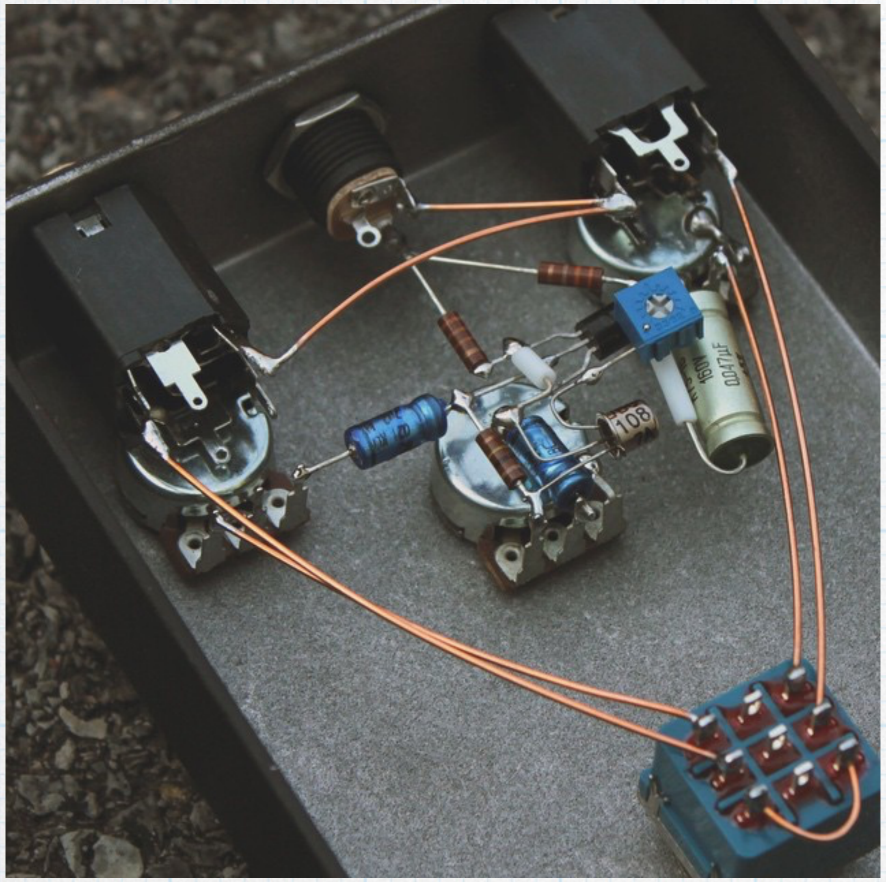
printed circuits boards (PCBs)

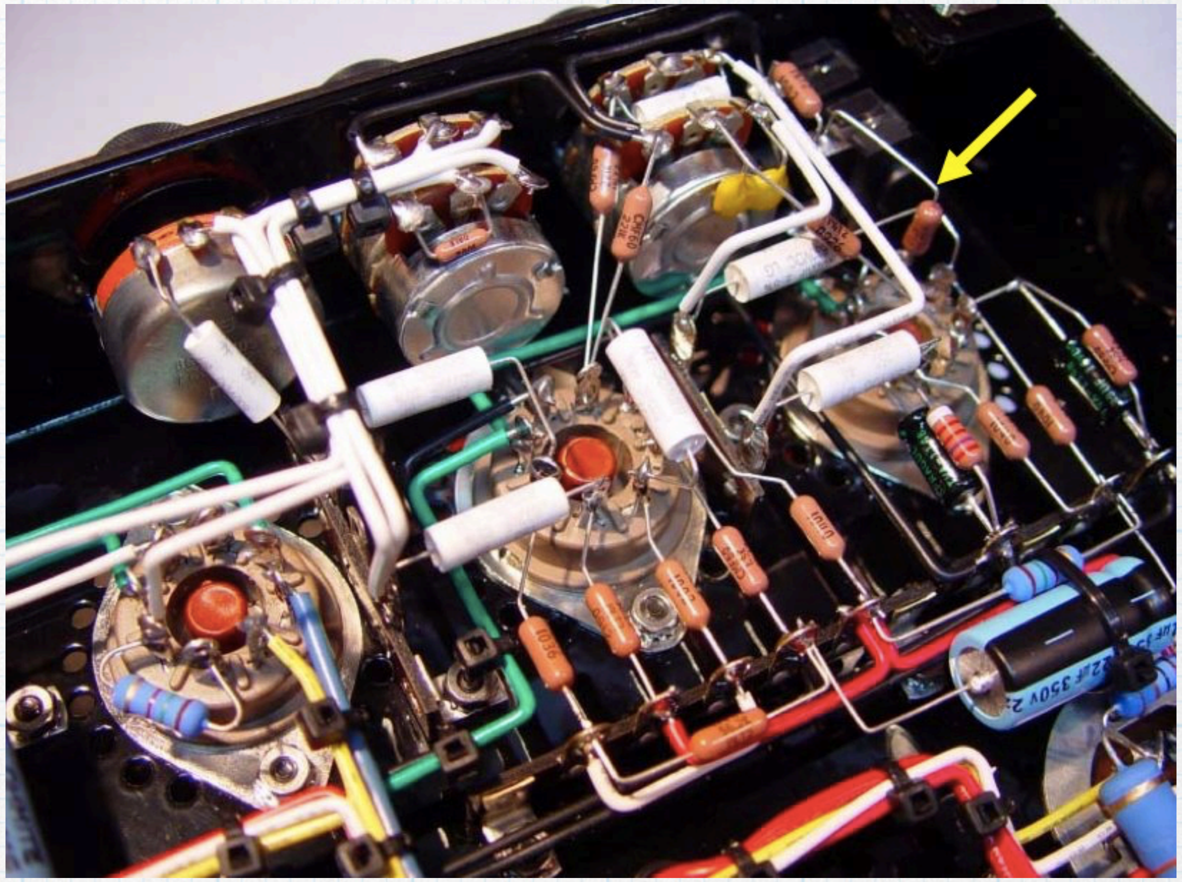
A PCB provides a substrate that simultaneously serves as a structural mount for the various components (chips, resistors, capacitors, sensors, switches, connectors, etc), insulation between isolated components, and metal inter-connections between the components. Since all of the wiring is contained in the PCB, building a circuit is a simple matter of soldering the components into place.

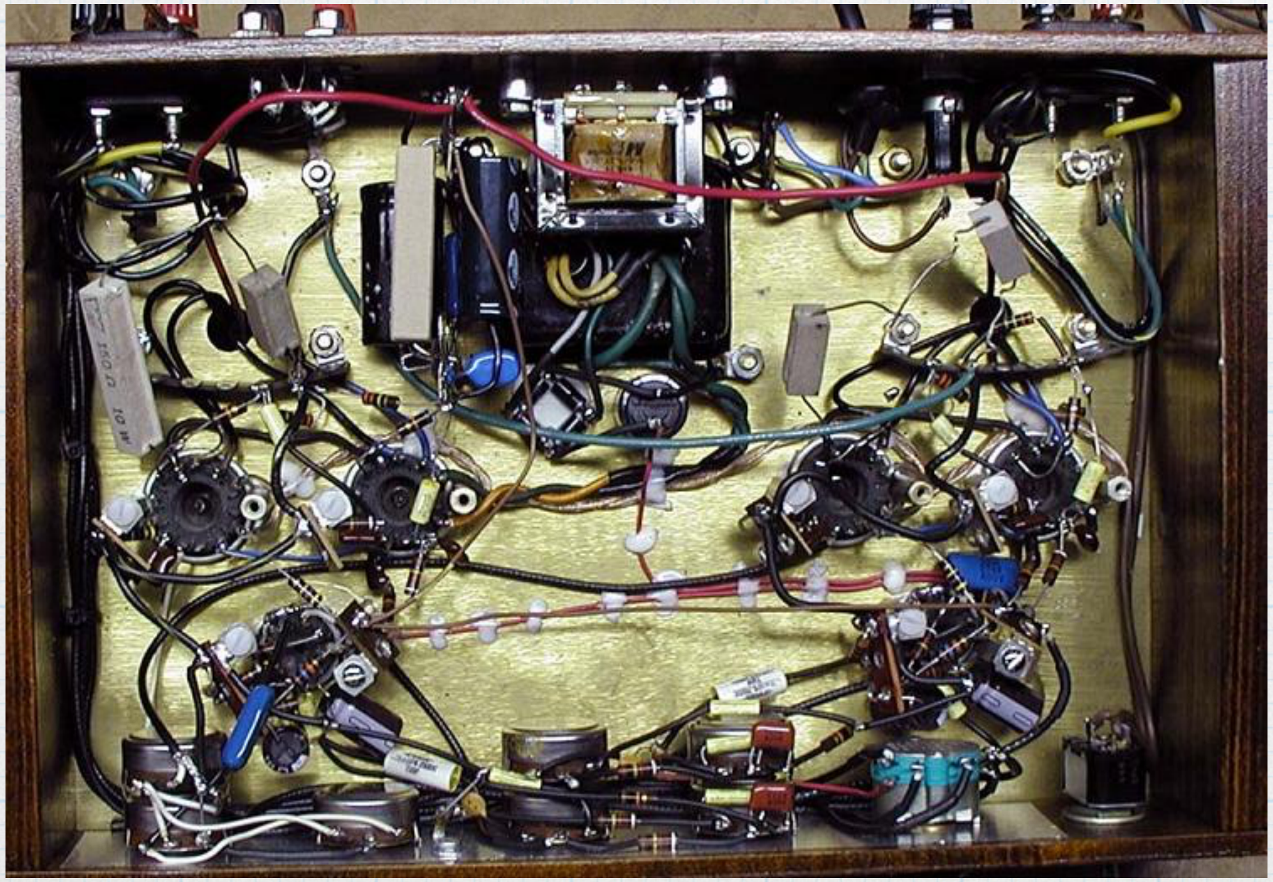
The advantages:

