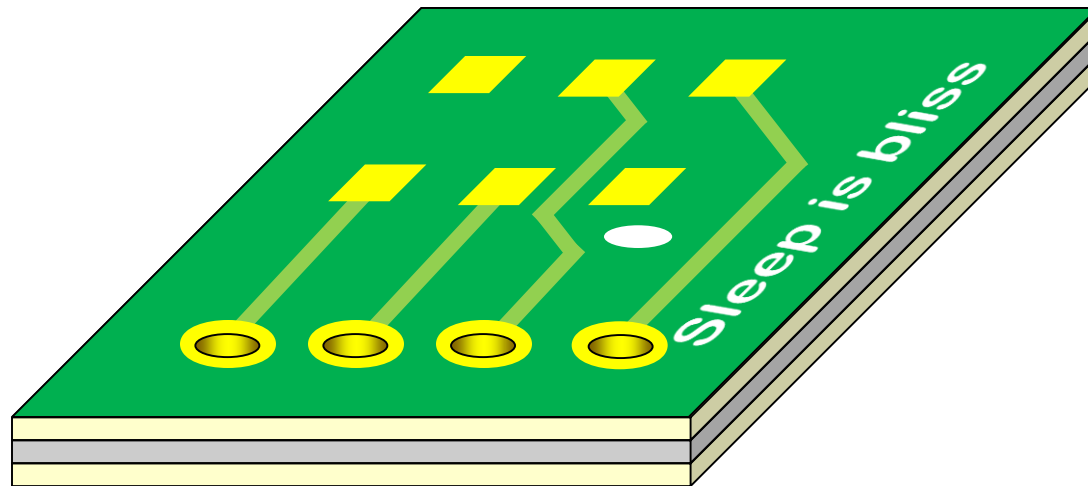
Tune-in next time for…

Your Soldering is Terrible (probably)

or

"How I learned to stop worrying and love the flux"

Fun fact: 98% of humans have parasites.