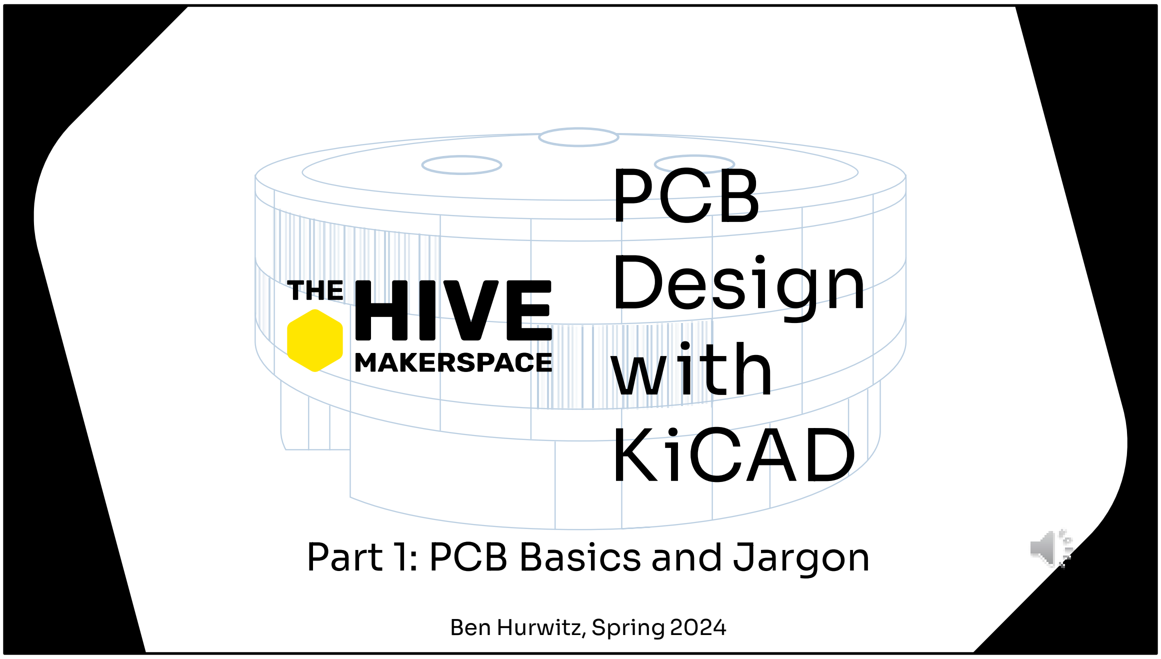
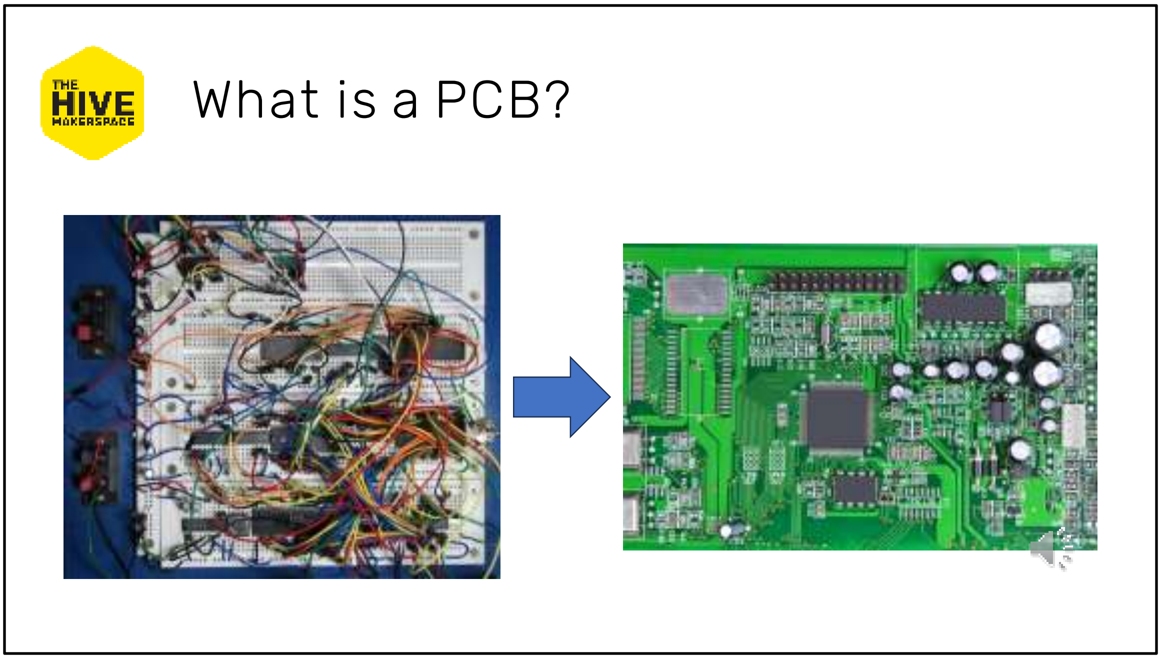
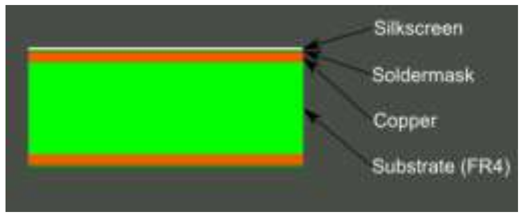