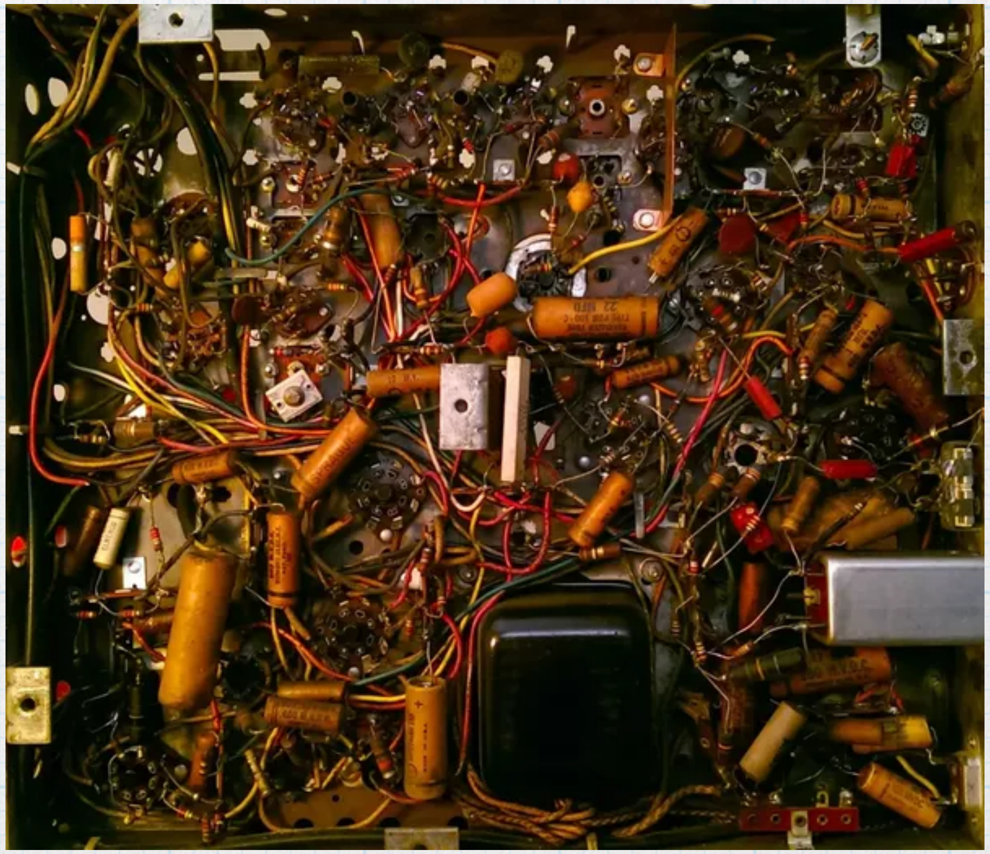
- Repeatability
- Better reliability
- Lower cost (automated manufacturing)
- Reduced size
- Faster circuit speeds (probably)
- Better heat handling (probably)

