CC2511 Week 2: Lecture 1

PCB Design

Review of last week

- Microprocessors are small, low-power and low-cost computer systems intended to drive embedded systems.
- CPUs have machine code instructions for tasks such as:
  - loading values from memory into registers,
  - performing arithmetic and logic on these values, and
  - then storing values from registers back into memory.
- Control registers for peripheral hardware are memory mapped (i.e. they appear in a special region of the memory layout).

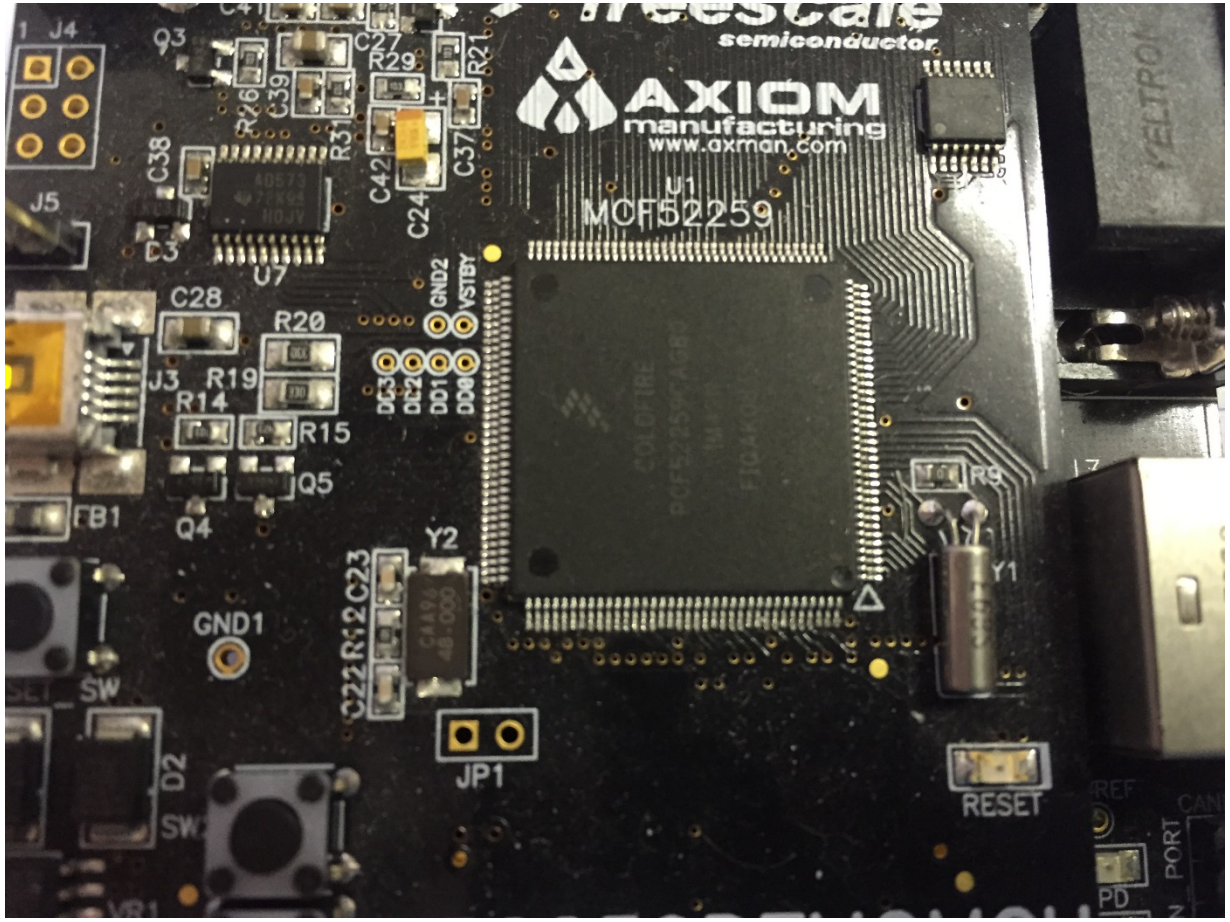
Today: Printed Circuit Boards

- The “what” and “why” of PCBs
- Computer assisted design software
- Design tips

What is a printed circuit board?

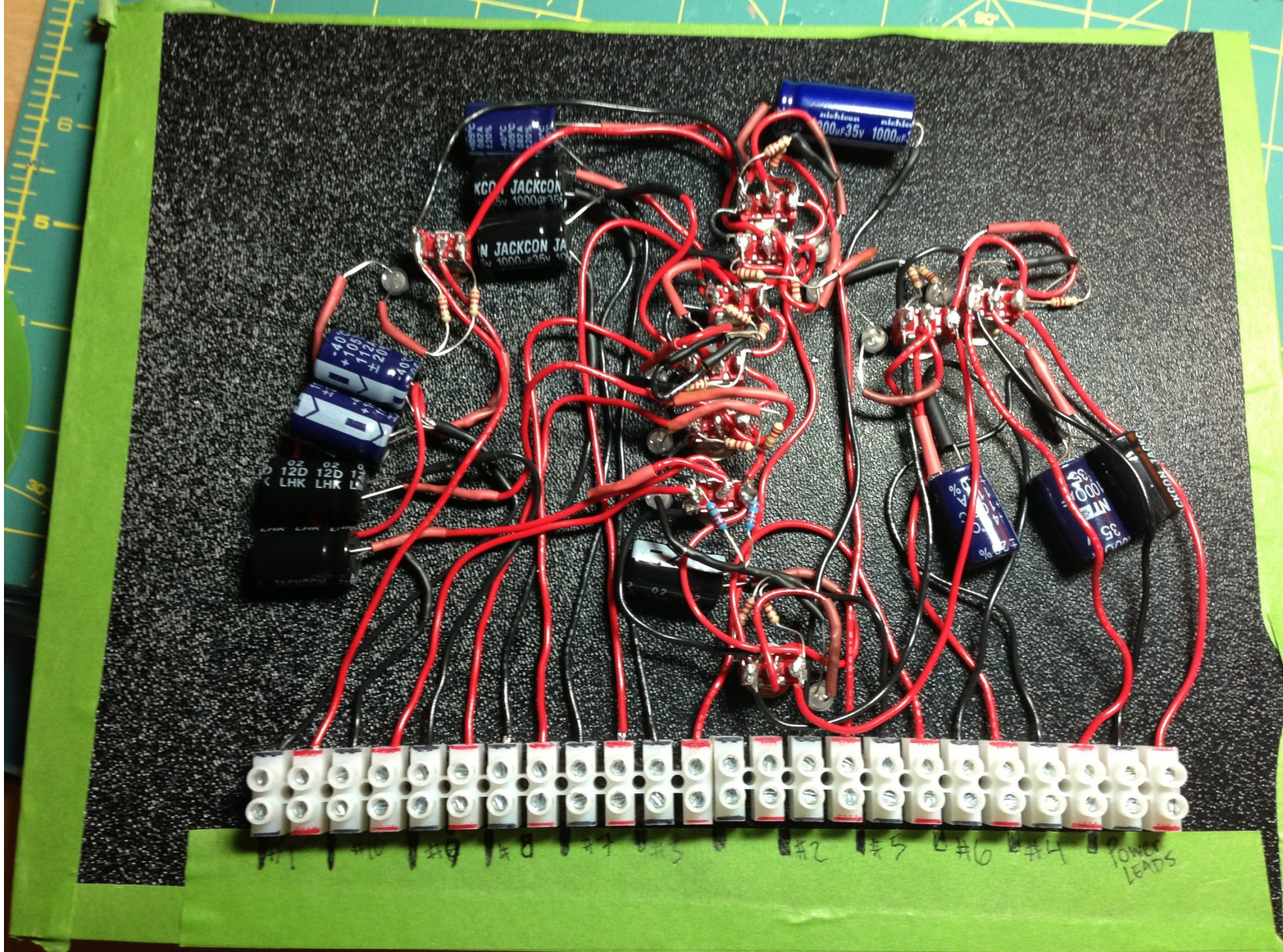
A printed circuit board (PCB):

- Implements electrical connections between components, and
- Physically holds components in place.



Why?

So that you don’t have to do this!



PCB Substrates

- Most PCBs are made with “FR-4” substrates. This is made of woven fibreglass cloths with a flame retardant epoxy resin.
- FR-4 is an excellent insulator and mechanically robust. It’s a good general purpose substrate.
- Typical substrate thicknesses are in the range of 1.2 mm – 1.6 mm. 1.6mm is standard.
- Specialist materials with precise dielectric properties are used for high frequency applications such as radio electronics.

Copper layer

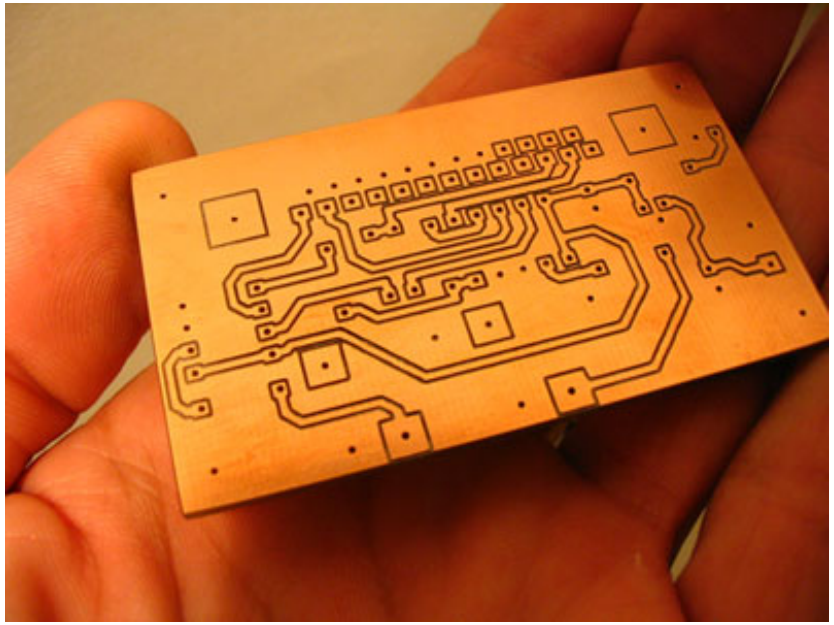
- The substrate is coated with a thin layer of copper.
- The usual thickness is 1 oz copper/ft^2, common pronounced as “1 ounce copper”.
- In metric units, 1 oz/ft^2 is equivalent to 35 um thickness.
- Thicker copper layers are available if electrical resistance is a problem. 2 oz copper is common.

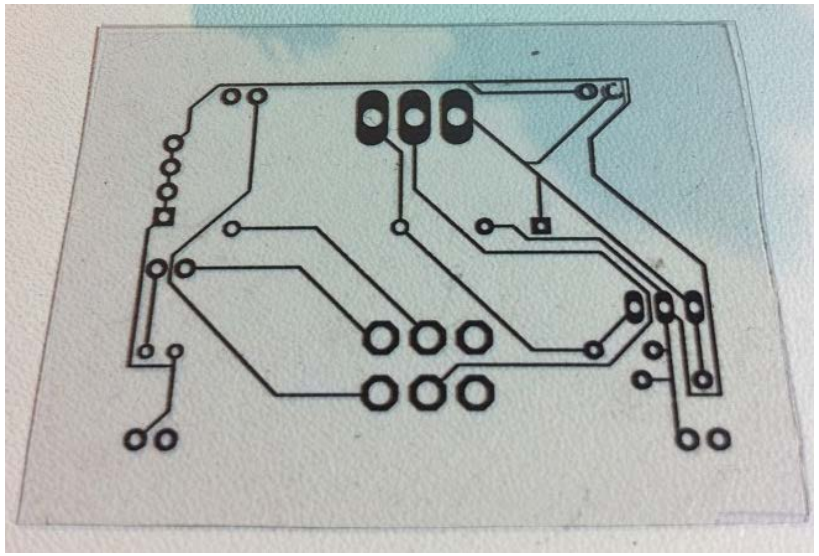
AE16, Double-Sided Plain Copper Ink Resist Board FR4 with 35µm Copper Thick, 100 x 160 x 1.6mm

RS Stock No. **219-2139**
Manufacturer CIF
Mfr. Part No. **AE16**
Page 713

Photo-etching

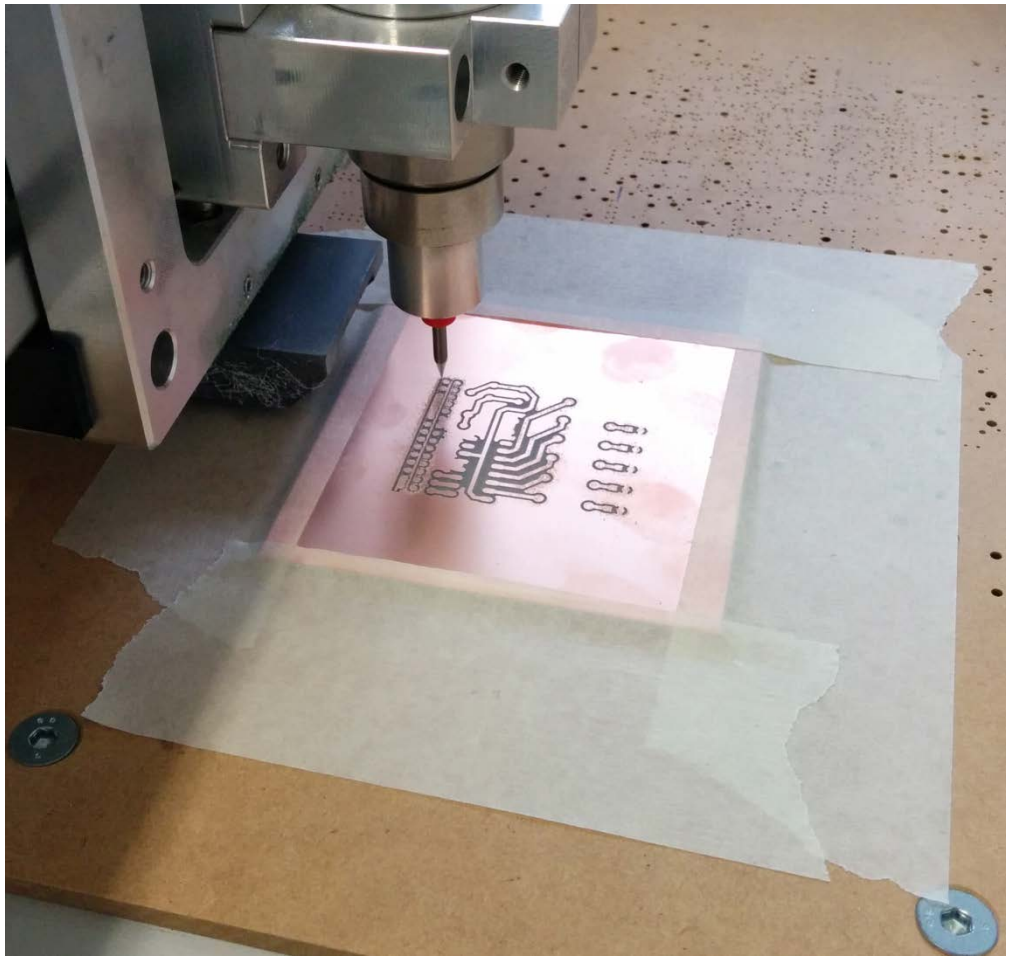
- The raw copper needs to be patterned to produce the PCB (top right).
- **Etching** processes use a chemical solution to dissolve unwanted copper.
- A transparent mask is used to control which regions of a photosensitive layer are exposed to light.
- The solution dissolves copper in the areas not exposed (or exposed, depending upon the chemistry in use).