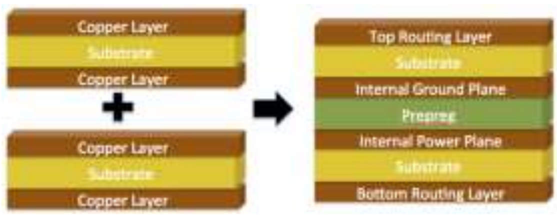
Hi, and welcome to part 1 in The Hive's PCB Design with KiCAD series. My name is Ben, and in this video, I'll be covering some PCB basics and terminology. Let's jump into it.



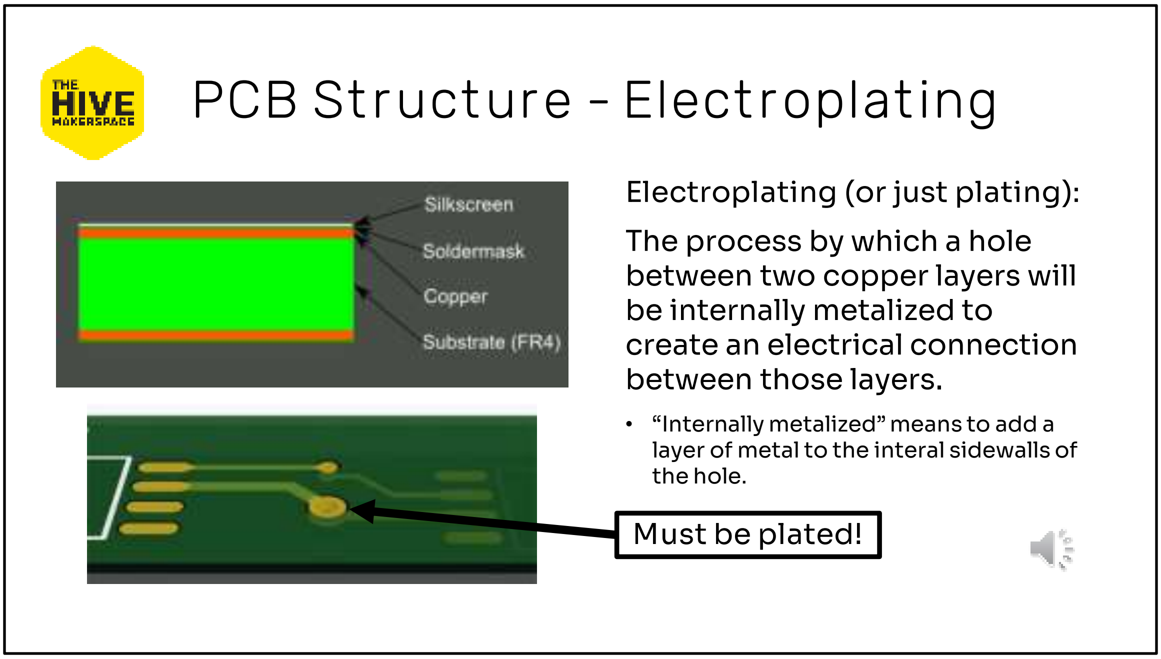
The traditional ways of connecting chips and components together has been through wire wrapping and breadboarding, but those come with a whole host of non-idealities and parasitics, are dreadful to debug, and are difficult to safely integrate into large systems. PCBs, short for “printed circuit boards” eliminate the wires and plug-in components in favor of metaphorically printing the wiring and connection points onto a flat, typically rigid, surface, and then soldering the components onto points for a mechanical and electrical connection. This helps to both alleviate many of the parasitics (though not eliminating them), as well as making the board much cleaner for visual acuity and mechanical safety and integration. PCBs are not truly printed these days, but the name remains.