The old days — Point-to-point soldering

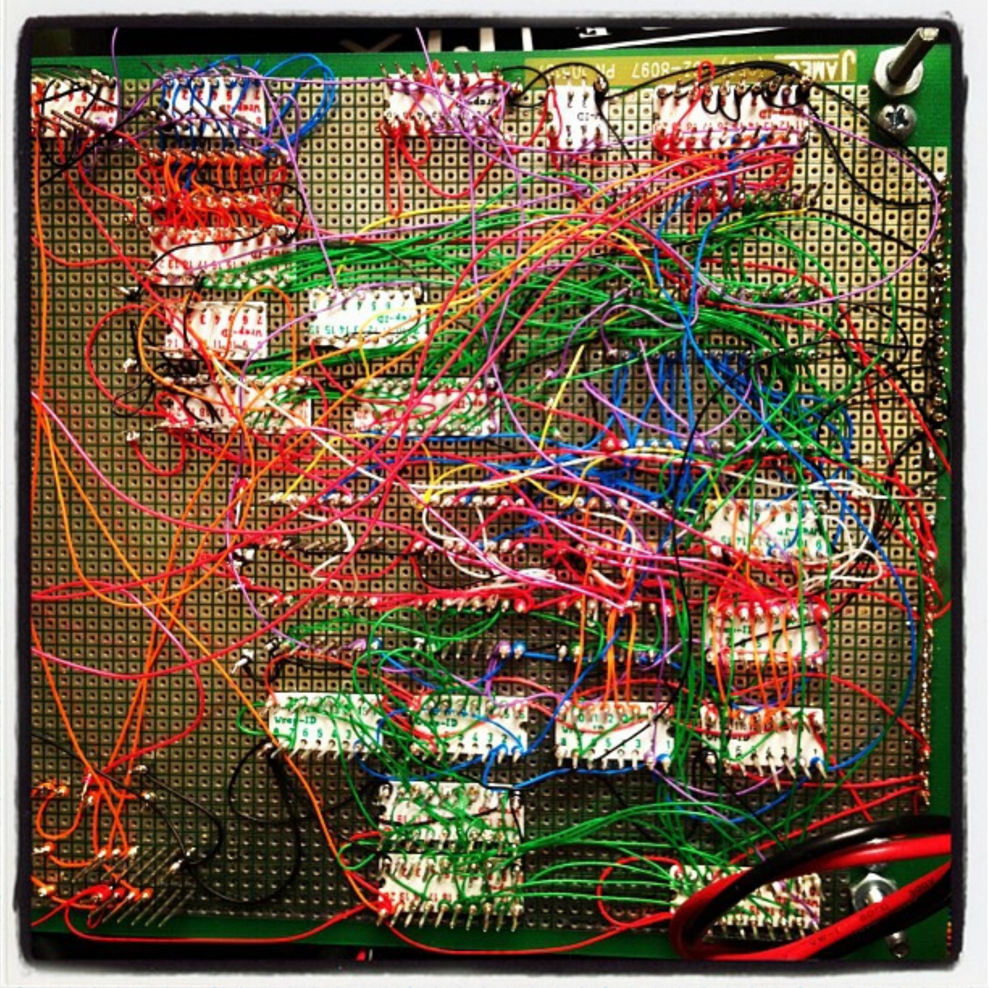


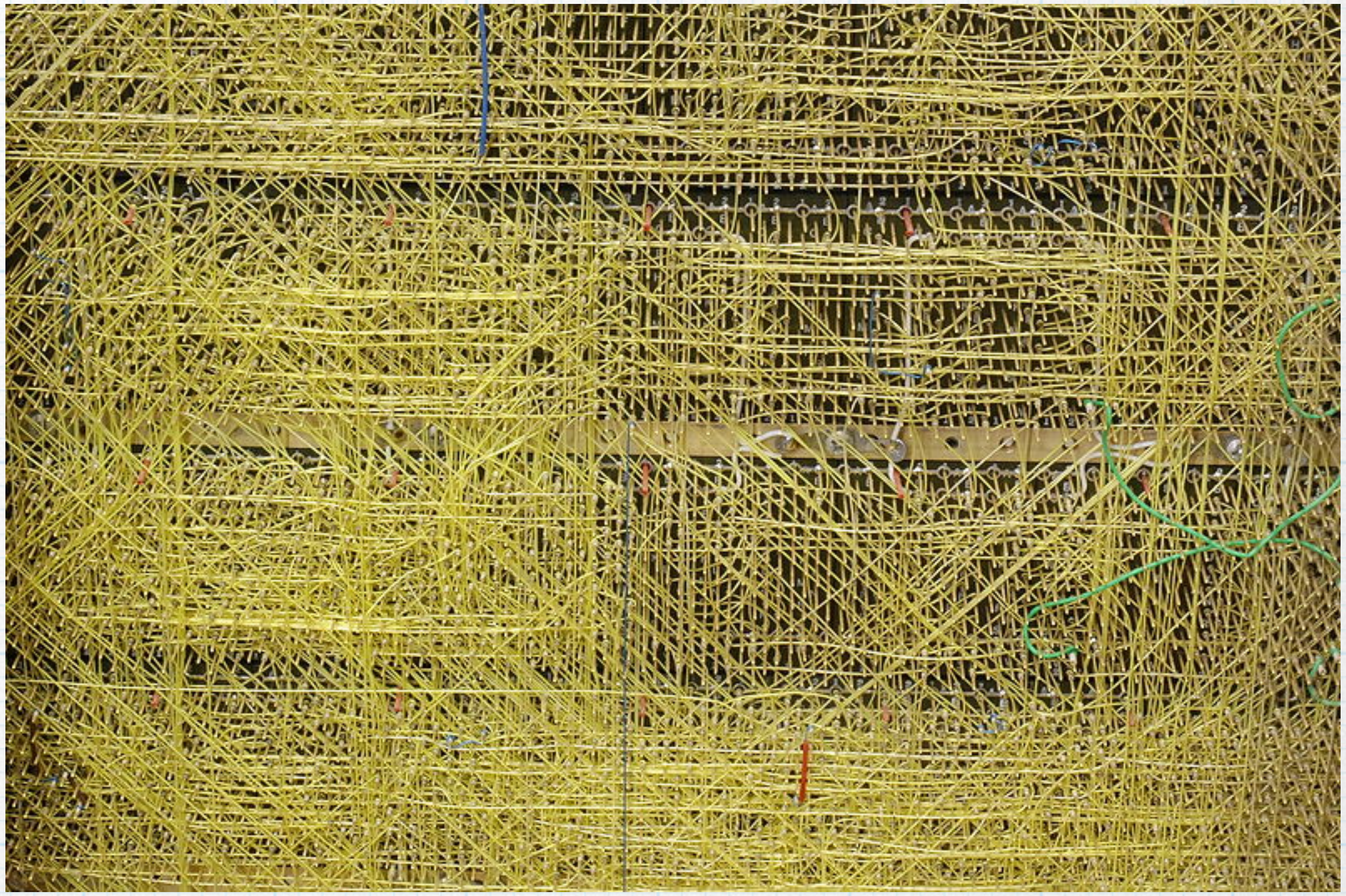


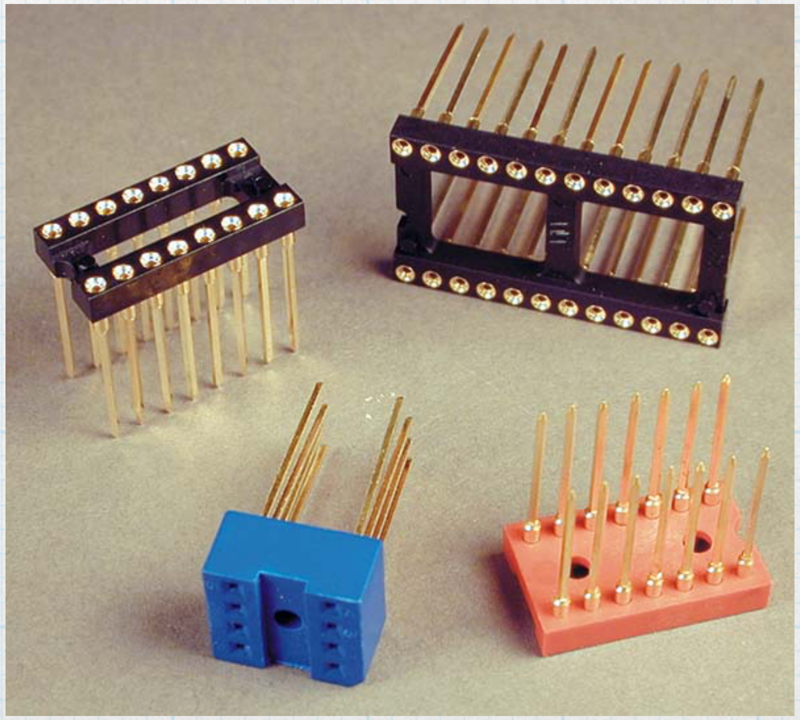




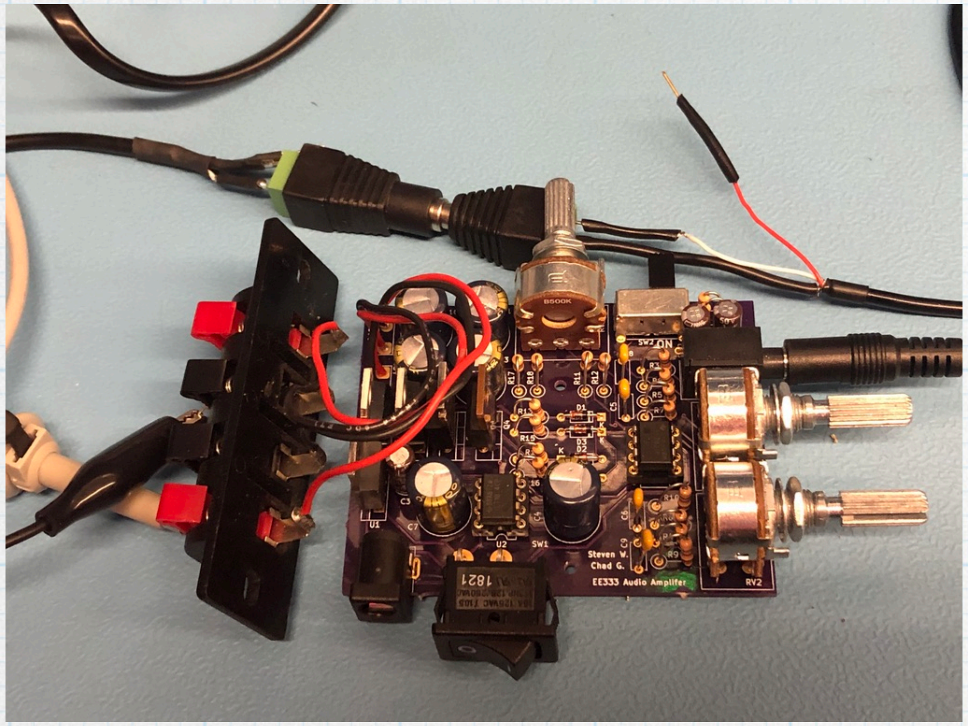
Wire wrap – no solder required, but what a tangle

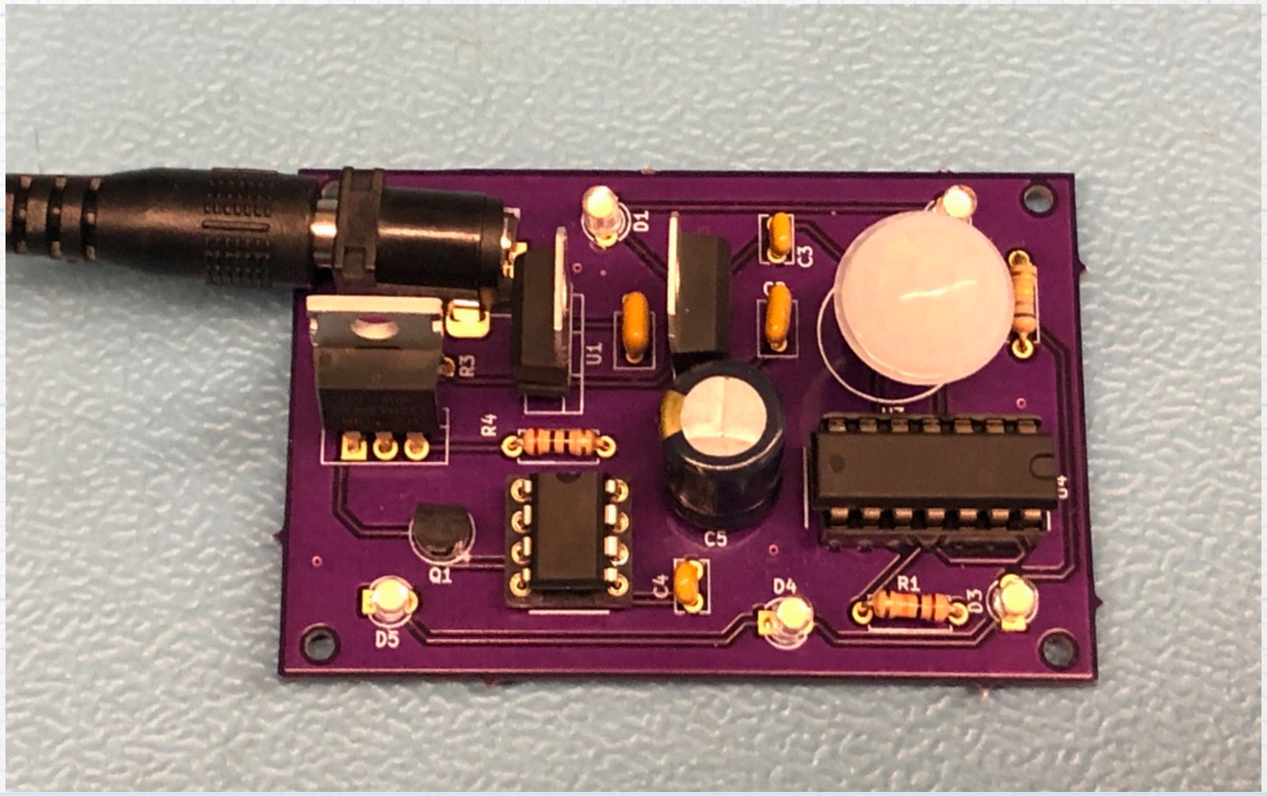


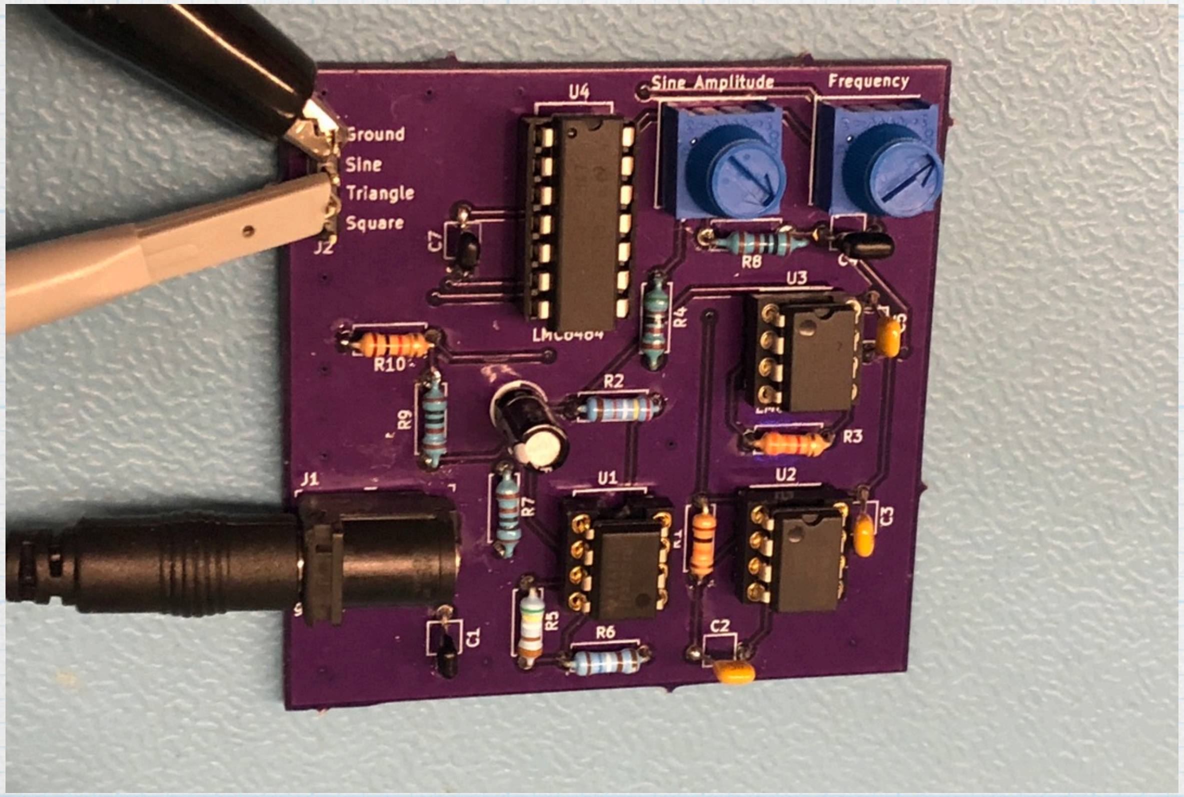


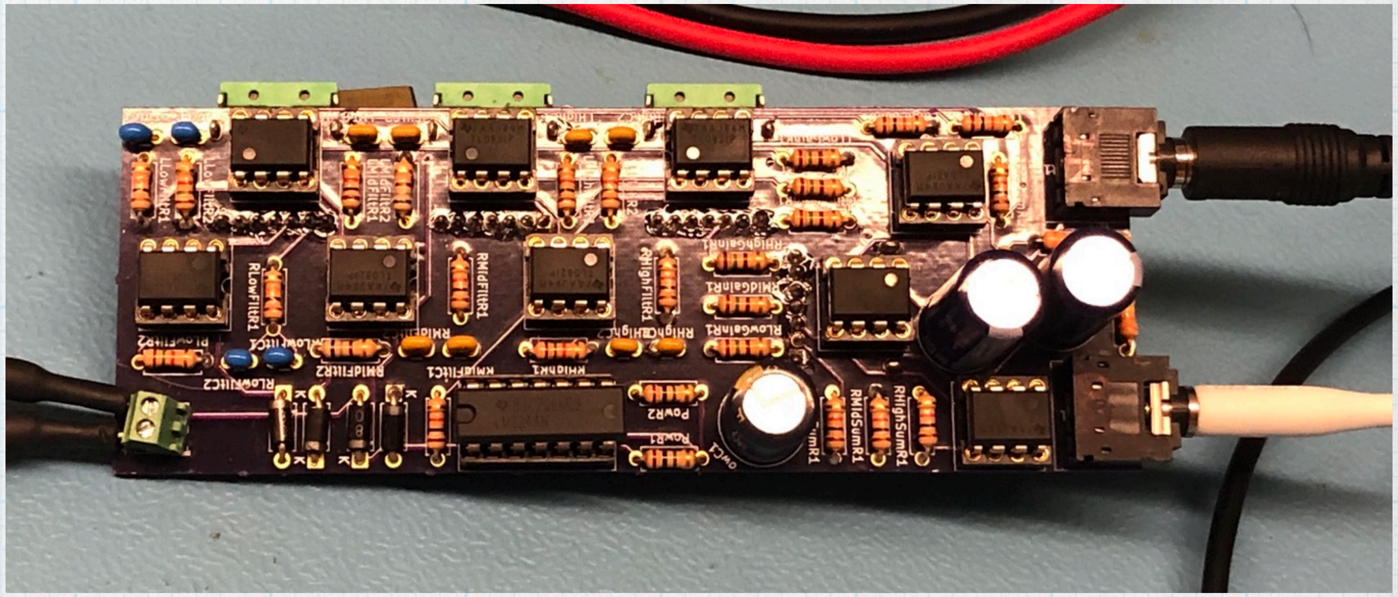


Printed circuit boards (examples from EE 333)









A comment about units

The International System of Units (generally known as metric units) are standard in most scientific and engineering situations. However, in the PCB business, measurements of physical length are still often expressed using Imperial units — board sizes, component dimensions, component thicknesses, trace dimensions, and separation between pins are given using inches and mils. When defining the features of a PCB, it is essential to know which units are being used. In the U.S., imperial units are common — a we will generally use inches and mils in EE 333 projects. In Europe and the rest of the world, millimeters are standard.