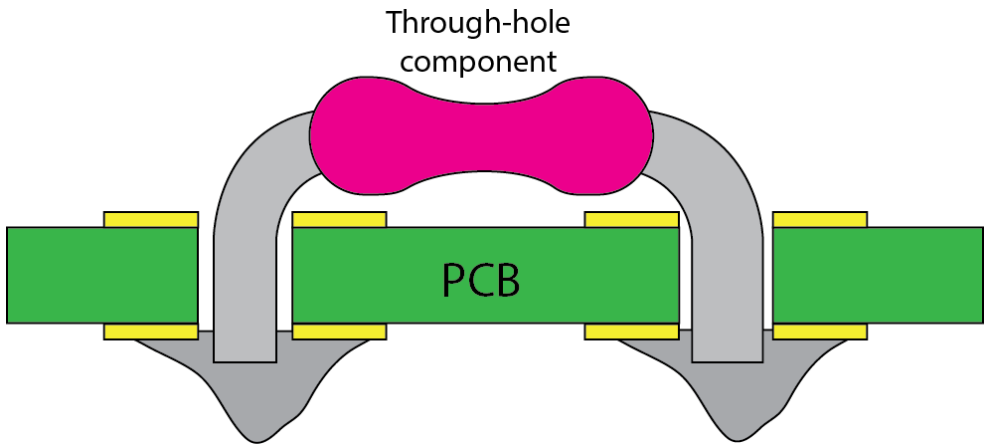
Milling

- An alternative to photo-etching is to cut away the copper with a milling machine.
- The photo is JCU's PCB fabrication machine.
- Milling is less precise than photo-etching but simpler (if you have access to a suitable machine).



Drilling

- The next step is to drill holes in the board.
- The holes enable **through-hole components** to be soldered.
- They also connect top and bottom layers (a feature called a "**via**").



Finishing steps (1/4)

Commercial fabrication will apply additional steps:

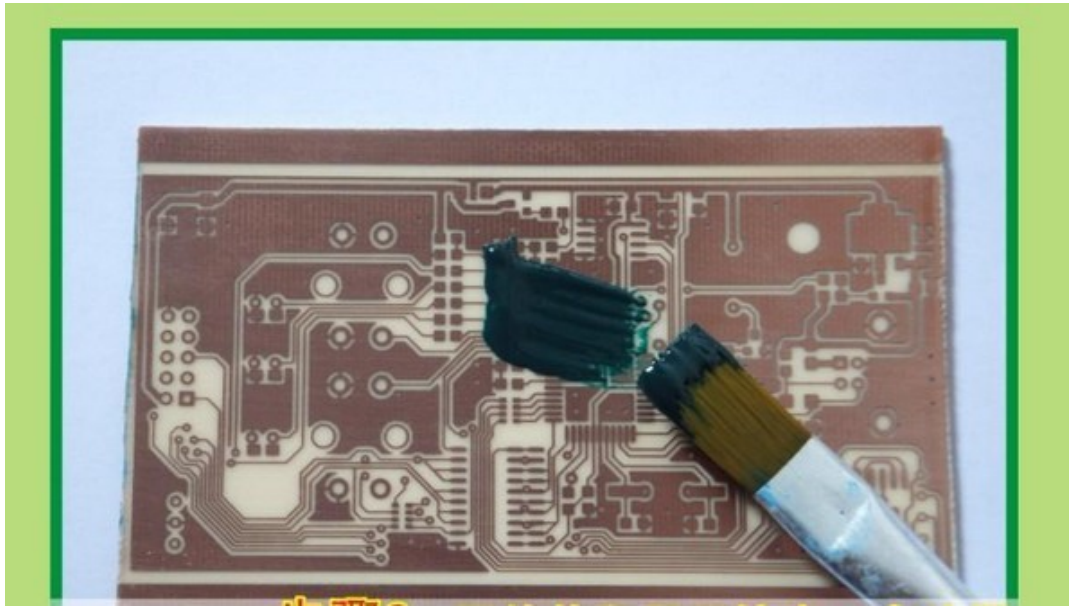
1. Through-hole plating, where the inner diameter of the holes are coated with metal using an electroplating process.

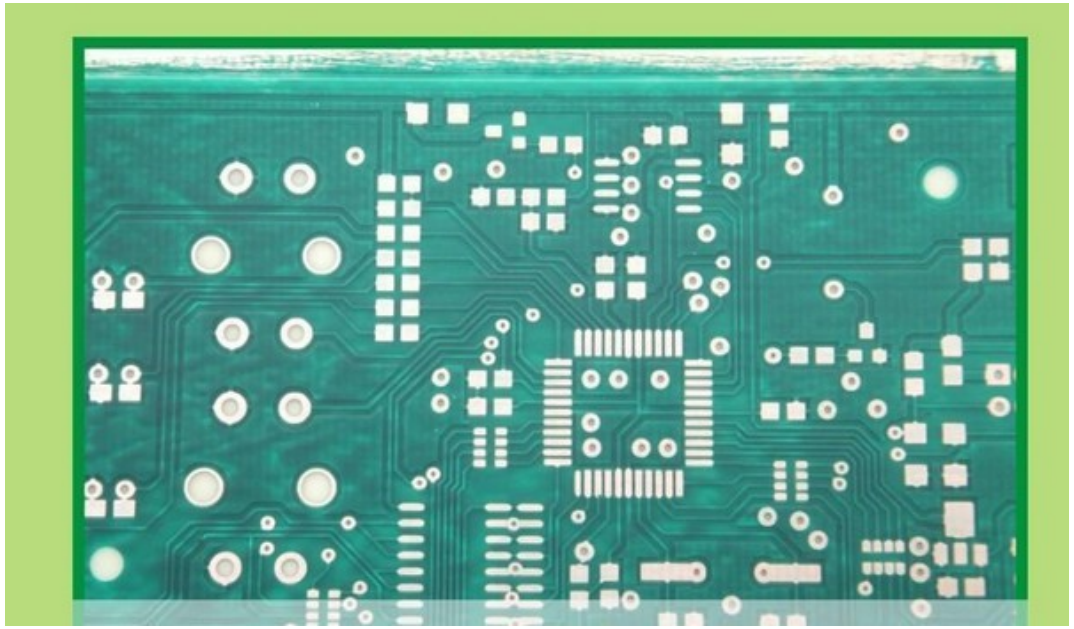
- Without through-hole plating, top and bottom sides of each hole are not connected.

Finishing steps (2/4)

2. A solder mask (also called solder resist) layer.

- This is painted on, cured with UV light, and then uncured mask is dissolved in a chemical bath.
- It protects the copper from oxidation and corrosion.
- Its surface tension repels liquid solder, making soldering easier.
- Various colours are available.





Finishing steps (3/4)

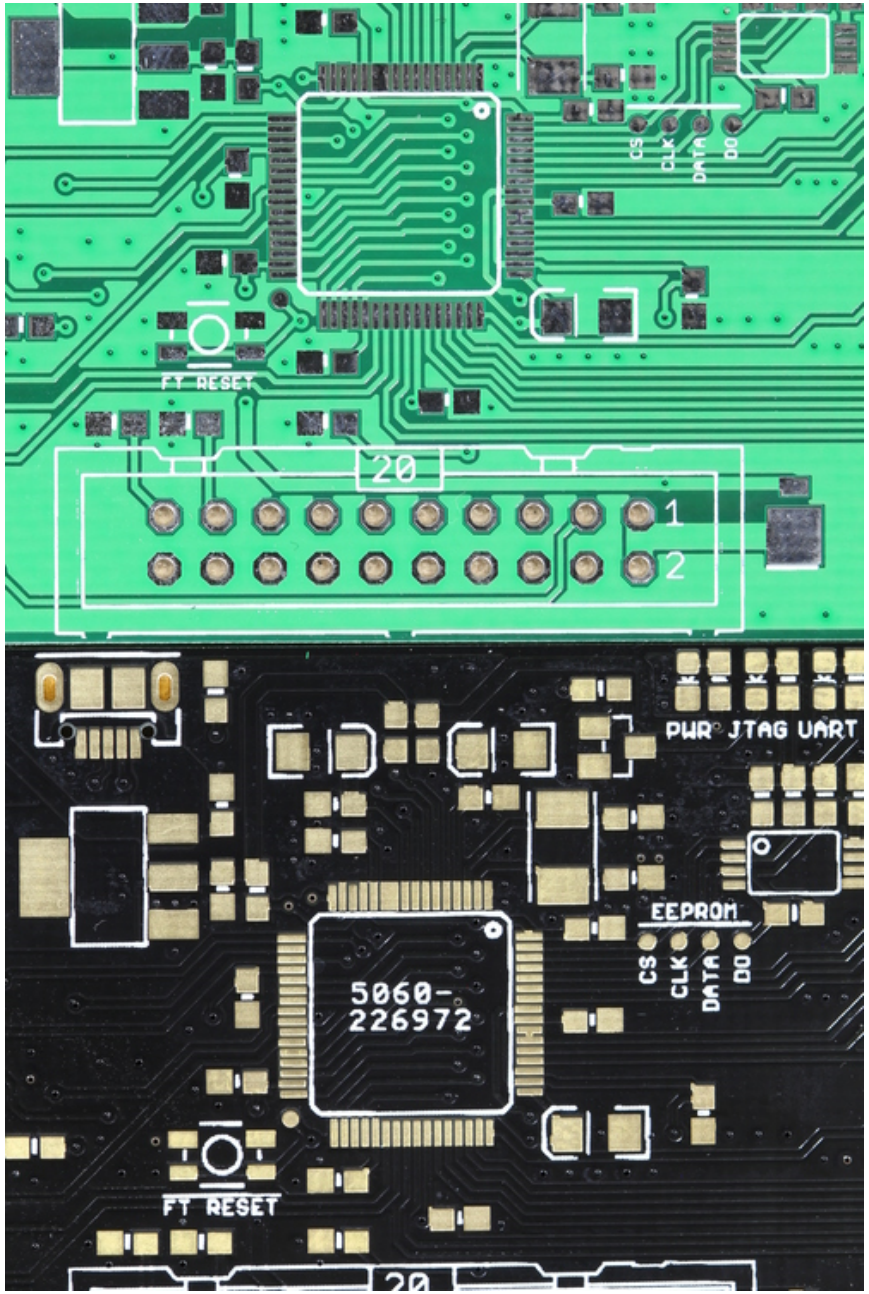
3. A surface treatment to protect exposed copper. Common methods:

Hot air solder levelling (HASL), where the board is dipped in molten solder and excess solder is removed by hot air.

- HASL is cheaper but tends to result in slightly uneven pads. (Top image)

Electroless nickel immersion gold (ENIG), in which nickel is chemically deposited and then finished with a surface layer of gold.