PCB Structure





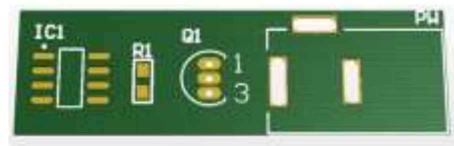
PCBs are physically composed of different layers that each fulfill different purposes. The solid non-conductive portion that forms the backing of the circuit is known as the substrate, and is typically made of a polymer-resin called FR4 (though there are many other options these days). PCB stackups (meaning the ordering and structure of the compositional layers) are generally described by the number of metal layers built in. It's most common that this metal is copper, though others are sometimes used. The thickness of the metal can vary; in the US, it's generally described by ounces per square foot, and you'll most commonly find quarter-ounce, half-ounce, one-ounce, and two-ounce weights, though others are possible, whereas internationally these would be described in microns, with half-ounce copper equating to 18 microns and full-ounce as 35 microns. Over the external copper is a layer of protective paint called soldermask, which protects the copper from oxidation, physical damage, and electrical impropriety. Finally, over the soldermask, an ink known as silkscreen may be applied to add text or graphics to either surface of the board. Additional internal copper layers may be built into the stack as well to create 4, 8, or even more layers for additional wiring area and heat conduction.

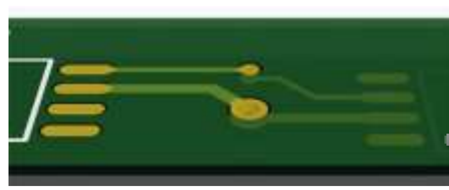


I won’t go into much detail about the fabrication process during these videos, but there is one process that needs to be understood, and that is electroplating. Electroplating described the electrochemical process by which copper can be grown onto a variety of surfaces, and is the most common way to add copper onto a substrate in the PCB world. Electroplating, or just “plating”, is most commonly heard when referring to vias. Consider a component sitting on the top side of the board. If it needs to cconnect to something that is also on the top side of the board, no problem – we just connect with a single trace (which is what we call the printed copped wire) along the surface. But what if we need to connect it to an object on the bottom side? A hole can be drilled through the substrate to make a path, but it is still non-conductive. Vias are electroplated holes that connect traces across different copper layers. These plated holes are usually quite small. Through-hole components also typically use plated holes. Some holes, such as mounting holes, may be left as non-plated through-holes.

If you’d like to know more about the actual fabrication process itself, The Hive has a basic set of tools with which you can learn a basic method of hobbyist-level fabrication, or Google to learn the nitty-gritty of production-grade manufacturing processes.

PCB Terminology

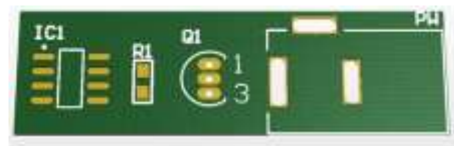


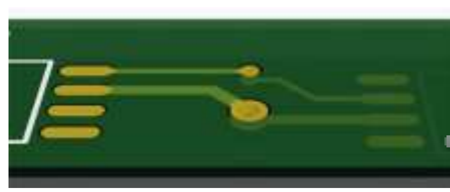


Let’s go through some PCB terminology. All of this will be in relation to the graphics on the right.