- 1 inch = 25.4 millimeters.
- 1 inch = 1000 mils (1 mil = 1/1000 of an inch.)
- 1 mil = 0.0254 millimeter = 25.4 micrometers.

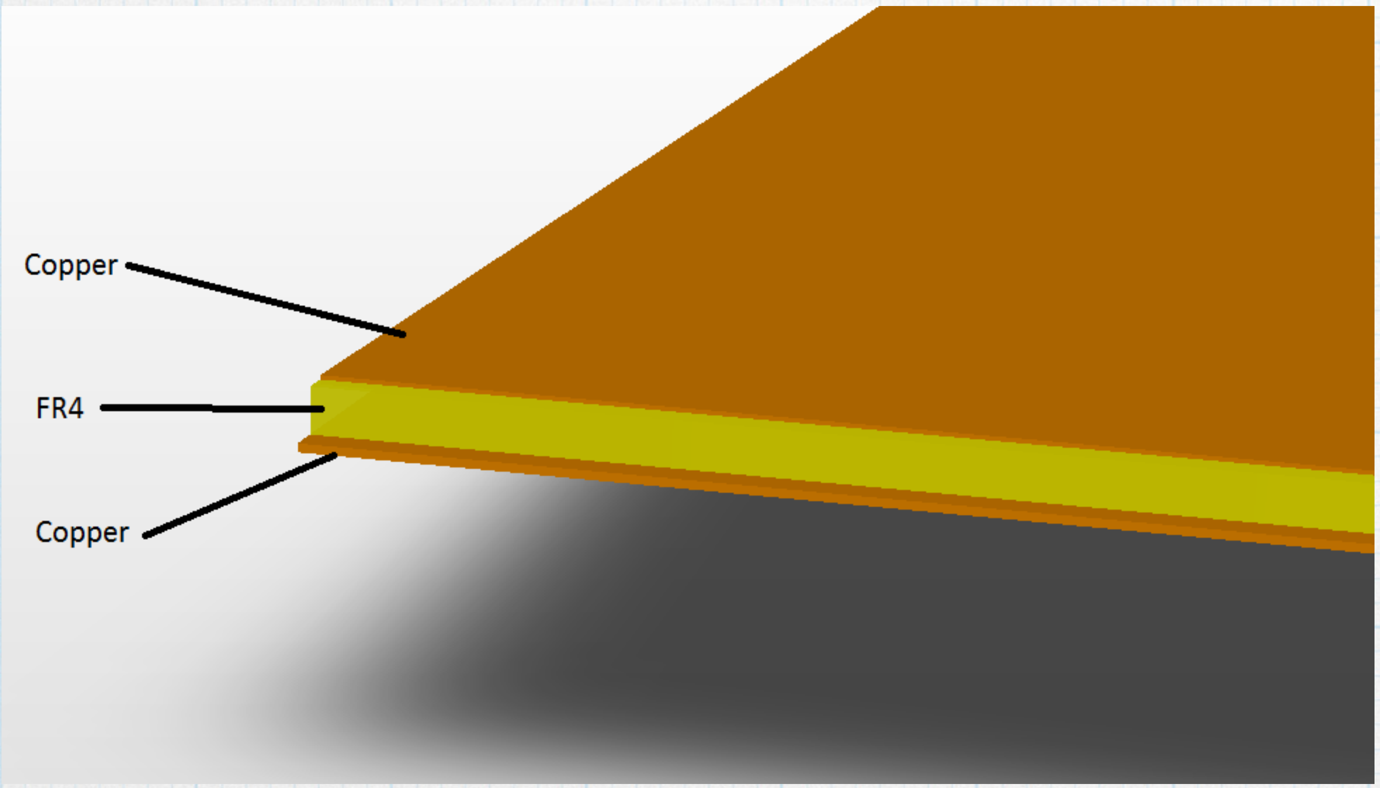
PCB structure - insulator

The basic insulating substrate is made using “FR-4”, which is an industry standard glass-reinforced epoxy laminate (fiberglass). FR-4 is a particular *flame retardant* composition that is the most widely used PCB material.

- FR-4 is mechanically rigid with a good strength-to-weight ratio, so that it serves as a solid substrate for holding the circuit components. It is easy to cut and drill holes without compromising the structural integrity.
- It is a good insulator. (Resistivity ≈ 10^{13} Ω·cm. Dielectric constant ≈ 4.4.)
- It is stable in a variety of atmospheric conditions. In particular, it does not absorb water easily.
- It can be made in a variety of thicknesses. The most common thickness for PCBs is 0.0625 in = 1/16 inch = 1.59 mm , but thinner and thicker boards are possible. Extremely thin layers can be used for flexible PCBs.

PCB structure - copper conductor

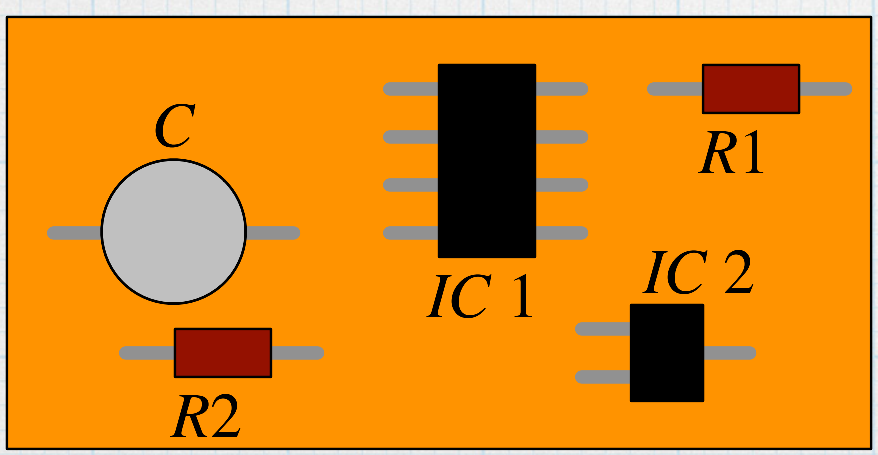
To make the PCB starting structure, the FR-4 insulator is laminated with layers of copper, top and bottom. Sometimes, this is known as the copper cladding. Sometimes only one side is coated with copper.

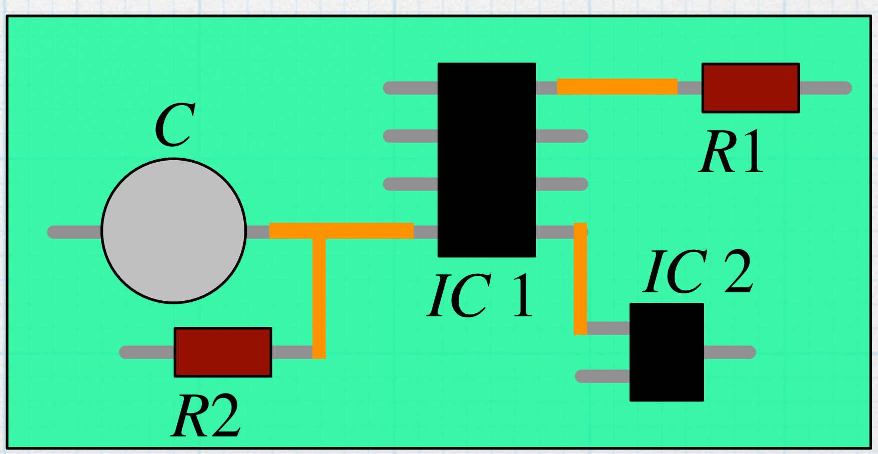


The thickness of a typical copper layer is "1 ounce". (Wait…what?) This number corresponds to the the weight of copper per unit area. "1 ounce of copper" means 1 oz. of copper per square foot. Using the density of copper (8.96 grams/cm^3) and working backwards, we find that 1 oz. of copper corresponds to a thickness of 1.344 mils = 0.034 mm (or 34 microns). Copper layers of 0.5-oz. (0.672 mil) and 2-oz. (2.688 mil) are also common.

PCB structure - traces

The copper/FR-4/copper sandwich is the starting point for making the PCB. The components are be attached to the surface of the board — in principle they can be on either side. The connections between the pins of the components are made by forming patterns — traces — in the copper between the component pins. The traces are made by removing the unwanted copper. This is known as a "subtractive process" and is very similar to how patterns are made in an integrated circuit. The pattern of copper traces define the circuit. Note that the traces on the board are defined *before* the components are put in place.





Trace resistance

In the idealized world of EE 201, wires were "perfect", having no resistance or inductance. This made calculations simple. In real PCBs, we must deal with the fact that copper is not ideal — it is a very good conductor, but the resistivity is not zero. Every copper trace will have some non-zero resistance. The traces must be sized to ensure that these "parasitic" trace resistances do not affect the operation of the circuit. This is a particular concern with traces that carry high currents, like output connections, power supply leads, and ground connections.

Resistors carrying current heat up. A trace that has significant resistance and lots of current can become very hot. This can be a real problem, particularly if the trace is "buried" in an internal layer of a PCB. A hot trace can burn out, ruining the circuit, or worse, it might start a fire.

PCB traces can be treated as simple rectangular resistors. Using the classic resistor formula:

$$R = \frac{\rho L}{A} = \frac{\rho L}{W \cdot t}$$

The resistivity of copper is $1.7\text{x}10^{-6}\ \Omega\cdot\text{cm}$, and 1-oz copper has $t = 0.0034$ cm. A trace with $W = 10$ mils and $L = 1$ inch would have a resistance of about 50 mΩ — seemingly small. But if the current flowing through the trace is 1 A, there is a 50 mV drop and it will dissipate 50 mW of power. The voltage drop may affect circuit behavior. The power may cause unwanted heating.