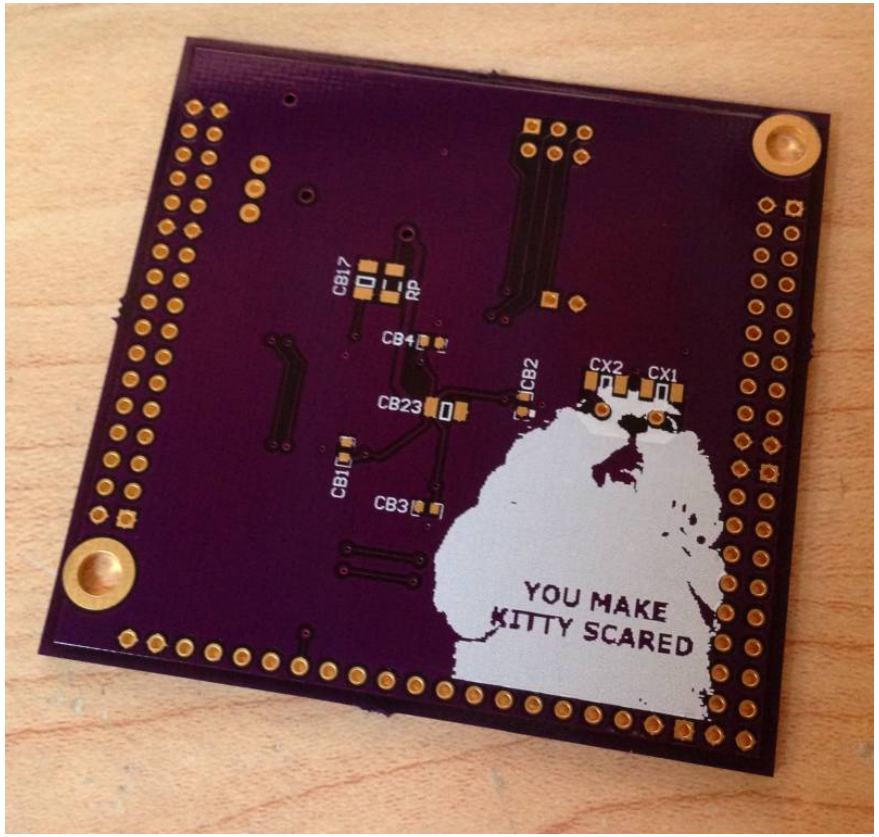
- ENIG results in flatter pads and greater mechanical strength. (Bottom image)



Finishing steps (4/4)

4. A silkscreen layer prints text/images onto the PCB.

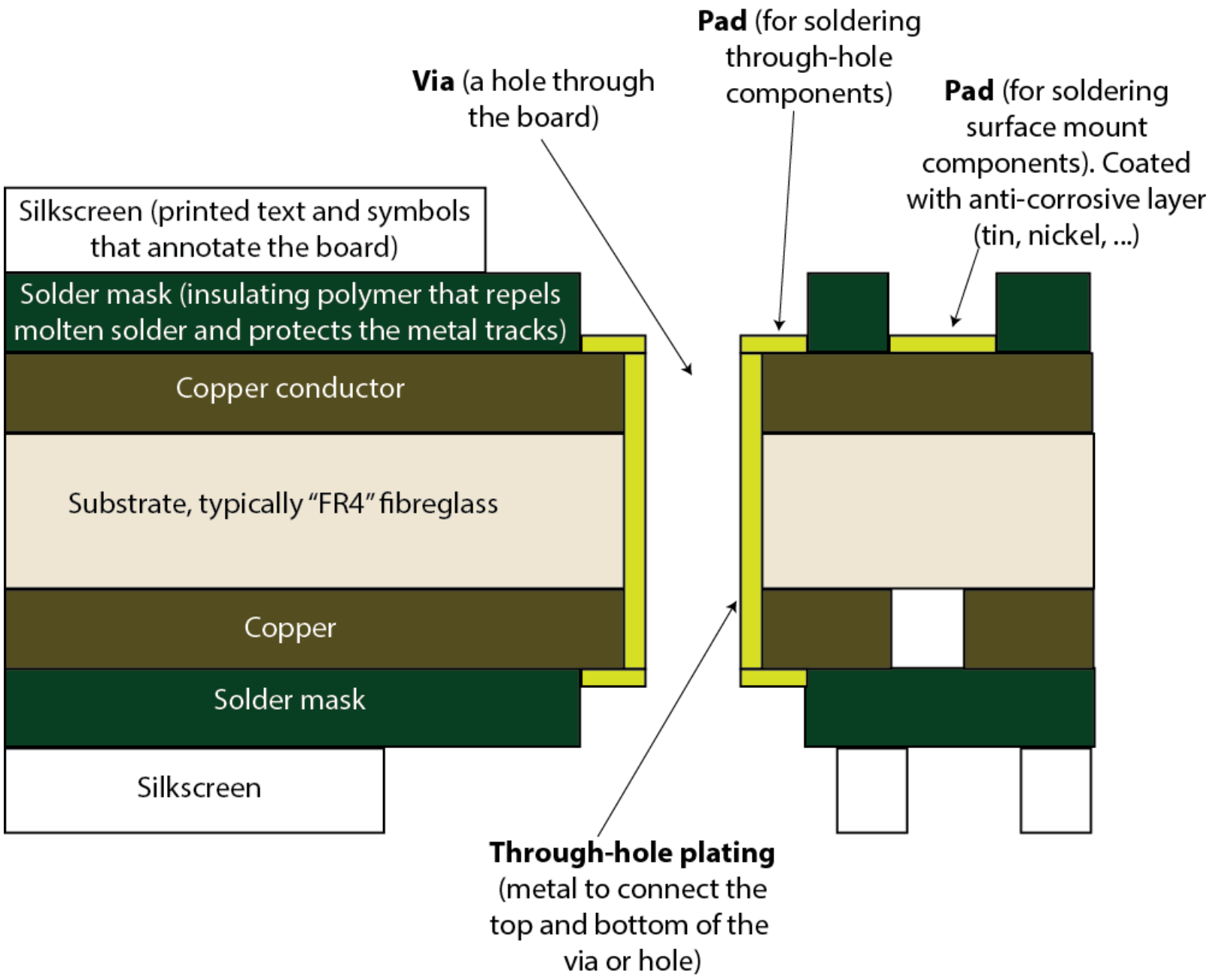
Traditionally done by screen printing, but can also be implemented with inkjet printing.



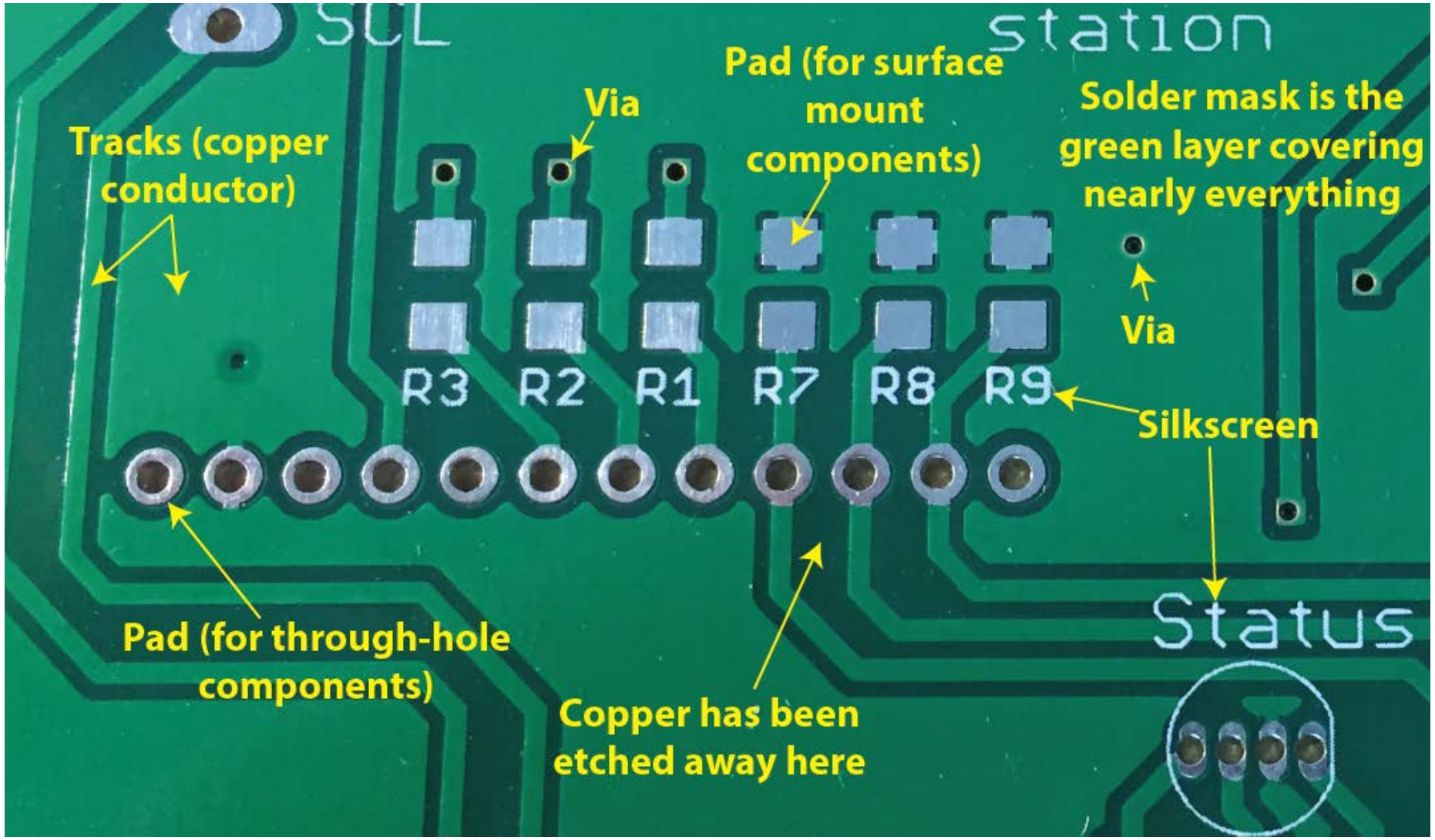
Commercial boards vs JCU-made boards

- Your Assignment 1 boards may be milled (and must be designed for milling).
  - Assume you will not have through-hole plating, solder mask, or silkscreen.

- Your Assignment 2 boards will be commercially made and will have all finishing steps.
  - Designs submitted late may be made in-house using the milling machine.

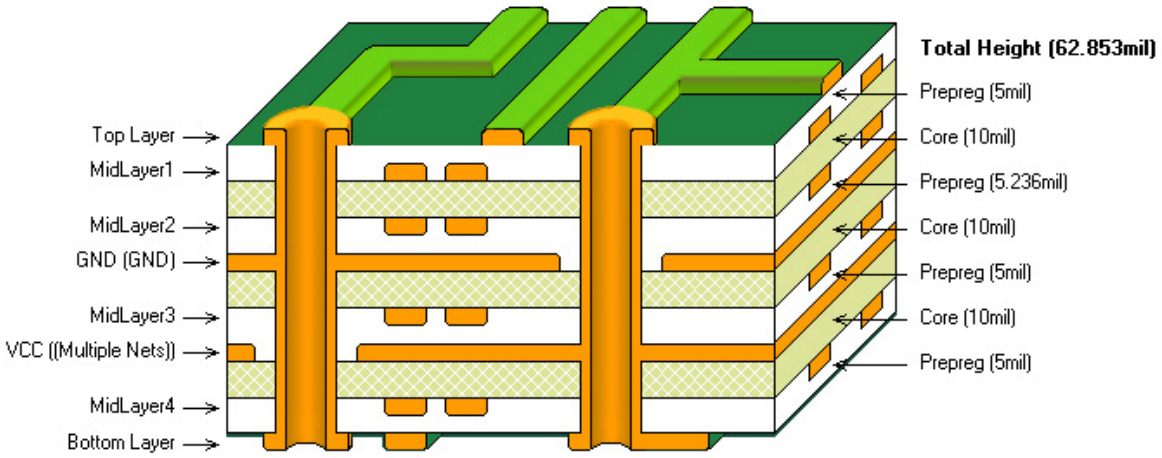

Pad (for soldering through-hole components)
Via (a hole through the board)
Pad (for soldering surface mount components). Coated with anti-corrosive layer (tin, nickel, ...)
Silkscreen (printed text and symbols that annotate the board)
Solder mask (insulating polymer that repels molten solder and protects the metal tracks)
Copper conductor
Substrate, typically "FR4" fibreglass
Copper
Solder mask
Silkscreen
Through-hole plating (metal to connect the top and bottom of the via or hole)

Unsoldered PCB (before assembly)



Multi-layer boards

- The cheapest and simplest PCBs have 1 or 2 layers of copper.
- More sophisticated designs have 4 – 8 layers (or more).
- These provide additional layers for routing tracks.
- Multi-layer boards are made as a sequence of thin 2-layer boards that are bonded together.
  - Therefore there will always be an even number of layers.



Defining the terms

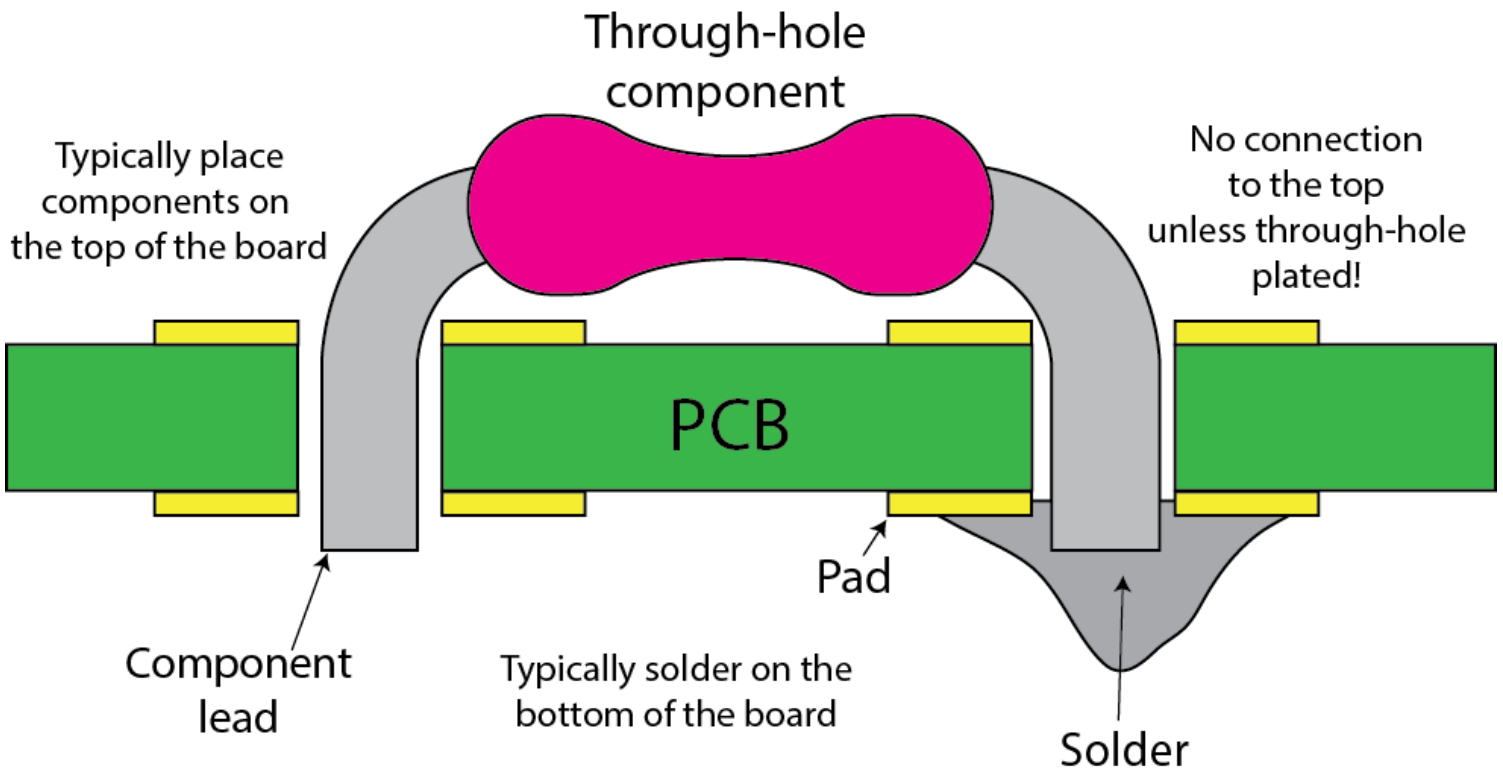
- **Via**: a hole drilled in the board that connects any two (or more) layers. In CC2511, vias always connect top and bottom layers.
- **Pad:** exposed metal intended to be soldered to.
- **Track:** a “wire” that connects pads and vias.
- **Footprint:** in CAD software, the “template” of pads and holes that represents a particular component.