PCB Terminology

- Green is soldermask
  - Electrically isolates copper
  - Mitigates corrosion and other physical damage

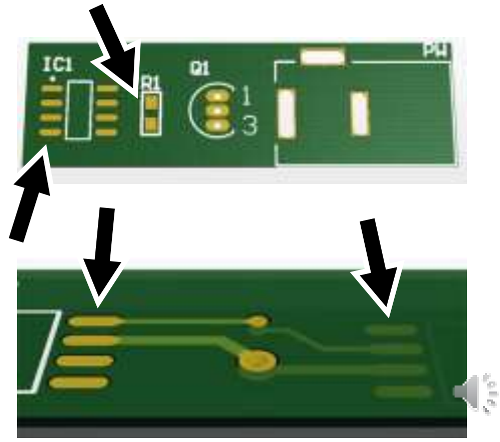




The green surface coating is the soldermask that, as I mentioned before, acts as a protective layer over any copper underneath. These days, soldermask comes in a wide variety of colors, including black, white, red, purple, and yellow.

PCB Terminology

- Green is soldermask
- In copper (gold):
  - Surface-mounted (SMD) pads
    - Connection points for surface-mounted legs/pins

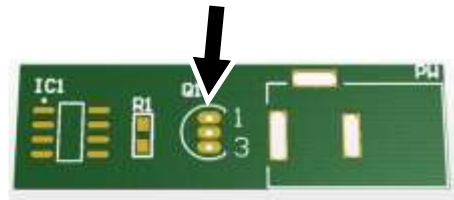


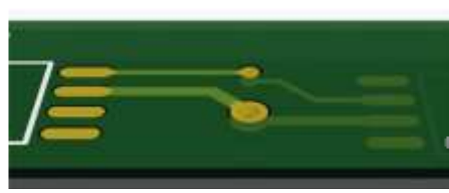
The gold is copper. All the places at which a component needs to be connected to the board will be free of soldermask to allow for soldering, but because copper oxidizes so readily in ambient atmosphere, the copper is nearly always coated in another less-oxidizable metal, a process known as finishing. Gold is a typical finish, which is why often the so-called “copper” actually is golden in color. Tin is a less-expensive common alternative.

Areas of exposed metal include surface mounted pads, where surface mounted devices connect to…

PCB Terminology

- Green is soldermask
- In copper (gold):
  - Surface-mounted (SMD) pads
  - Through-holes (TH), with rings
    - Connection points for through-hole leads, with copper rings for soldering
    - Plated by default (unless no plating available)

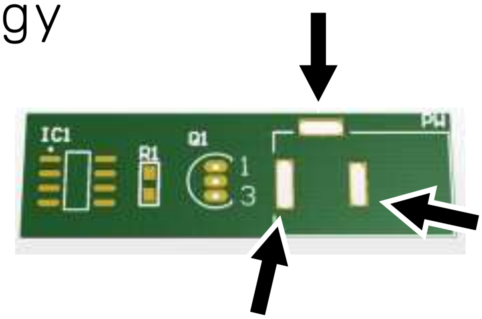


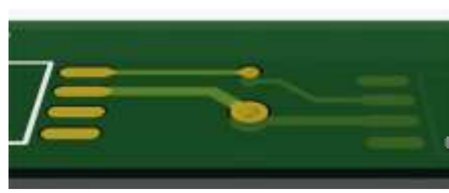


…. Annular rings around through-holes…

PCB Terminology

- Green is soldermask
- In copper (gold):
  - Surface-mounted (SMD) pads
  - Through-holes (TH), with rings
  - Slots, with rings
    - Non-circular THs
    - May or may not be plated, depending on design

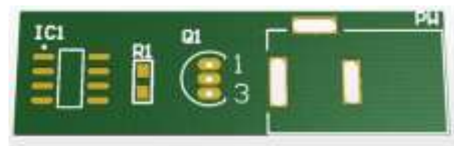


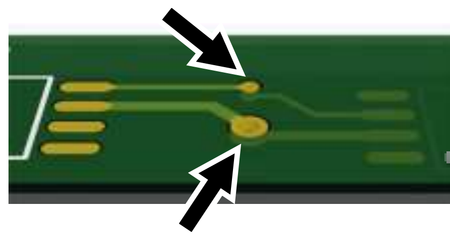


… slots….