Through holes (vias)

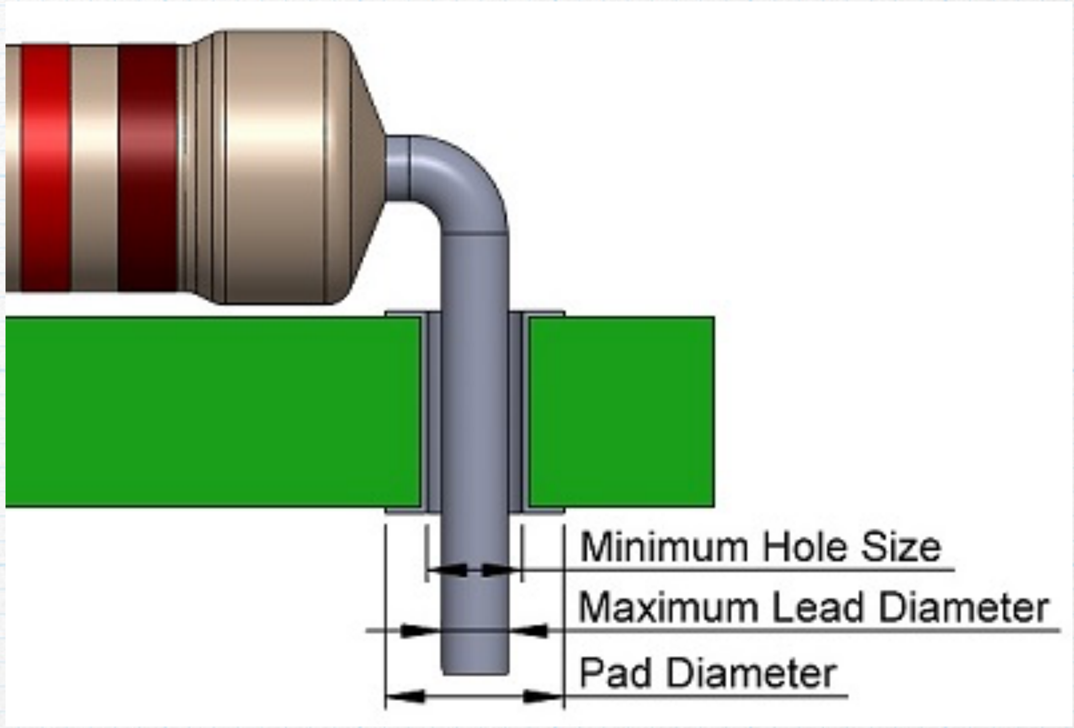
There are two reasons why it is necessary to have some holes through the PCB. The first is to provide mounting holes for “through-hole” components. (More on these shortly.) The other is provide electrical connection between top and bottom copper traces.

The holes are drilled with the standard method of spinning mechanical drill bits. The holes can be pretty much any diameter, although each manufacturer will specify a minimum limit. (10 mil = 0.01 inch in the case of OshPark.) If the via is meant to hold a component lead, it is obvious that the via diameter must be bigger than the thickness of the lead wire.

Most holes will be “plated”, meaning the sidewalls of the holes will be covered with metal, forming a connection from the top copper to bottom copper layer. Plated through holes are also easier to solder.

Through holes (vias)

Plated through hole.



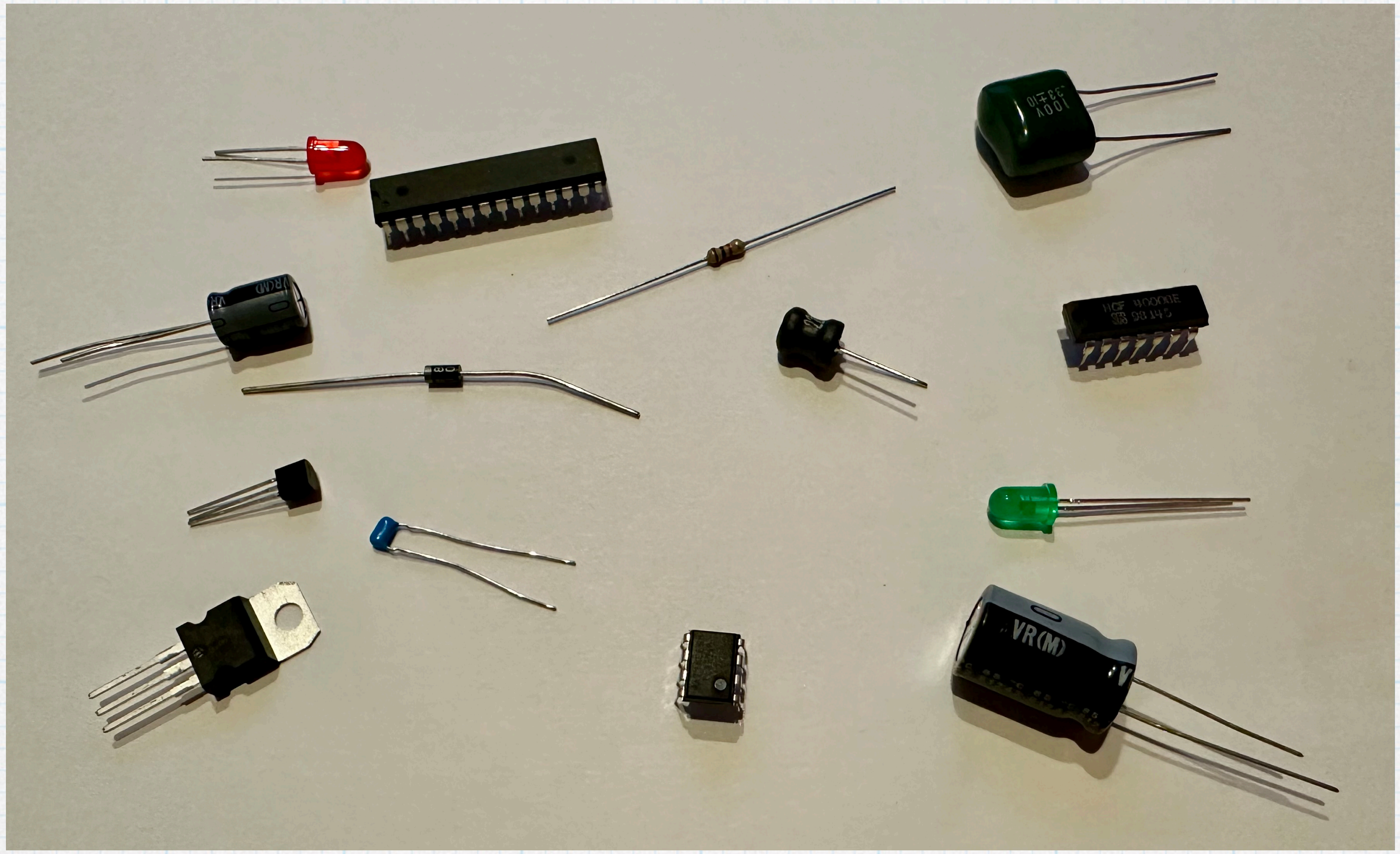
Make sure the hole is big enough for the lead wire on the component.

Fabrication videos:

https://www.youtube.com/watch?v=sIV0icM_Ujo

https://www.youtube.com/watch?v=rEB0pl8a5C0

Through-hole components



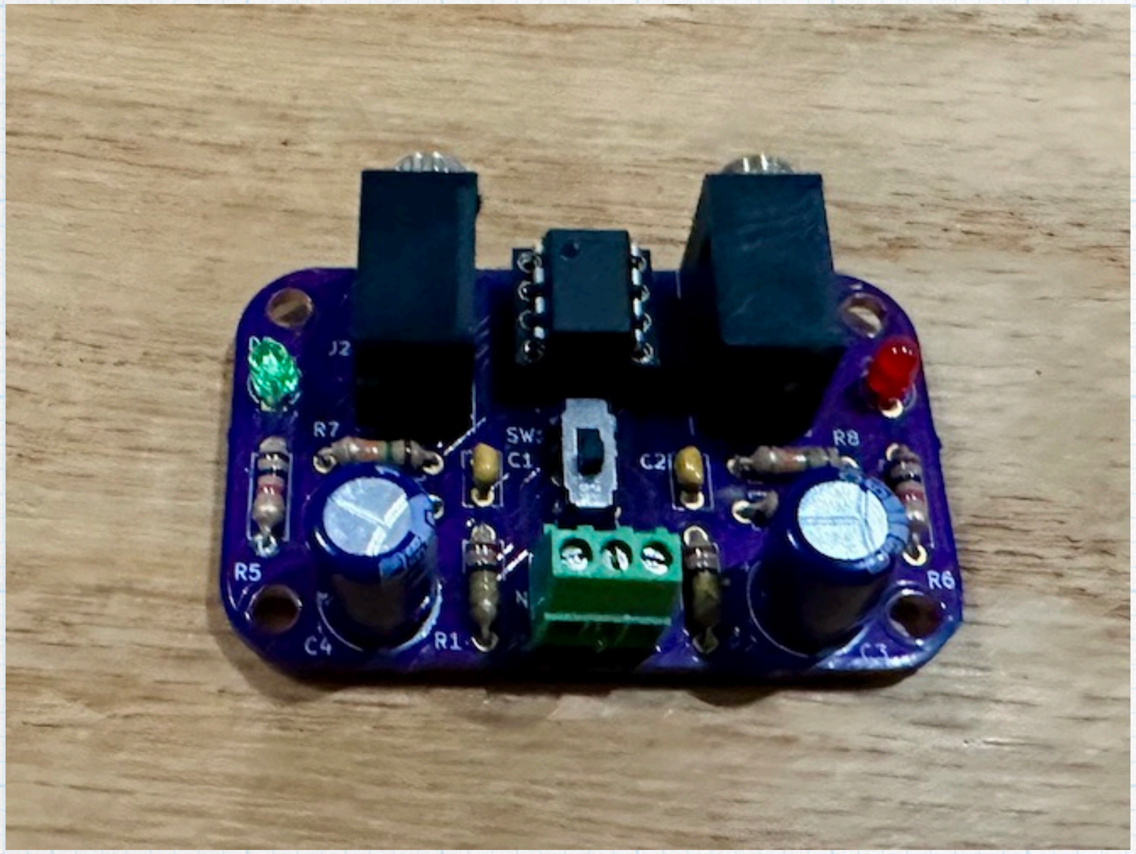
Familiar to anyone who has taken EE 201 or higher. Every component has wire leads that are used to connect to other components.

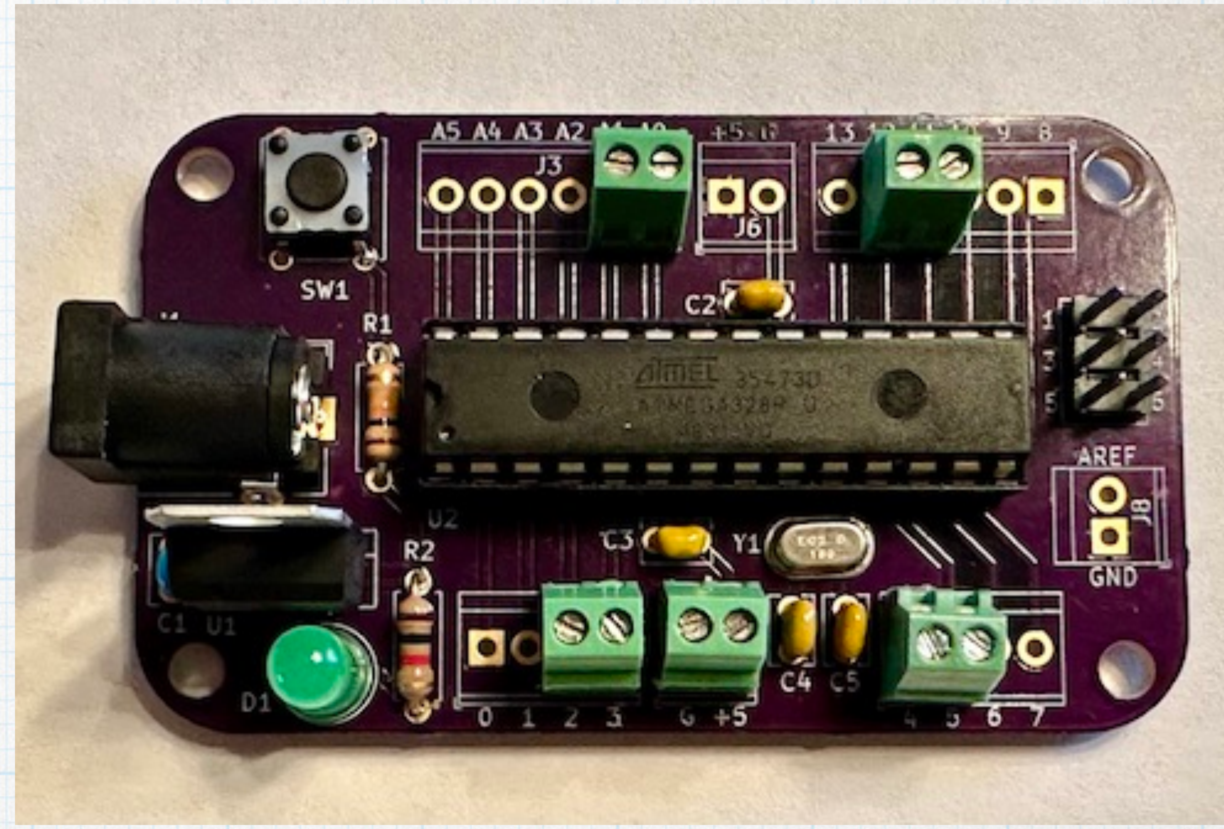
- Two-terminal components — resistors, capacitors, inductors. The leads can be axial (like resistor) or radial (like many capacitors).
- Semiconductor (diodes, LEDs, transistors, etc) have a variety of packages, depending on the number of leads.
- Chips are in “dual in-line packages” (DIPs).