Units of measurement

- Most component footprints are specified in inches.
- Note that a “mil” is 1/1000th of an inch.
- 1000 mils = 1 inch.
- Do not mistake “mils” as slang for “millimetres”!

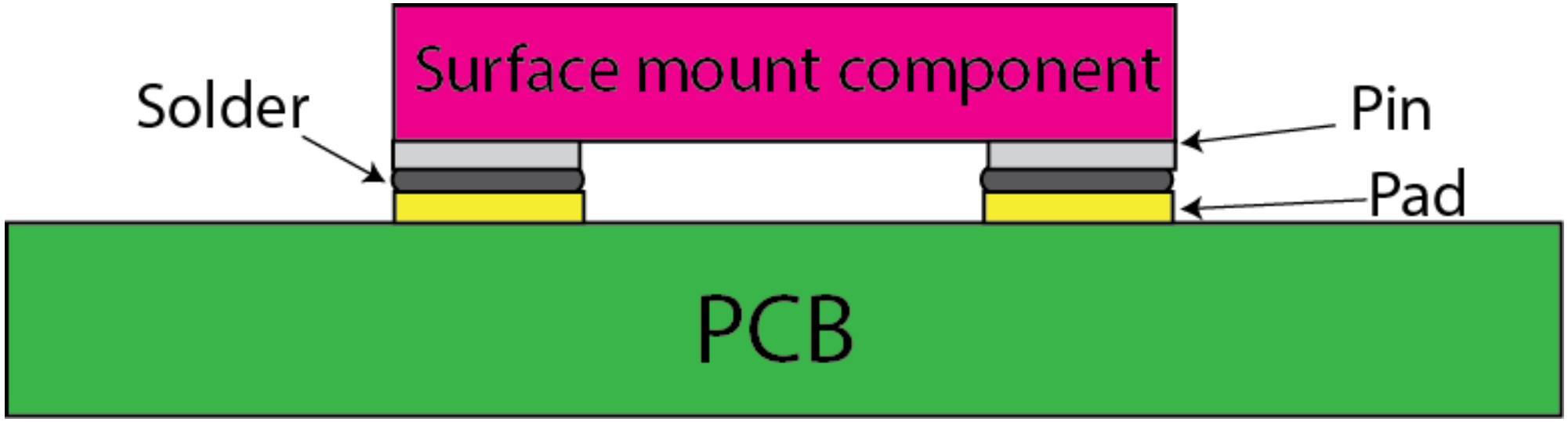
Types of components: Through-hole

- For hand-soldering, the easiest is through-hole technology.
- A common acronym is PTH (plated through-hole).



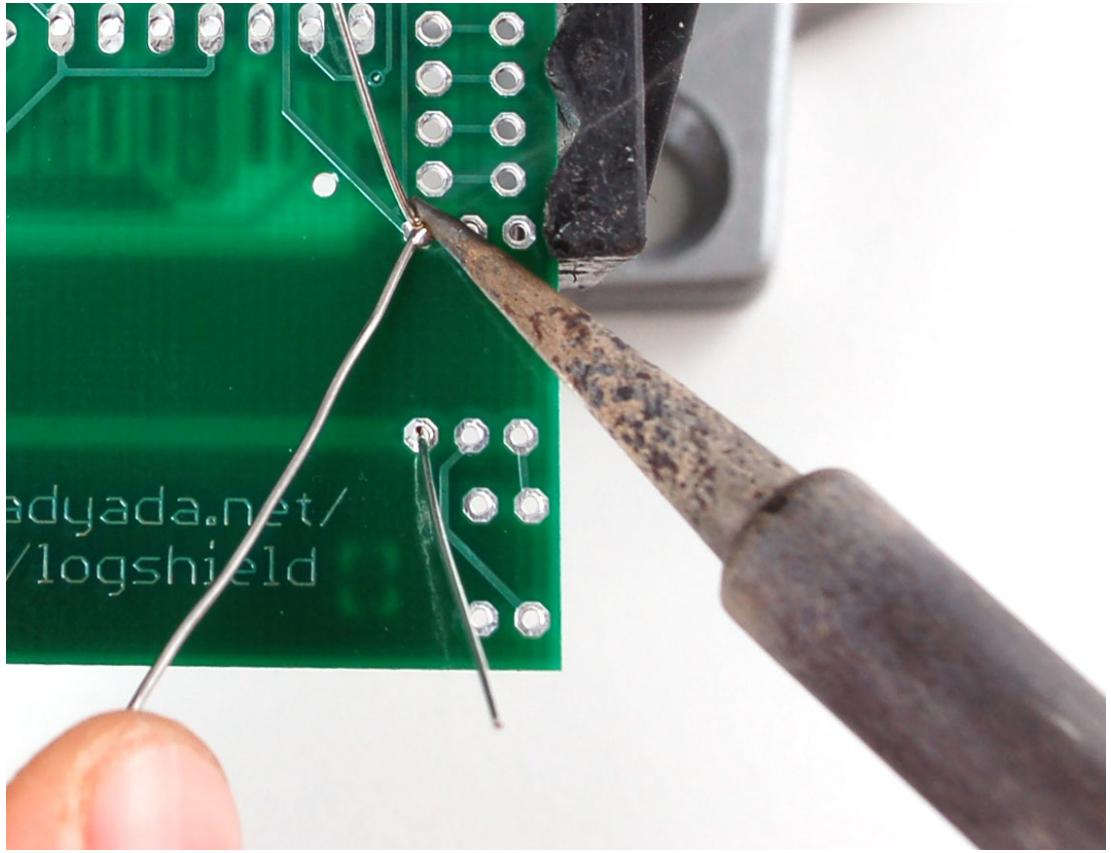
Types of components: Surface-mount

- Surface-mount technology (SMT) is becoming more and more common.
- A common acronym is surface-mount devices (SMD).
- It saves space and is easier to mass-manufacture.



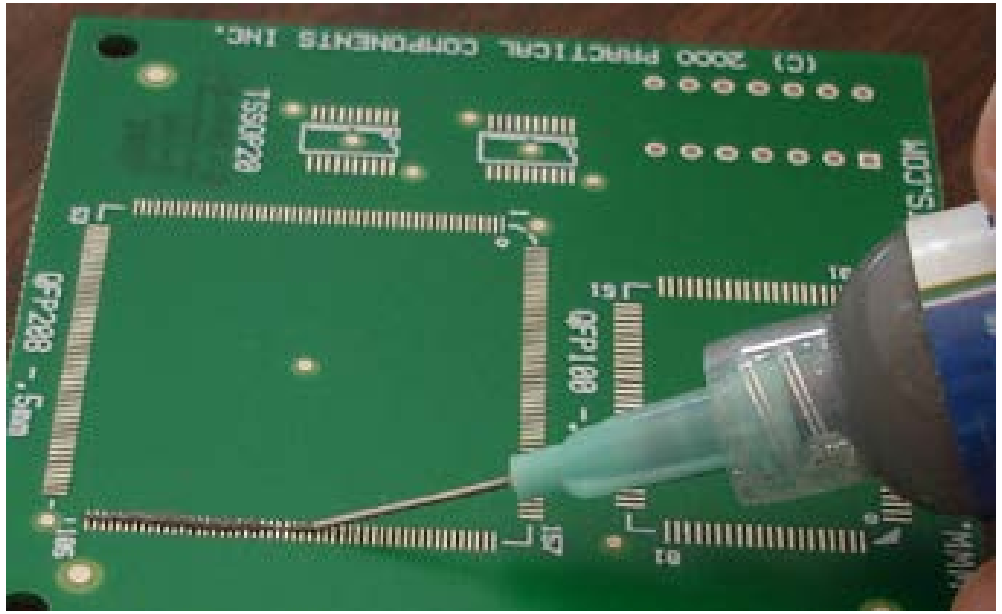
Soldering techniques

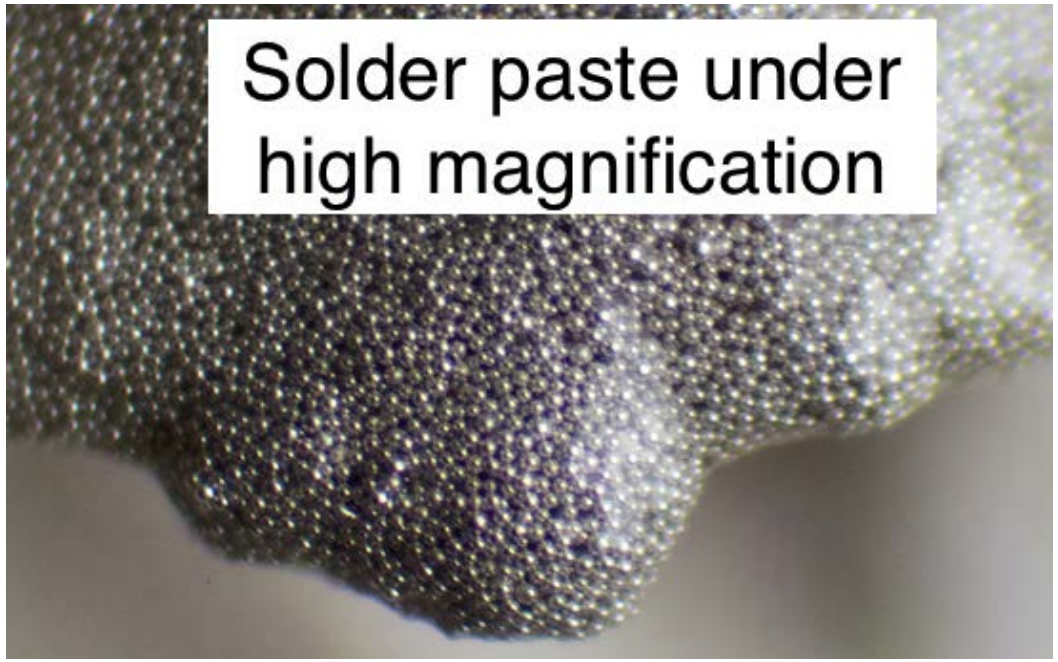
- Hand soldering with a soldering iron is useful for through-hole components and some surface-mount components.