PCB Terminology

- Green is soldermask
- In copper (gold):
  - Surface-mounted (SMD) pads
  - Through-holes (TH), with rings
  - Slots, with rings
  - Via, with annular ring
    - Connects traces across copper layers through the substrate
    - Also used for heat dissipation

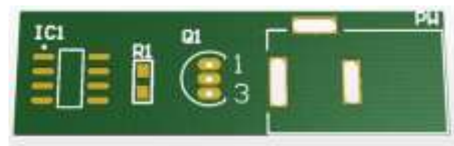


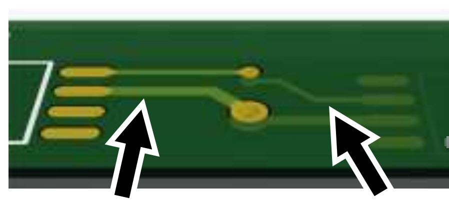


… and vias. These rings of copper allow for solder joints to be made between components and the covered traces; without them, soldering would be much more challenging. Through-holes describe any circular hole through which a component, electrical or not, is slotted. As mentioned before, these are typically plated by default. Slots are non-circular through-holes, and may or may not be plated. Vias are the inter-layer connection holes that must be plated, and can also be used for heat conduction and dissipation.

PCB Terminology

- Green is soldermask
- In copper (gold):
  - Surface-mounted (SMD) pads
  - Through-holes (TH), with rings
  - Slots, with rings
  - Via, with rings
  - Traces/tracks/routes
    - Connects pads or rings (i.e. components)





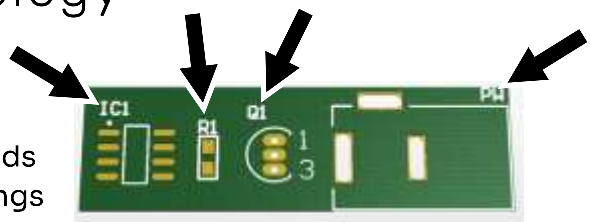
Finally, in copper, we have traces, also known as tracks or routes, that run between pads and plated holes to electrically connect components and devices together. These are usually hidden by the soldermask, but can often be seen as slight ridges pushing up from underneath.

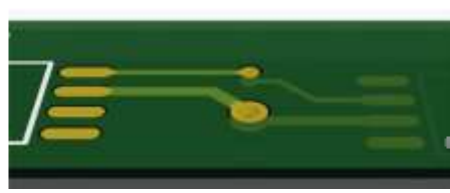
PCB Terminology

- Green is soldermask
- In copper (gold):
  - Surface-mounted (SMD) pads
  - Through-holes (TH), with rings
  - Slots, with rings
  - Via, with rings
  - Traces/tracks/routes

In silkscreen (white):

- Reference designators
  - Part identification for assembly





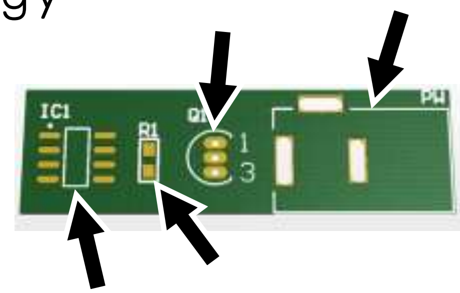
In these graphics, the silkscreen is in white, though modern fabrication houses can (like with soldermask) print silkscreen in a wide range of colors. Silkscreen is generally used to print informational text and graphics onto the board with a non-oxidizing ink.

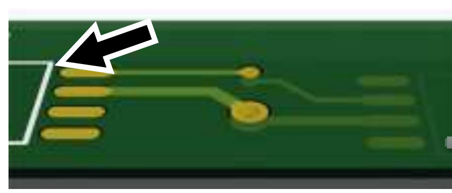
This informational text can include reference designators, which are part identifiers for assembly…

PCB Terminology

- Green is soldermask
- In copper (gold):
  - Surface-mounted (SMD) pads
  - Through-holes (TH), with rings
  - Slots, with rings
  - Via, with rings
  - Traces/tracks/routes

In silkscreen (white):
- Reference designators
- Part outlines
  - Shape and orientation; “keep-out” area for avoiding part overlap



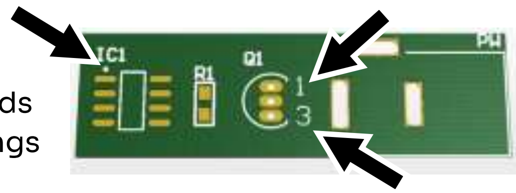


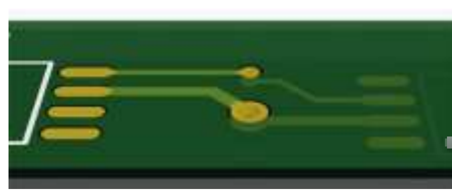
…. Part outlines used for orientation and avoiding part overlap….