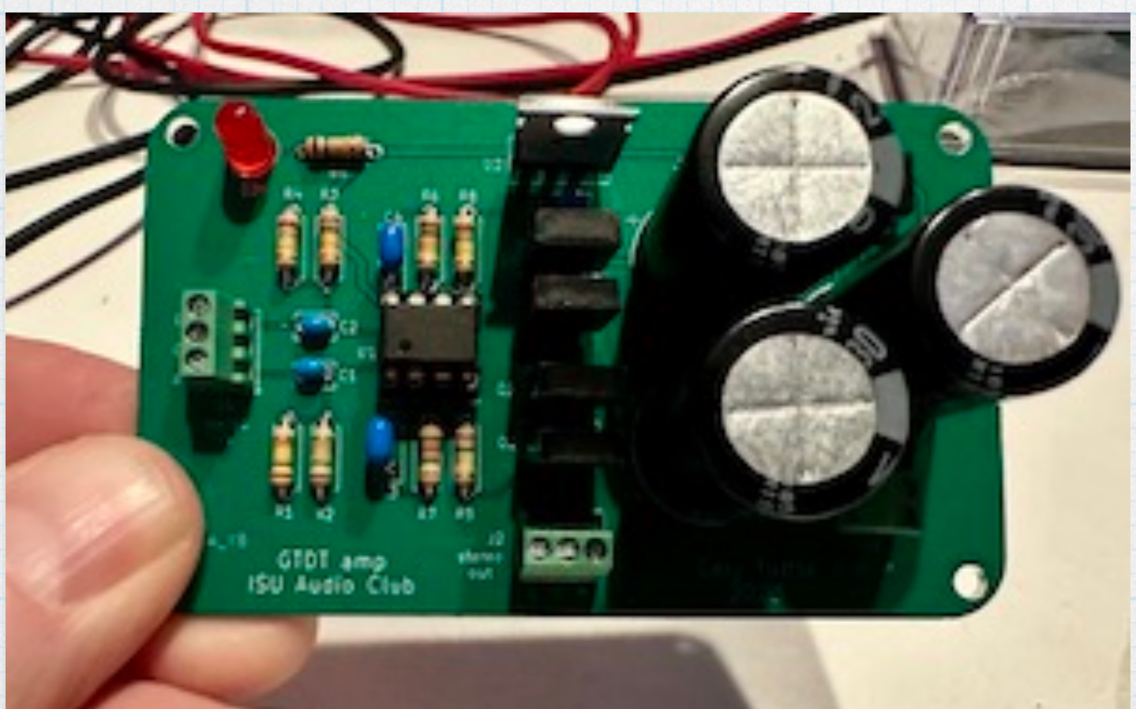
Through-hole components

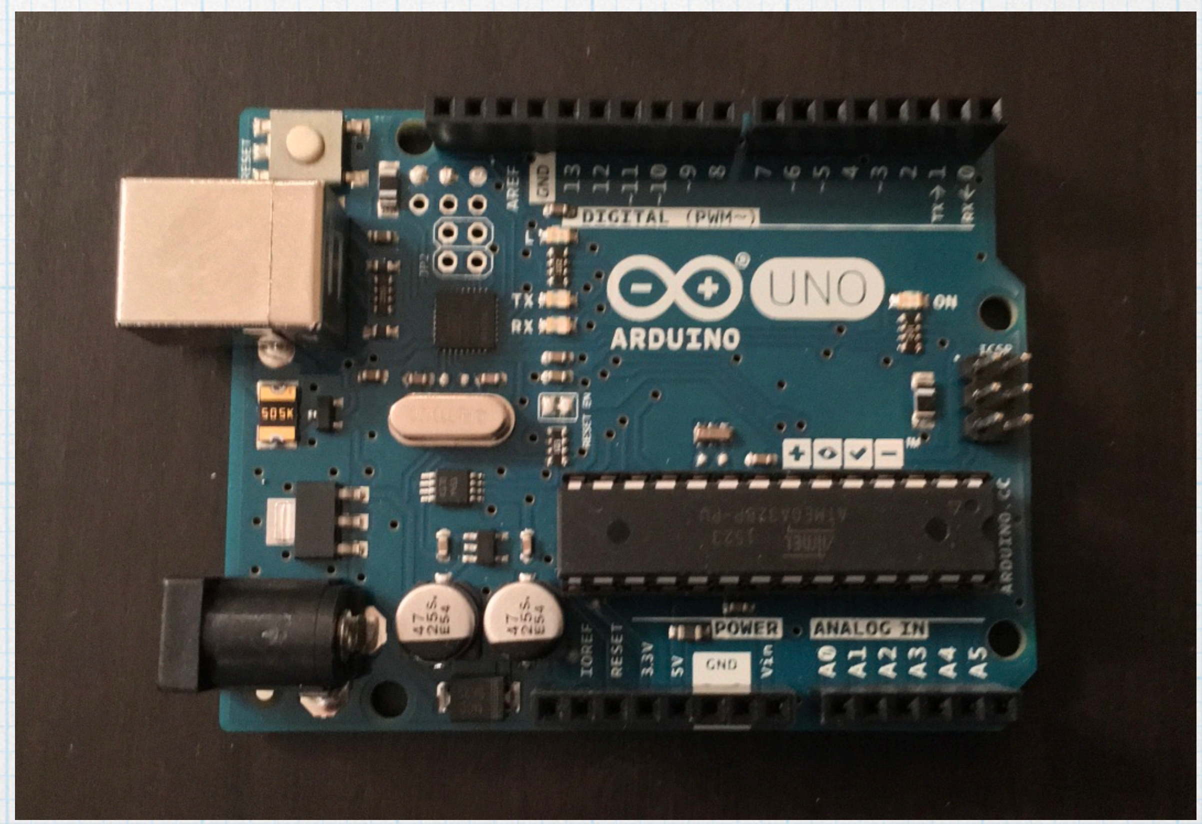
- Older technology. Most familiar components are readily available and usually inexpensive.
- Chips (DIPs) all have pins spaced 0.1 inched apart. Standard sizing makes for easy arrangement on a PCB.
- Other components also use standardized packages.
- Since the leads go through the vias of a PCB, hand assembly is easy and hand soldering is simple. Prototyping is easy.
- Automated means for assembling and soldering through-hole circuits are well developed and used throughout the electronics industry.

The major downside of through-hole technology are the sizes of the packages and the holes needed for the leads. For semiconductor parts, the actual chip is a small fraction of the size of the package. But the package size is still determined by the 0.1-in spacing and the diameters of the vias. It is hard to reduce the size of the PCB.


J2
R7
C1
C2
R8
R5
R6
C4
R1
C3


A5 A4 A3 A2
J3
J6
13
9 8
SW1
R1
C2
AREF
GND
J8
U2
C3
Y1
R2
C1 U1
D1
C4 C5
0 1 2 3
G +5
4 5 6 7


GTDT amp
ISU Audio Club


UNO
ARDUINO
DIGITAL (PWM~)
AREF
GND
13
12
~11
~10
~9
8
7
~6
~5
4
~3
2
TX→1
RX←0
TX
RX
ON
IOREF
RESET
3.3V
5V
GND
Vin
POWER
ANALOG IN
A0
A1
A2
A3
A4
A5

Surface-mount technology

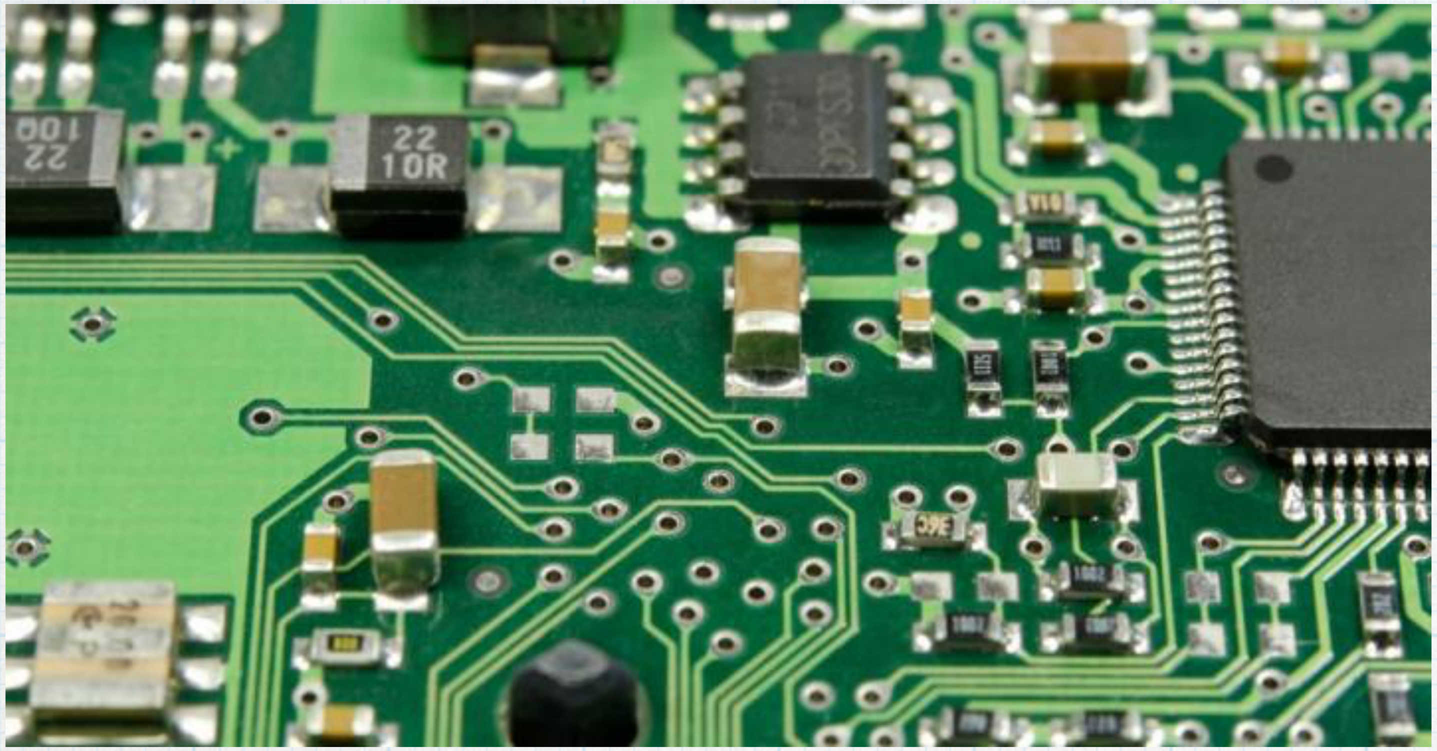
In order to shrink the PCB, the components must get smaller. To do this we eliminate the longs leads on the components and the corresponding through holes on the PCB.