Solder paste

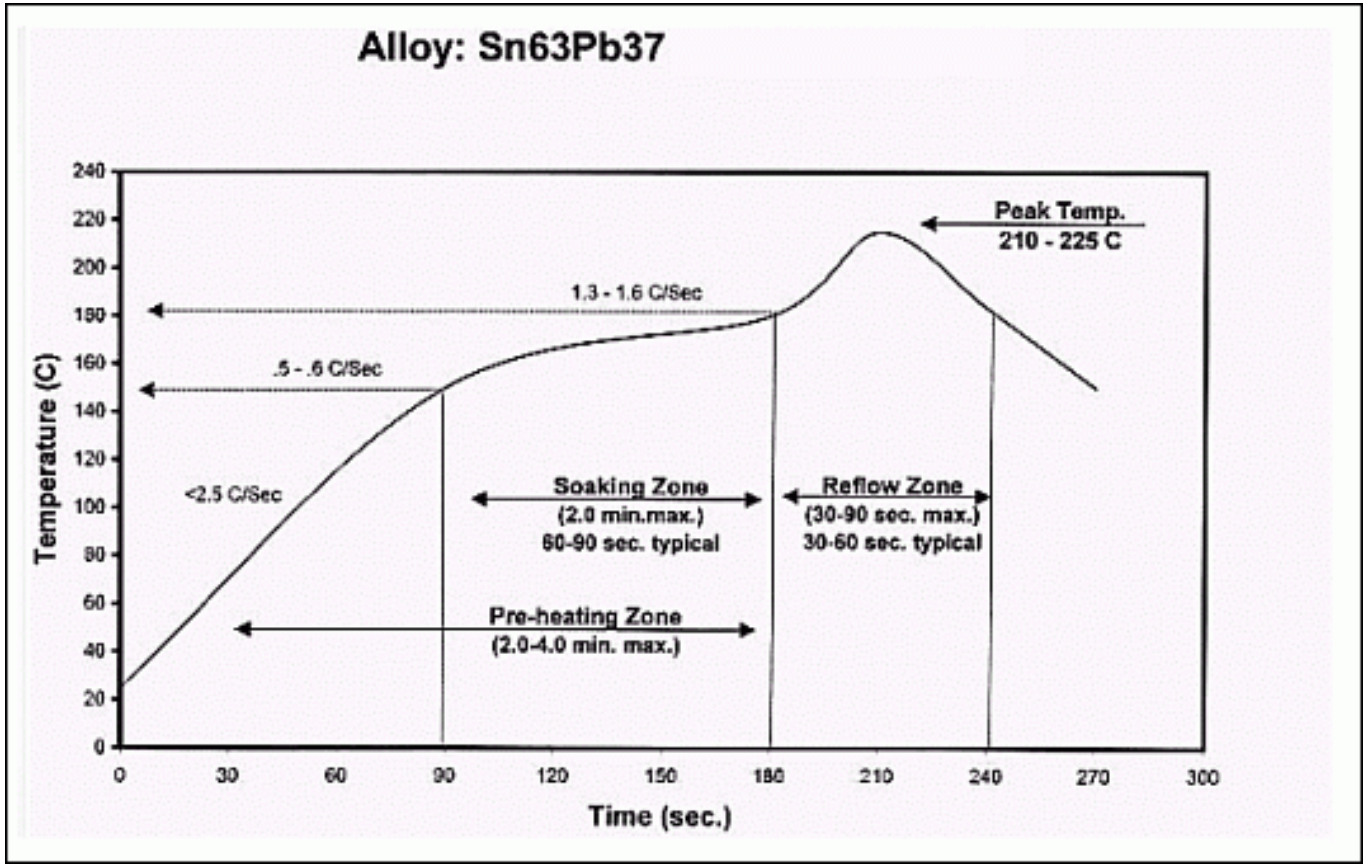
- For smaller components, **solder paste** and a **reflow oven** are used.
- Solder paste is powdered solder suspended in a putty-like medium called flux.
- The flux improves the flow of the molten solder, and evaporates away during soldering.
- Some fluxes leave behind a residue that should be cleaned off the board.



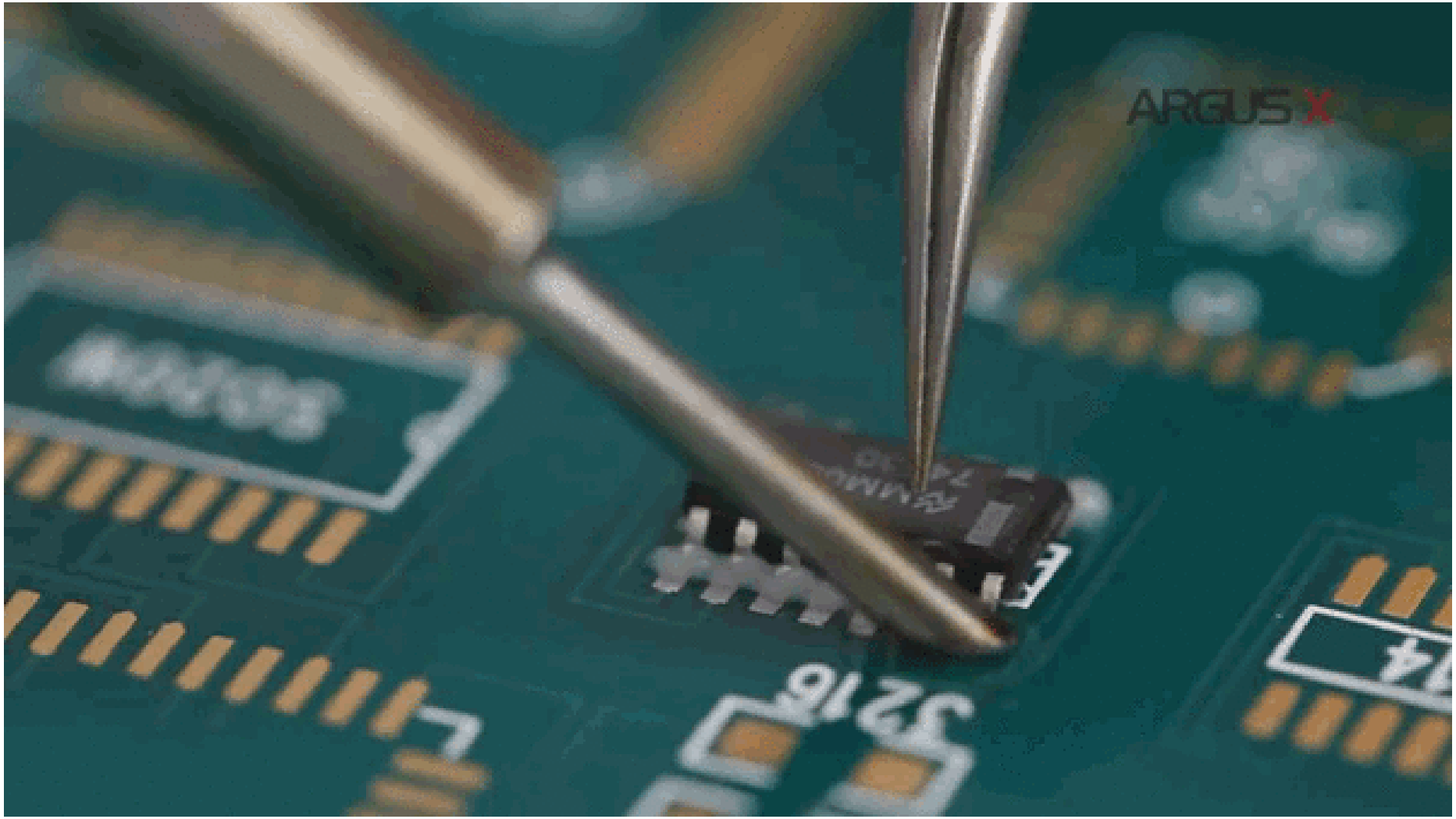


Reflow soldering

- A reflow oven controls the temperature according to a particular programmed pattern.

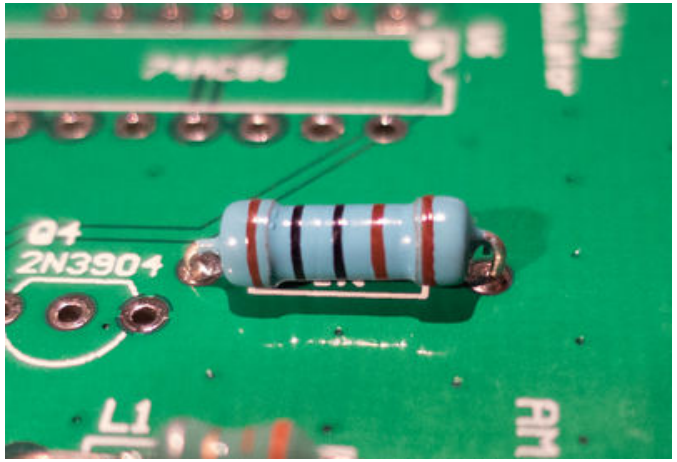


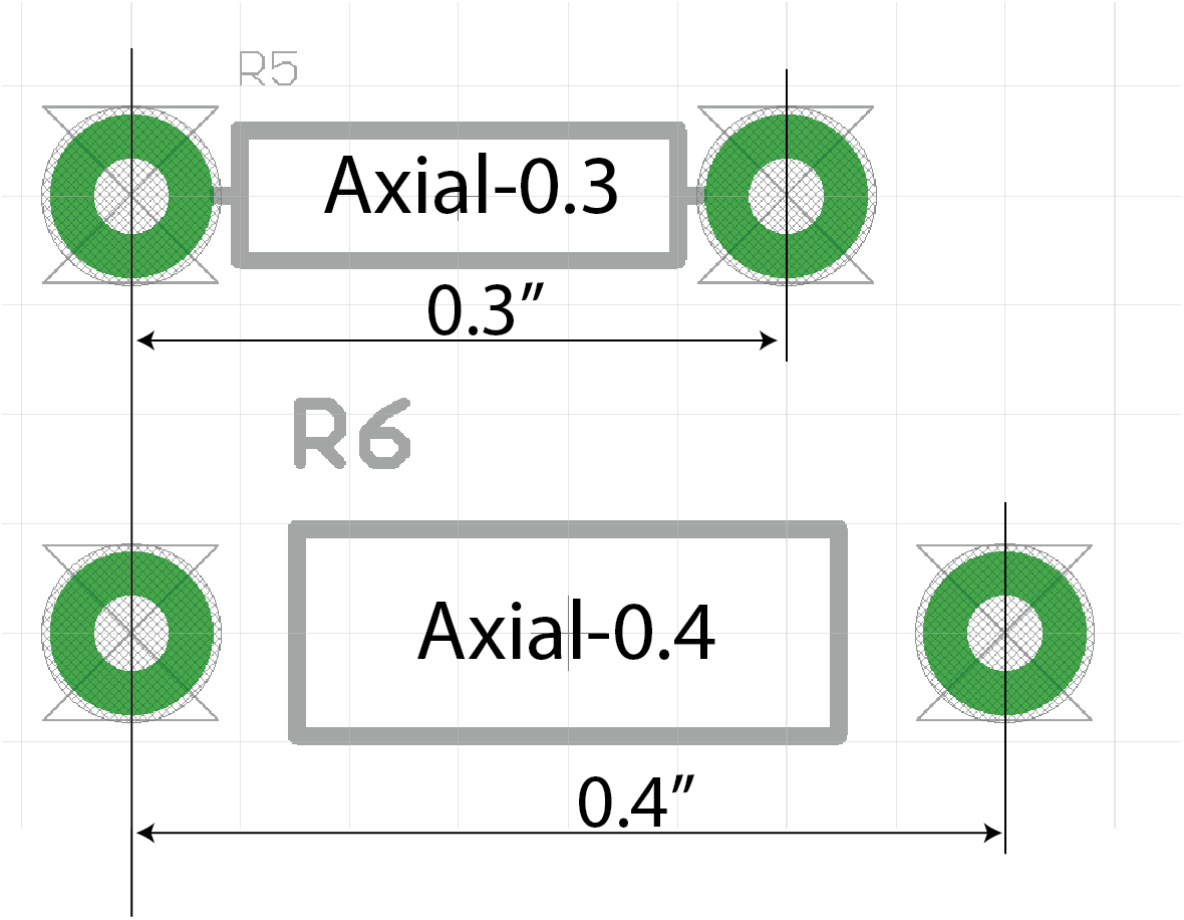
Solder paste surface tension



Axial resistor footprints

- Standard through-hole resistor footprints are called “axial” and named for the separation of the holes in inches.





Surface mount resistors

- Two standards: metric and imperial.
- Always a 4 digit code specifying “length” and “width”.
- Common sizes:

Size code (imperial)	Length (l)	Width (w)	Size code (metric)	Length (l)	Width (w)
0603	60 mils = 0.06”	30 mils = 0.03”	1608	1.55 mm	0.85 mm
0805	80 mils = 0.08”	50 mils = 0.05”	2012	2.0 mm	1.2 mm
1206	**120 mils = 0.12”**	**60 mils = 0.06”**	**3216**	**3.2 mm**	**1.6 mm**

SMD (surface mount device) resistor markings

If the marking has an “R” at the start or in the middle:

- The R is a decimal point.
- Examples:

  “R10” = 0.10 Ω

  “10R0” = 10.0 Ω

If the marking consists entirely of numbers:

- The last digit is a power of 10.
- Examples:

  “331” = $33 \times 10^1 = 330\ \Omega$

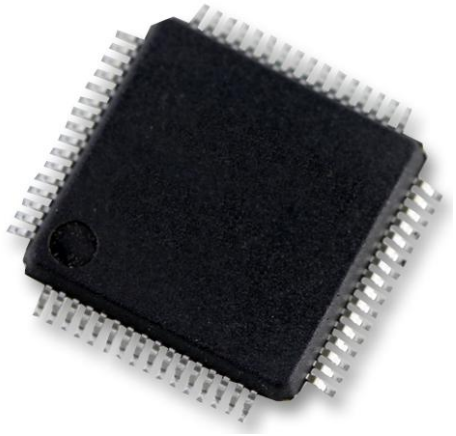
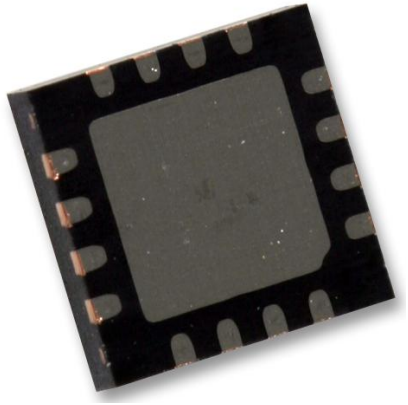
  “100” = $10 \times 10^0 = 10\ \Omega$

  “105” = $10 \times 10^5 = 1\ \mathrm{M}\Omega$

NB: There is another standard “EIA-96” used for 1% tolerance (and sometimes 5%). EIA-96 uses numbers and letters, e.g. “22A”. These require a look-up table. Confusingly, one of the letters is R! The EIA-96 standard always places the letter last.