PCB Terminology

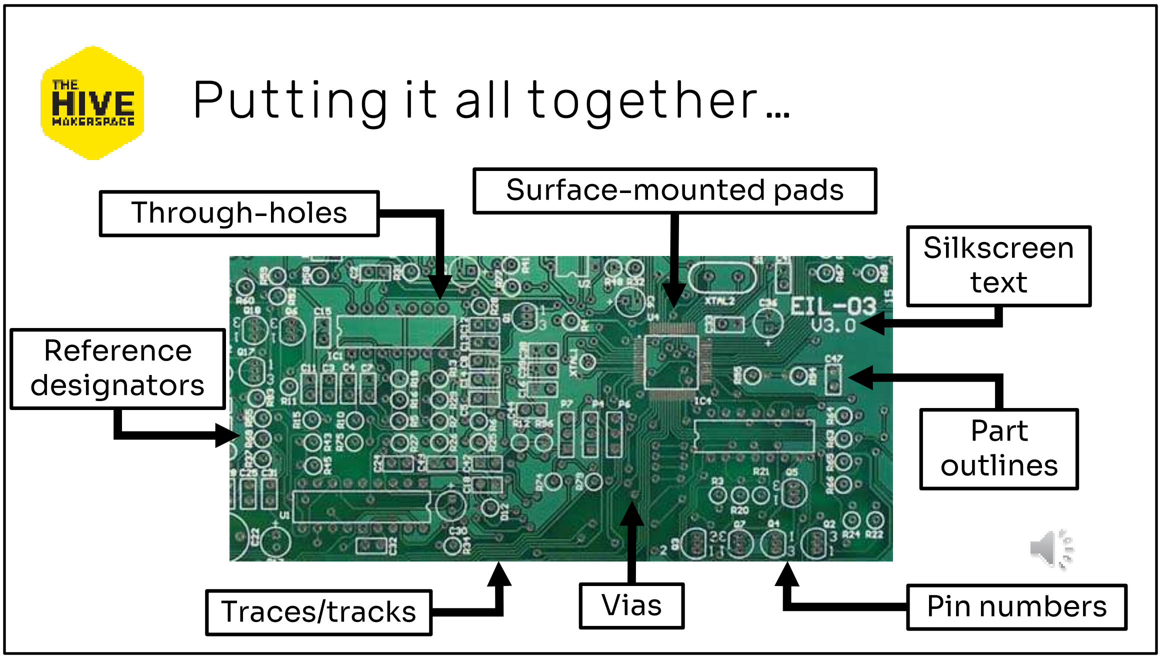
- Green is soldermask
- In copper (gold):
  - Surface-mounted (SMD) pads
  - Through-holes (TH), with rings
  - Slots, with rings
  - Via, with rings
  - Traces/tracks/routes

In silkscreen (white):
- Reference designators
- Part outlines
- Pin numbers/indicators
  - Pin function ID and part orientation



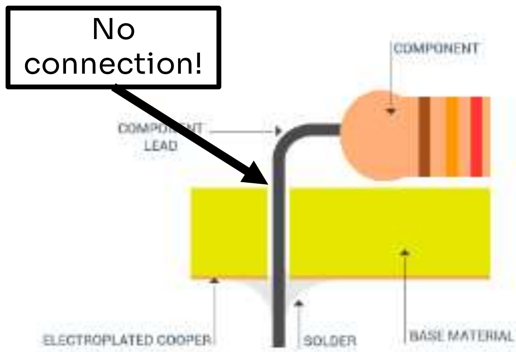


…. And pin numbers or indicators for functional referencing and part orientation. A common use-case is a small dot indicating pin one of an IC. Silkscreen is also used to identify the designer, projects, revision, year, company, warnings, and important information such as input range or mechanical restrictions.