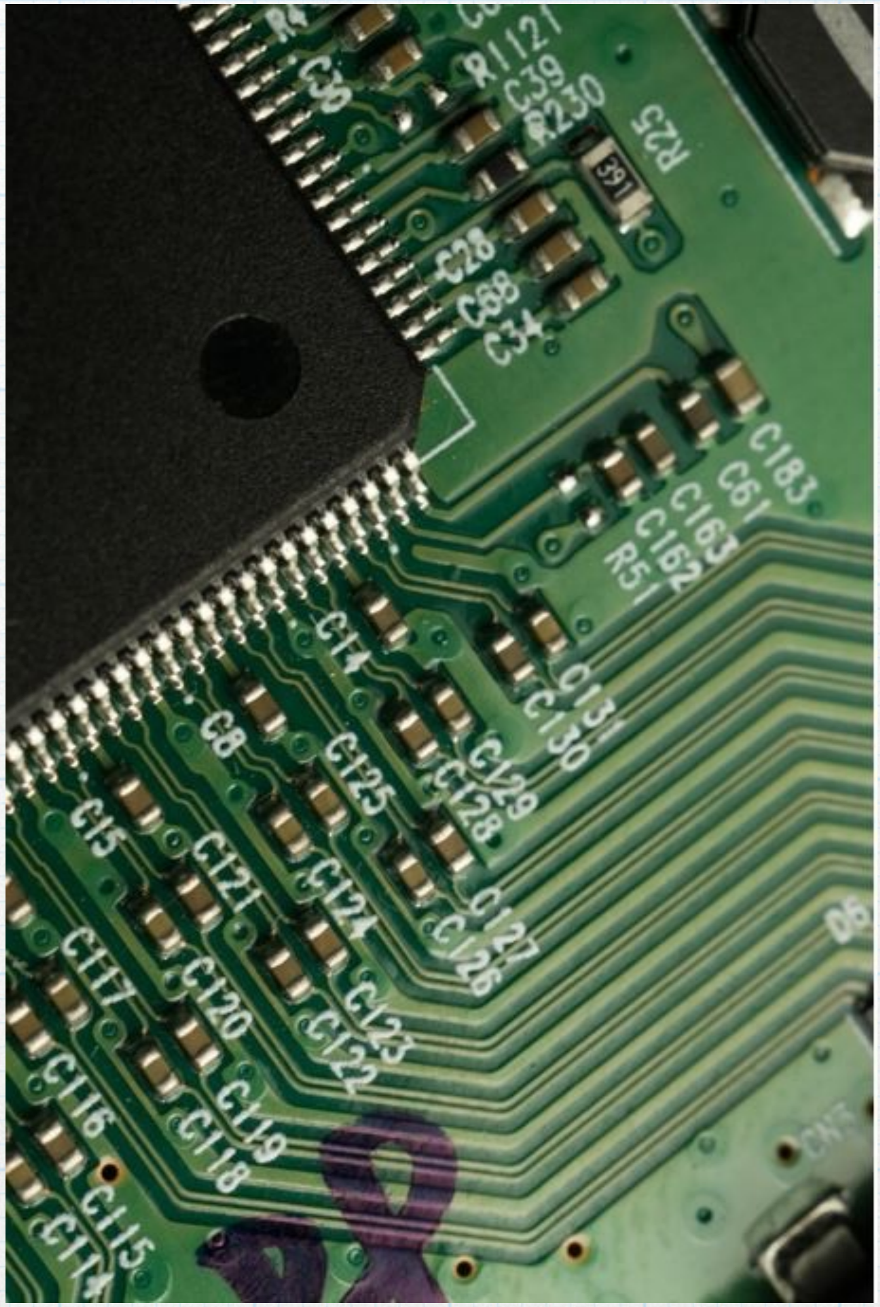
- Without wire leads, resistors and capacitors become *much* smaller.
- Without the need for through holes, the leads sticking out on the sides of the chips become much smaller. In some versions, there are not peripheral leads.
- Instead of through holes, component leads attach to “pads” on the surface of the PCB. (Hence the name “surface mount”.
- It is easier to use both sides of the PCB.

All of this together leads to a tremendous reduction in PCB size.

Another benefit is that SM components are generally cheaper than the corresponding TH versions.