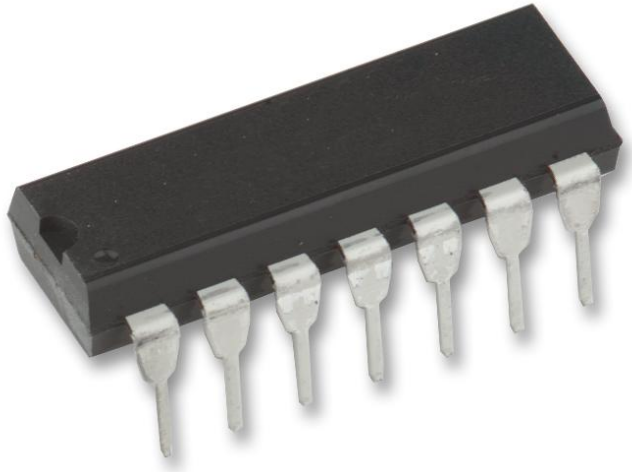
IC packages

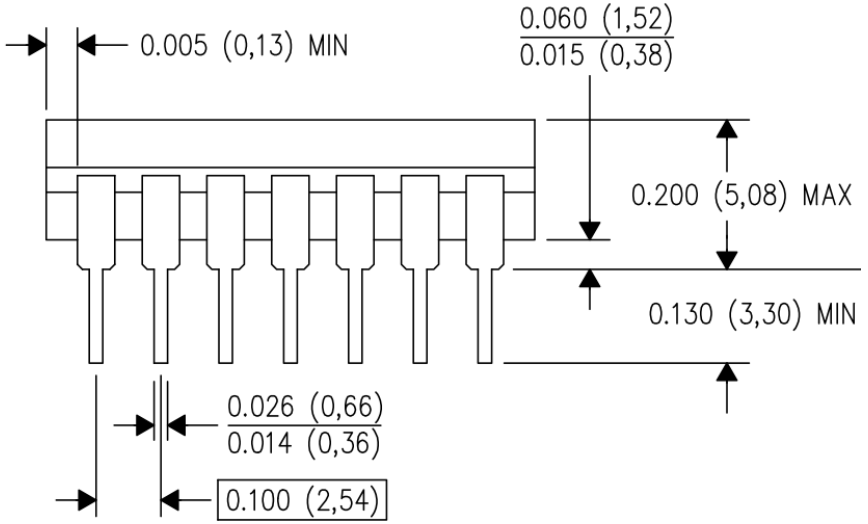
- There are many IC packages but it’s helpful to remember a few broad classes.

Integrated circuit packages: DIP

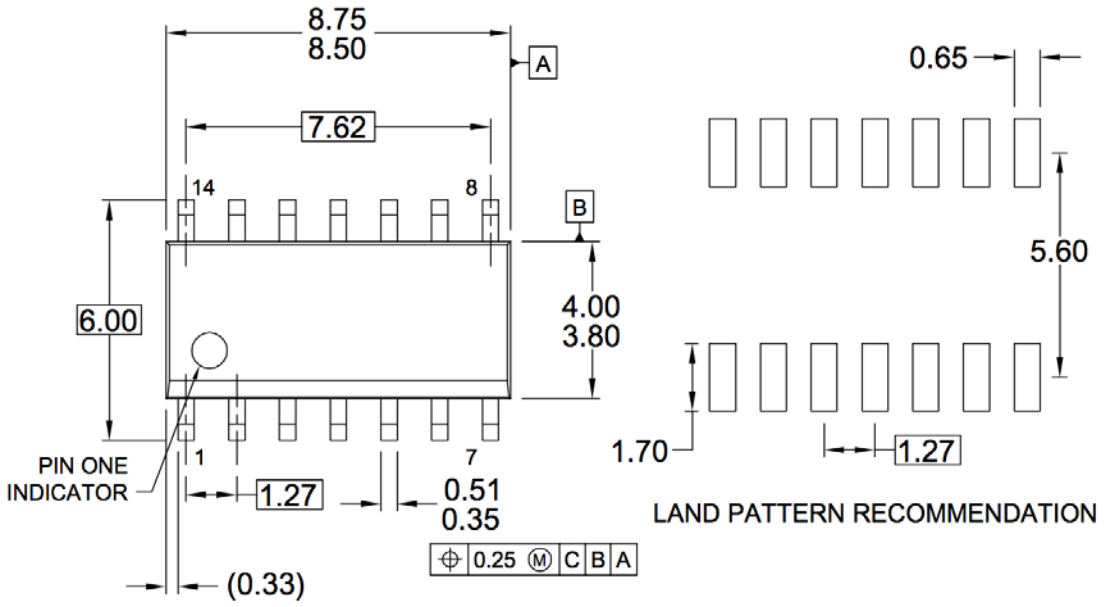
- **Dual-inline package (DIP)** is the most common footprint for through-hole ICs.
- The pin pitch (distance between pins) is 0.1 inches, or 2.54 mm.
- DIP packages fit in breadboards.





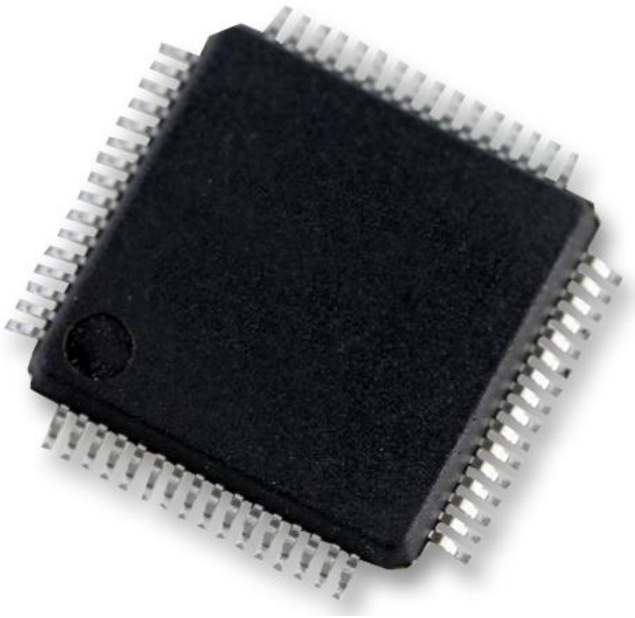
IC packages: SOIC

- **Small outline integrated circuit (SOIC)** packages are surface mount footprints with half the pitch of a DIP.



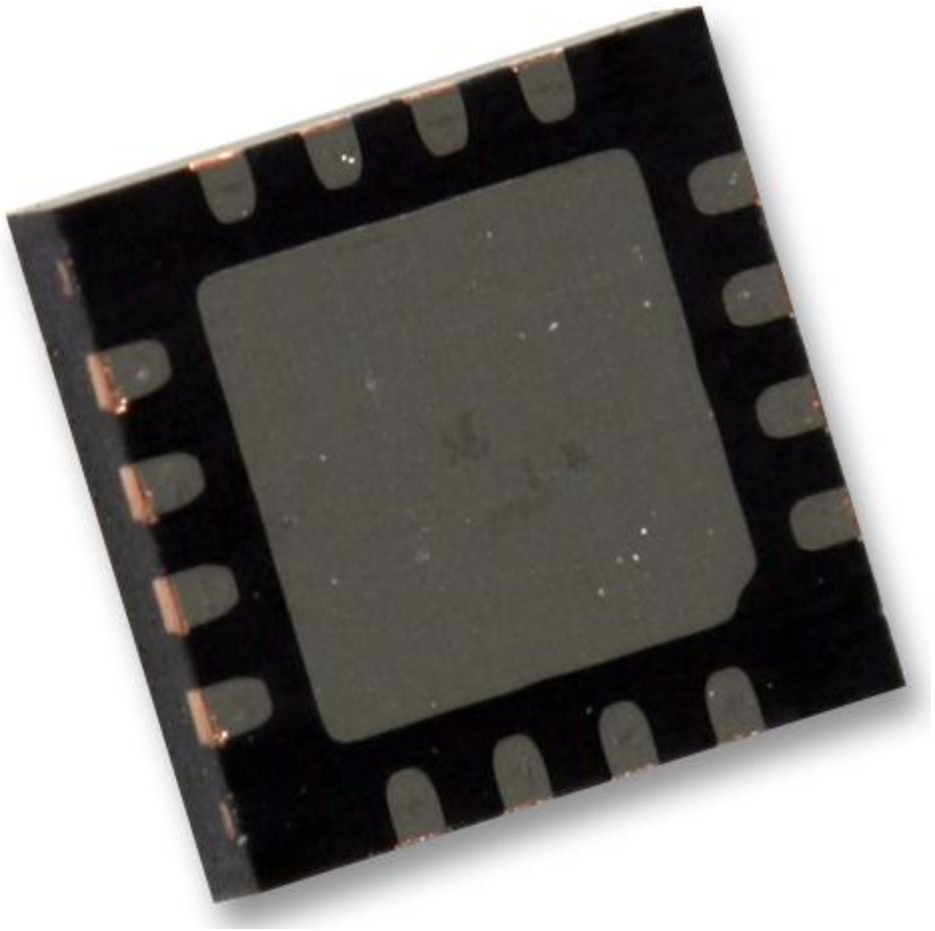
IC packages: LQFP

- Low Quad Flat Package (LQFP) is an even smaller surface mount package common for parts with many pins, e.g. microcontrollers.
- Typical pin pitch is ~ 0.5 mm.
- LQFP can be soldered by hand using solder paste and a hot air gun.



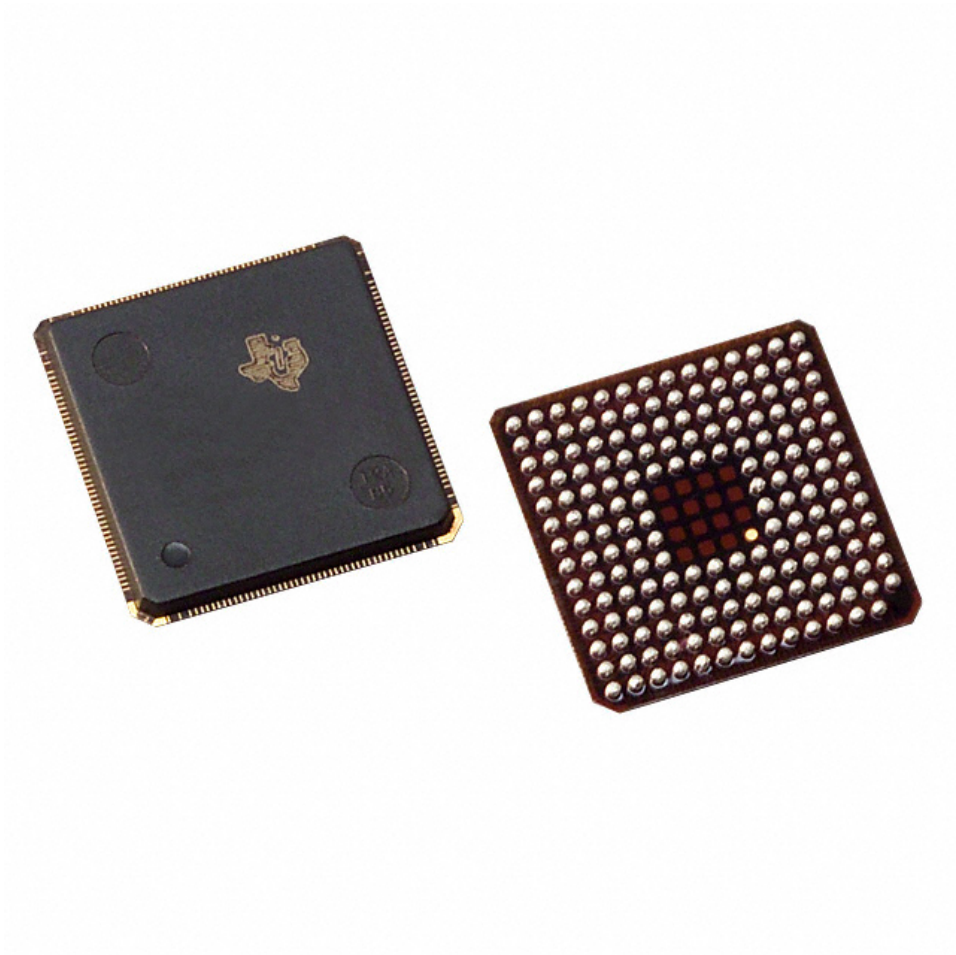
IC packages: QFN

- **Quad flat no leads (QFN)** are soldered underneath the package.
- Solder paste is applied to the pads, then the component is placed on top and the board is run through a reflow oven.



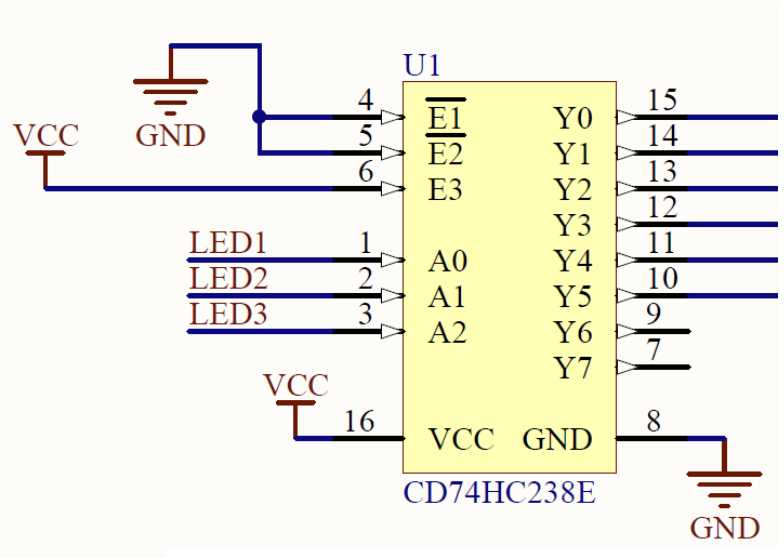
IC packages: BGA or LGA

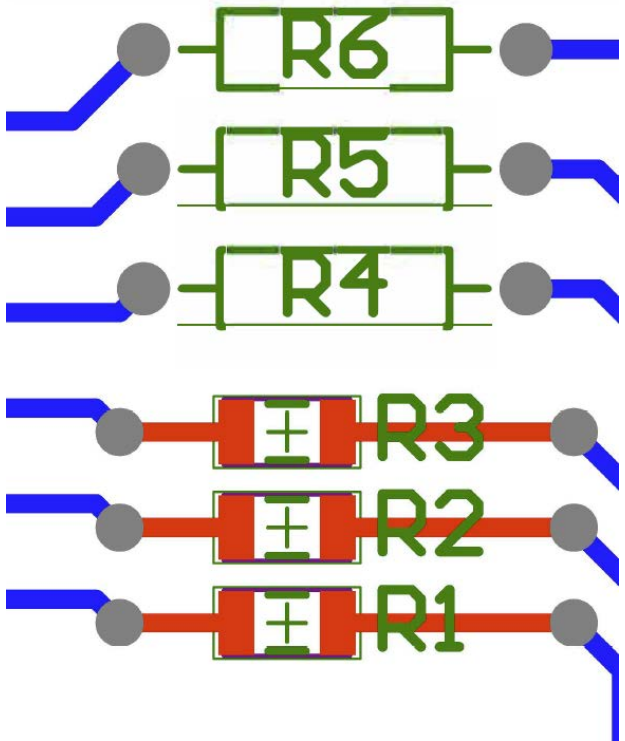
- **Ball grid arrays** (BGA) and **Land grid arrays** (LGA) are soldered underneath the component.
- These require specialist pick-and-place equipment to deposit the solder paste and place the component.