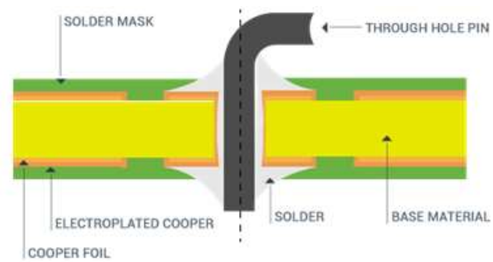


PCBs are commonly filled with these elements as identified here, with reference designators, through-holes, surface-mounting pads, text, part outlines, pin numbers, vias, and traces, all being present to enable assembly and use of the board.

Important fabrication note!

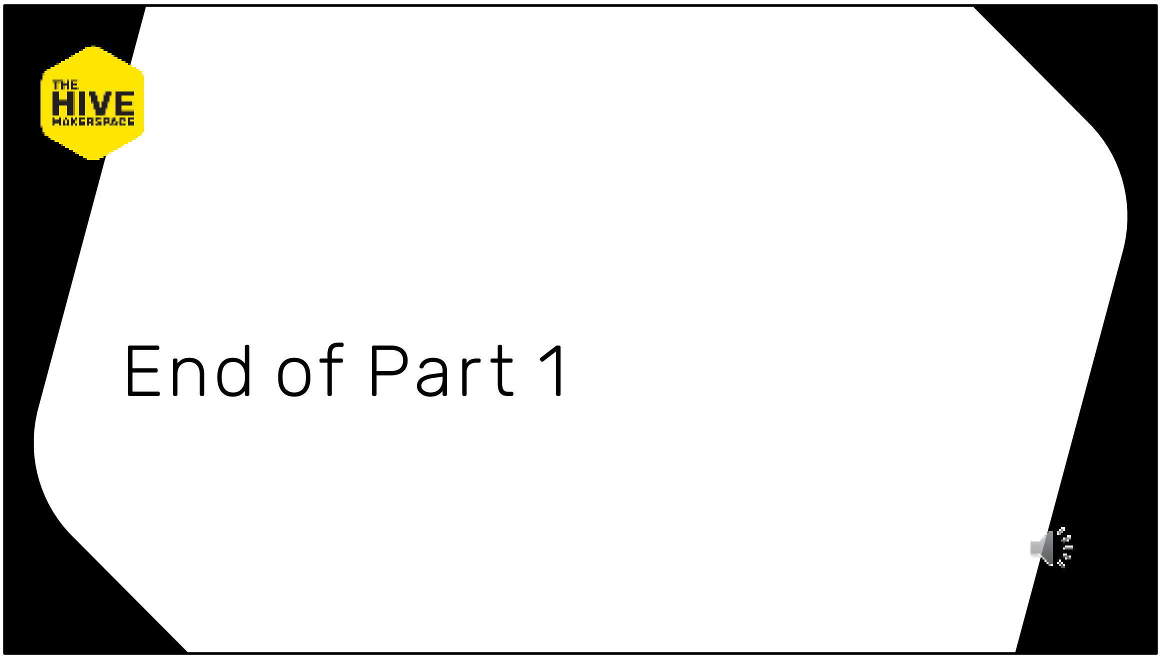


Single-sided solder



Double-sided solder

One last important note about fabrication that you should keep in mind while designing is the idea of where the component sits relative to where the traces are, and how the component will connect to the trace. Surface-mounted components, for example, require vias to connect between layers. Through-hole component have built-in holes, but if those holes are not plated, you must be aware of where the trace comes from. Consider the case on the left, a through-hole component with a non-plated hole. If the trace is on the bottom side of the board, there's not problem because the solder joint connects to lead to the annular ring of the hole (and therefore the trace). However, if the track is on the top side of the board, the component lead will not be connected to the trace because the hole is non-conducting, and the lead is not soldered on the top side. This issue can be avoided by either: 1) placing traces and components properly with non-plated through-holes to avoid this; 2) using the double-sided solder method shown on the right, if possible; or 3) always plating your component through-holes. Consider your fabrication house, cost, and time when making these choices.



And that ends part 1 of The Hive's PCB Design Tutorial with KiCAD. Today, we covered what PCBs are, and a lot of jargon and terminology surrounding their design and fabrication. A PDF of this video is available as well, linked in the description and hosted on The Hive's Wiki.

In part 2, I'll introduce PCB design software with a broad overview of how this family of software works, bring KiCAD into our lexicon, and end with a generic PCB design flow that we'll try to follow throughout the subsequent videos.

See you then!