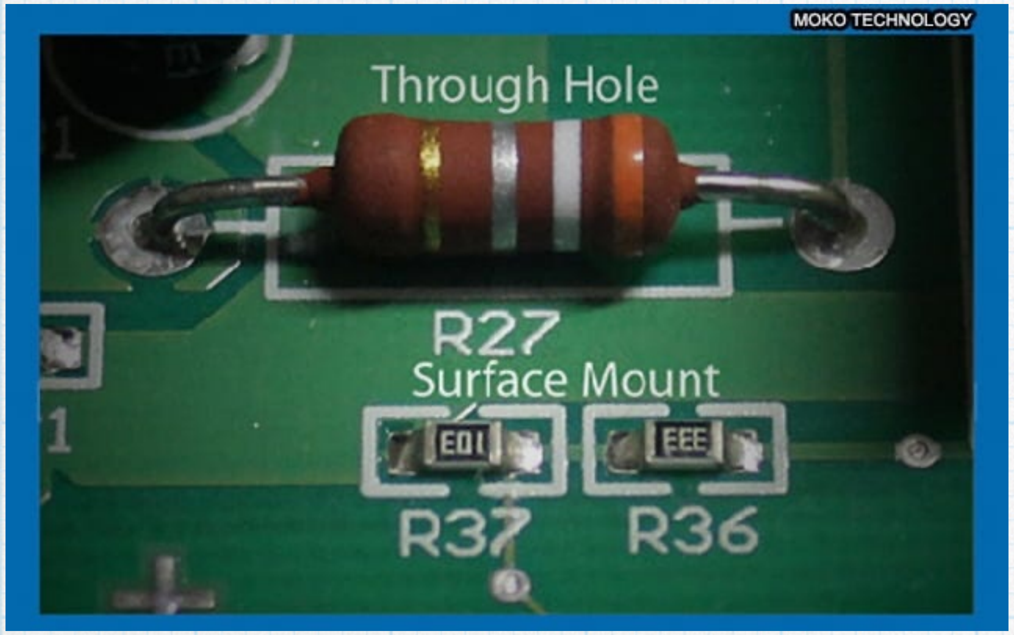
Surface-mount technology

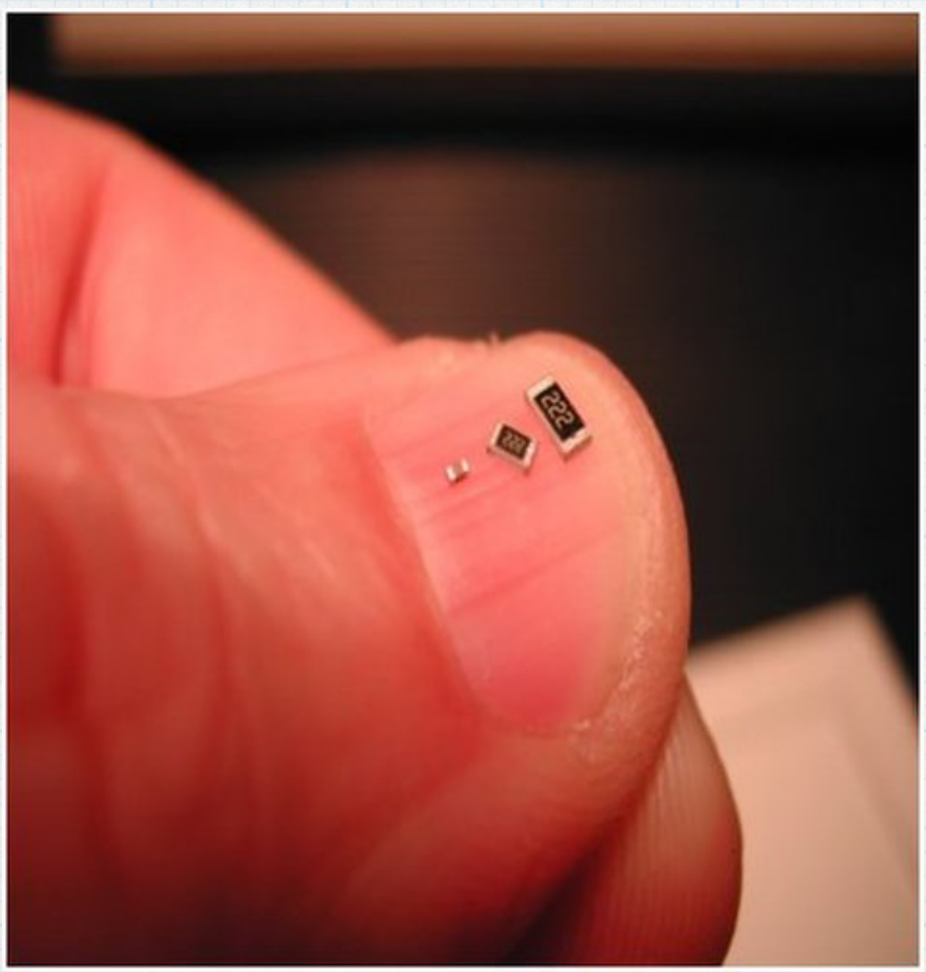


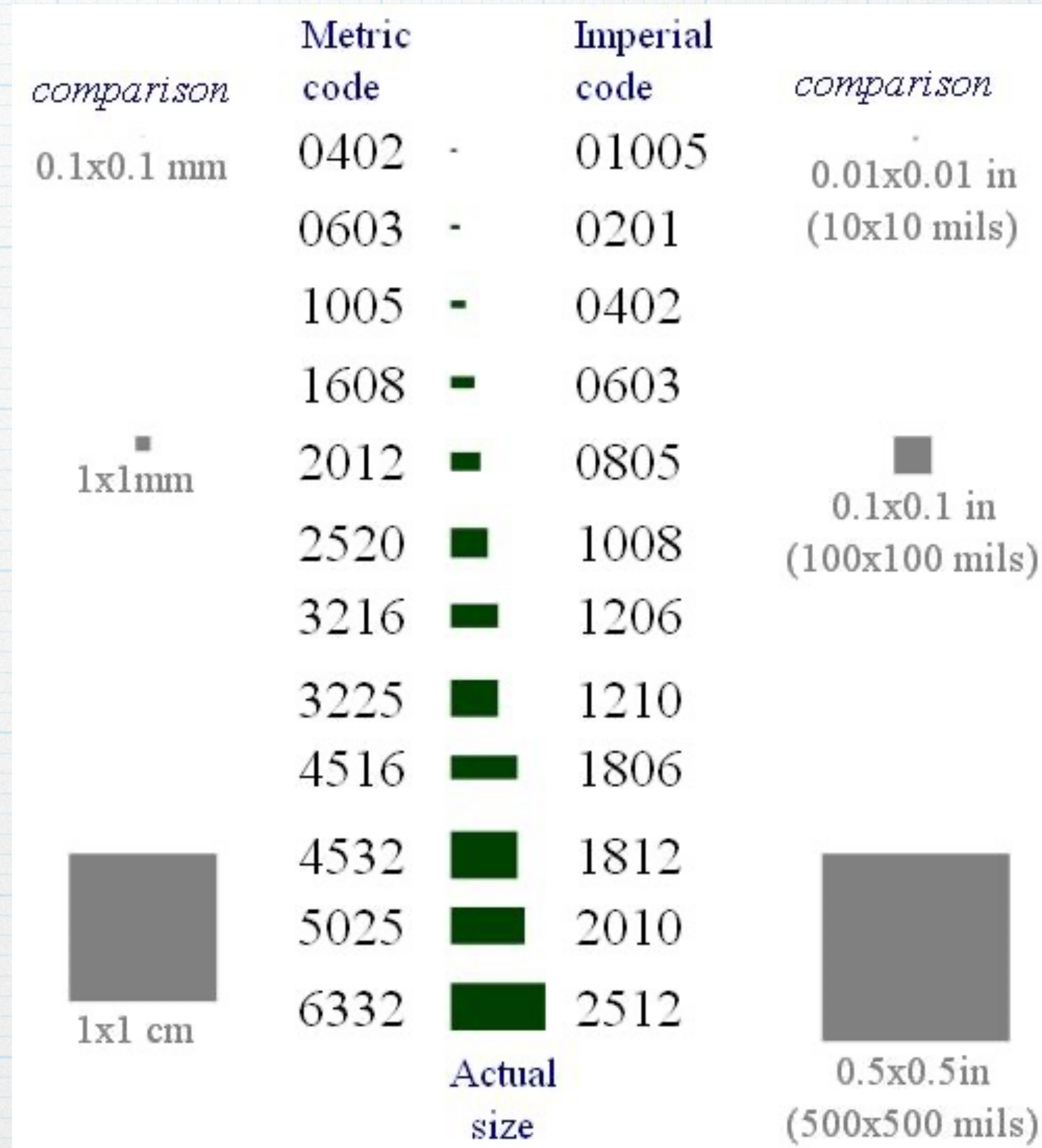


Surface-mount resistors and capacitors

Resistor and ceramic capacitor sizes are shrunk and standardized.



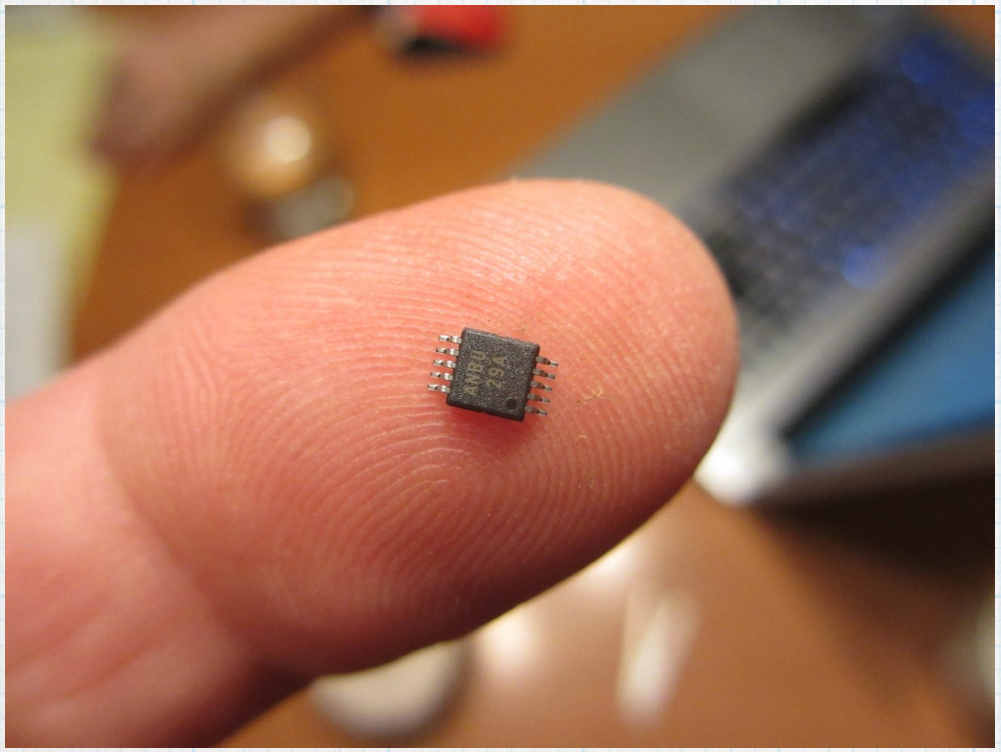




comparison	Metric code		Imperial code	comparison
0.1x0.1 mm	0402		01005	0.01x0.01 in (10x10 mils)
	0603		0201	
	1005		0402	
	1608		0603	
1x1mm	2012		0805	0.1x0.1 in (100x100 mils)
	2520		1008	
	3216		1206	
	3225		1210	
	4516		1806	
	4532		1812	
	5025		2010	
1x1 cm	6332		2512	0.5x0.5in (500x500 mils)
		Actual size		

Surface-mount chips

Chips are also hit with the shrink ray. There are a variety of standardized packages and sizes — Small Outline Transistor (SOT) and Small Outline Package (SOP).

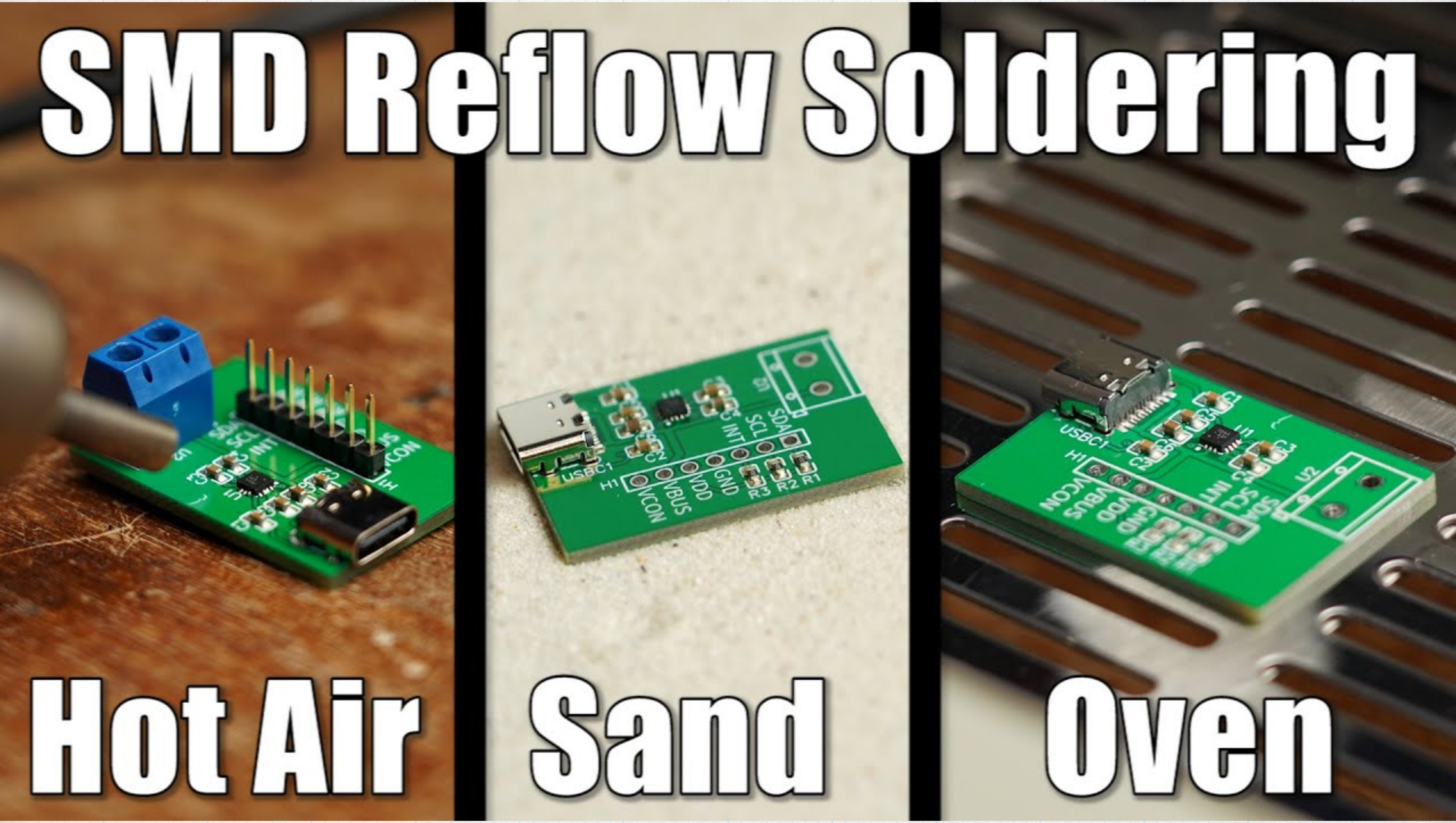


Surface-mount technology

Surface-mount technology does have one big disadvantage for us: Because everything is so small, hand soldering becomes more difficult. While hand soldering is certainly still possible — and we will try a SM project or two — SM tech is definitely less DIY friendly. Often we need to use tweezers and some sort of visual magnification in order to work with the tiny parts.

A common method for soldering SM parts is to use the “solder paste & reflow” technique. Solder paste is a combination of powdered solder alloy and flux that has a very sticky composition.

- The paste is placed on the pads of the PCB. This can be done manually using a pointed applicator or by using stencil.
- The components are placed into the paste. The sticky viscosity of the paste holds the parts in place on the PCB.
- Once all the parts are in position, the board is heated to a temperature that causes the solder to melt and the flux to evaporate. The combination of the flux and liquid solder causes the solder to “reflow” onto the pads and component leads. When the solder cools, it holds the components in place mechanically and makes electrical contact, just like conventional solder.


SMD Reflow Soldering
Hot Air
Sand
Oven