The PCB design process

1. **Schematic**: Shows the components that are present and how they are connected.
   - Layout should be easy to read, and not necessarily correspond to the physical positioning.
2. **PCB Layout**: Gives the physical positioning of components, tracks, vias, etc.
   - This is what is used for the fabrication step.





Computer-aided design software

- In this class we will use PCB design software called **Altium Designer**.
- Other PCB design software includes Eagle, KiCad, DesignSpark PCB, and others.

Access to Altium Designer

- JCU has an Altium license available to all students.
- Altium is preinstalled on the lab computers.
- To use Altium on your own computer, see LearnJCU for the installer and log in details.

How to use Altium Designer

- **Watch the Altium screencasts before this week's lab!**
- Some written documentation is also available on LearnJCU for your reference.
- Overview:
  1. **Schematic file:** Place all components onto the schematic.
  2. Create connections by drawing wires between components.
  3. **PCB file:** Create PCB file from schematic, and arrange components.
  4. Route tracks between pads to implement the connections defined earlier.

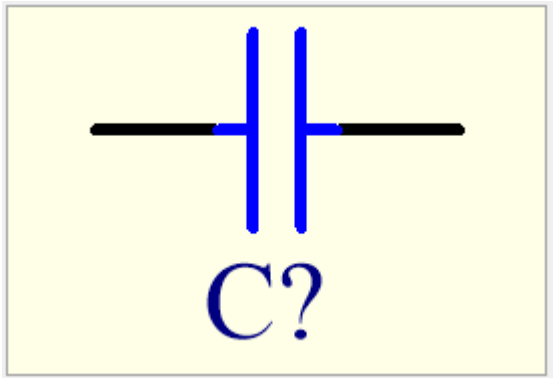
Altium concept: component

- **A component represents a physical part**, e.g. capacitor, 1206 footprint, 0.1 uF +/- 5%, rated for 50V.
  - “0.1 uF cap” is not enough information to buy a part!

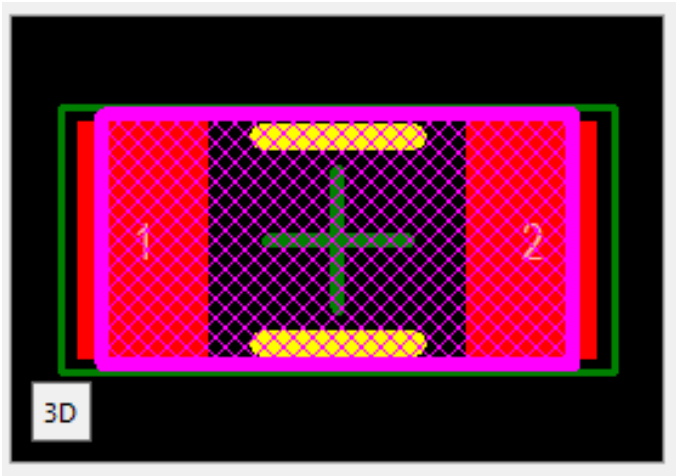
12065C104JAT2A	2280701 Data Sheet RoHS compliant	AVX - SMD Multilayer Ceramic Capacitor, 1206 [3216 Metric], 0.1 µF, 50 V, ± 5%, X7R Capacitance 0.1µF Ceramic Capacitor Case 1206 [3216 Metric] Voltage Rating 50V + See all product info	23 available for next business day delivery 27,218 available for 6 - 7 business days delivery Check Stock & Lead Times	Each (Supplied on Cut Tape) Cut Tape Packaging Options	1+ $0.615 10+ $0.335 100+ $0.188 500+ $0.167 1000+ $0.147 More Pricing...
12061C104JAT2A	2332842 Data Sheet RoHS compliant	AVX - SMD Multilayer Ceramic Capacitor, 1206 [3216 Metric], 0.1 µF, 100 V, ± 5%, X7R Capacitance 0.1µF Ceramic Capacitor Case 1206 [3216 Metric] Voltage Rating 100V + See all product info	2,691 available for next business day delivery 22,310 available for 6 - 7 business days delivery Check Stock & Lead Times	Each (Supplied on Cut Tape) Cut Tape Packaging Options	1+ $0.574 10+ $0.326 100+ $0.199 500+ $0.178 1000+ $0.151 More Pricing...
C1206C104K5RACTU	1414715 Data Sheet RoHS compliant	KEMET - SMD Multilayer Ceramic Capacitor, 1206 [3216 Metric], 0.1 µF, 50 V, ± 10%, X7R, C Series Capacitance 0.1µF Ceramic Capacitor Case 1206 [3216 Metric] Voltage Rating 50V + See all product info	14,259 available for next business day delivery 199,364 available for 6 - 7 business days delivery Check Stock & Lead Times	Each (Supplied on Cut Tape)	10+ $0.039 100+ $0.029 250+ $0.022 500+ $0.018 1000+ $0.017

A component contains:

- A schematic symbol (how it appears on your schematics)

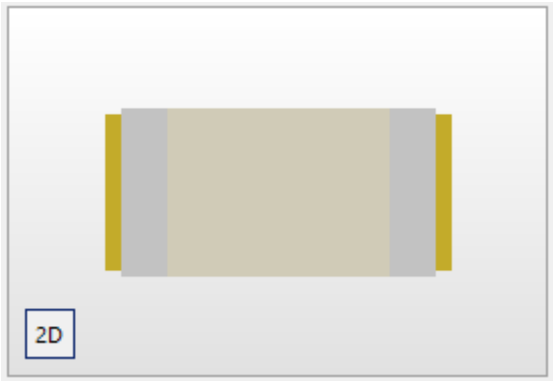


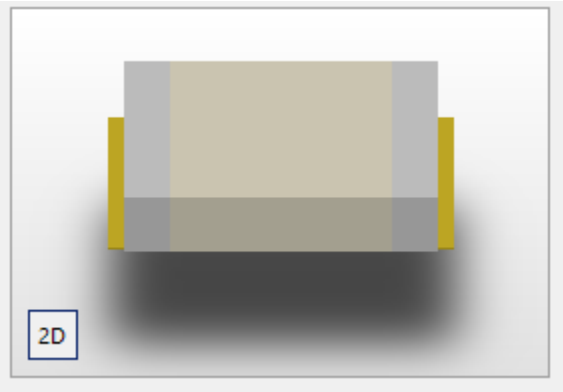
- A PCB footprint (how big the pads are, where they are located, what silkscreen is drawn, marks for pick-and-place machines, …)



A component contains:

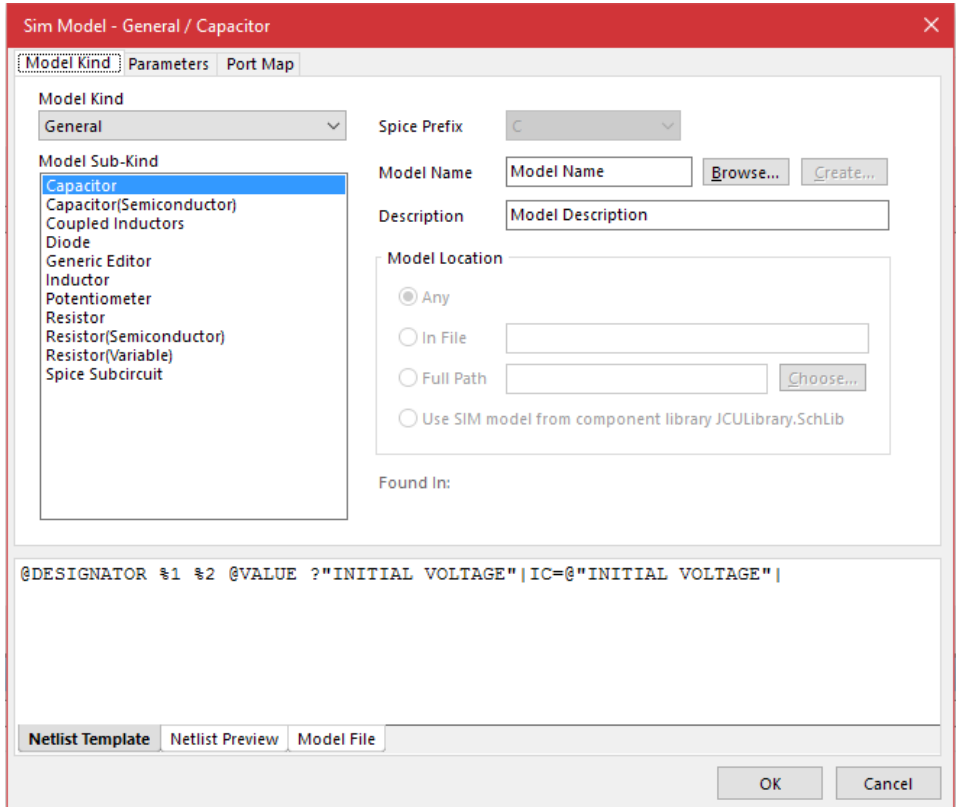
- Optionally, a 3D model to help with mechanical co-design, checking clearances between components, renderings of the board pre-manufacture, etc.





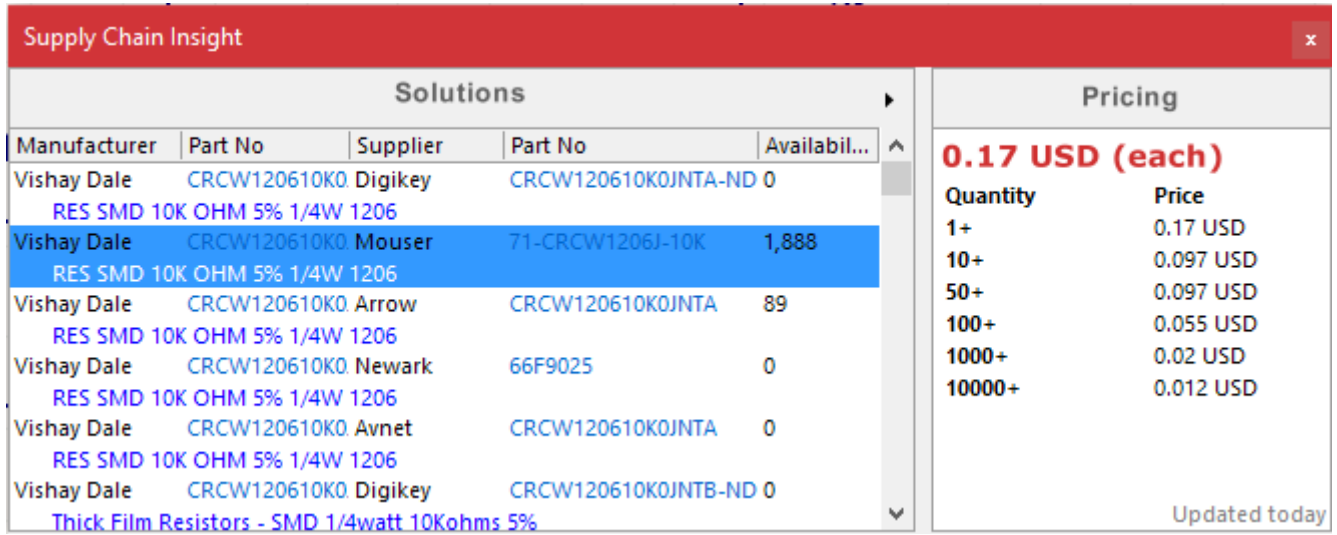
A component contains:

- Optionally, a SPICE model for circuit simulation.
  - You can link external SPICE models for modelling complex components like semiconductors.



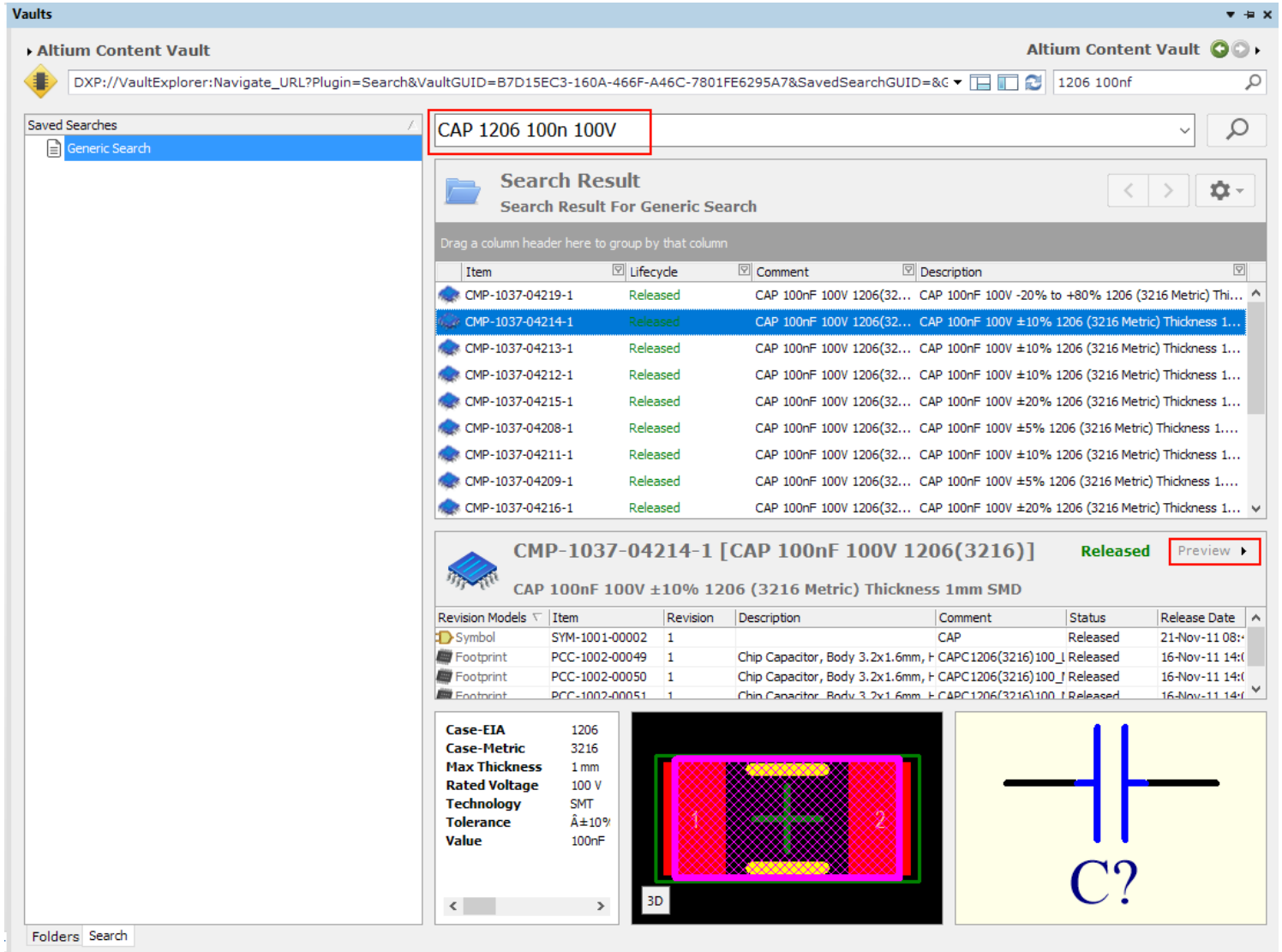
A component contains:

- Optionally, links to suppliers so you can check prices & availability, and export a complete bill of materials (BOM).



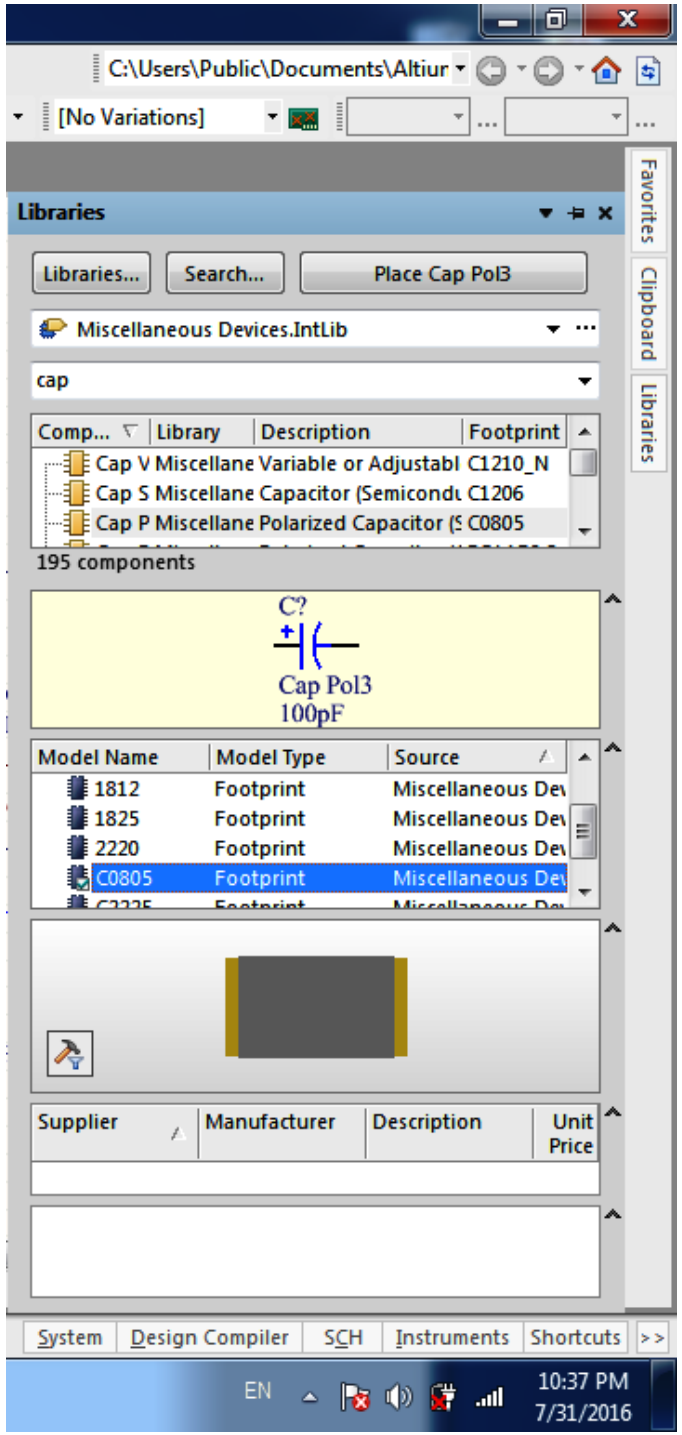
Looking up components in the Vault

- The “Content Vault” is an online collection of pre-made components.
- Click the “preview” link the Vault to see the schematic symbol and PCB footprint.
- Click “3D” to toggle the 3D model and footprint.



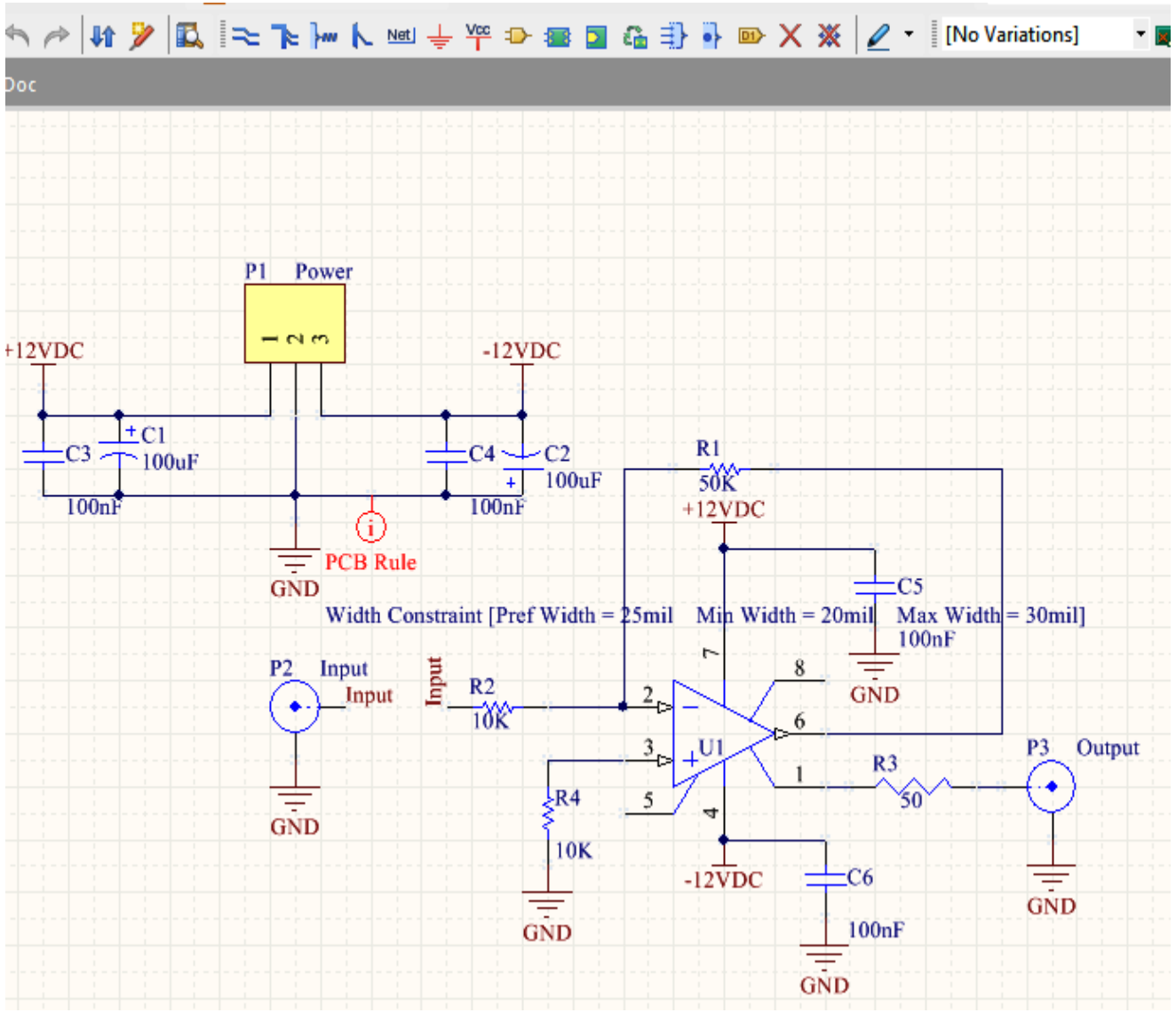
Libraries

- You can also build your own components and store them locally in a **library.**
- Some libraries break the ideal of one component = one part, e.g. with generic resistor and capacitor parts.
- This works for small scale projects if you're careful. However, it's not best practice PCB design, so try to avoid it.



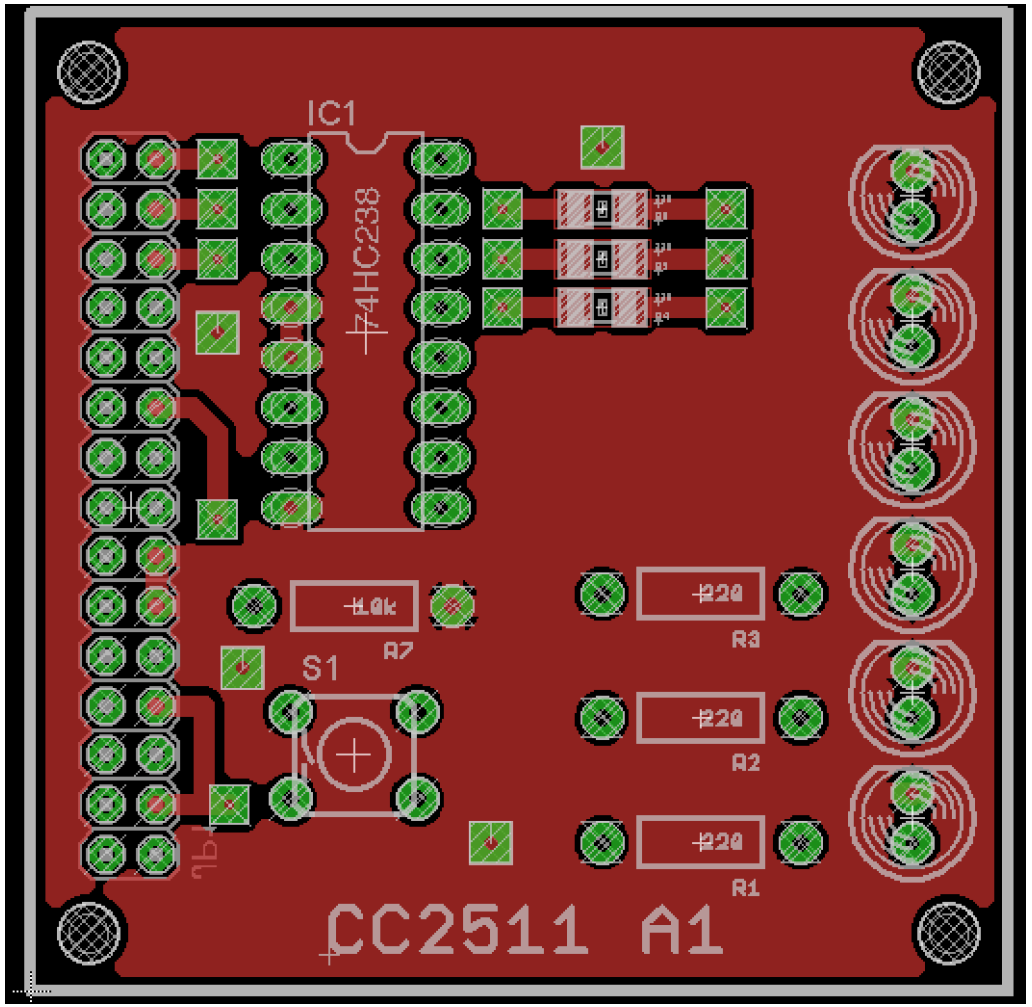
Connecting components

- Components are connected using wire.
- Every component has a designator, e.g. “C1”, “C2”, …
- Components can be edited by double clicking on them.



Polygons

- A **polygon** is a large segment of copper, commonly used for a **ground plane**. (The large red area to the right.)
- Commonly both the top and bottom of the board will have a ground plane polygon.
- Some multi-layer boards place a Vdd polygon on an inner layer.



Summary

- Today we looked at the basics of PCB design and fabrication.
- In your labs you'll start on your first PCB in Assignment 1.