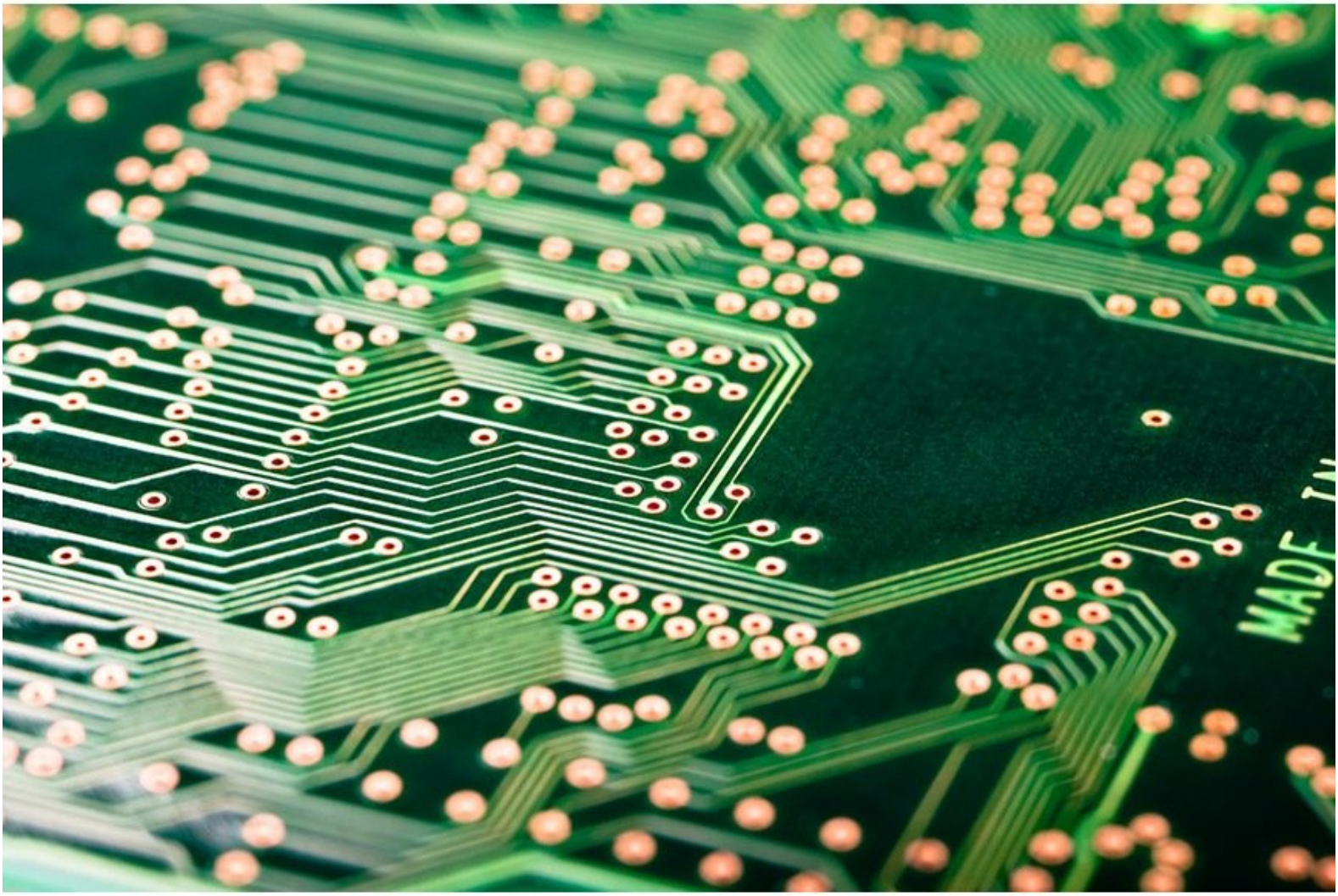
# PCB Fundamentals

Ever wondered how PCBs work? We will go over the different parts of a PCB including Vias, Traces, Through-Holes, Solder Pads, Layers, and Solder Mask

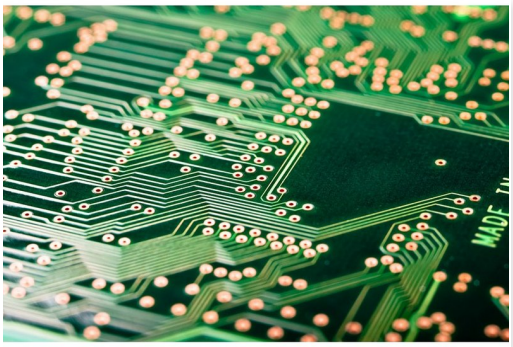
**Written By: Nathan Dillon Martin**

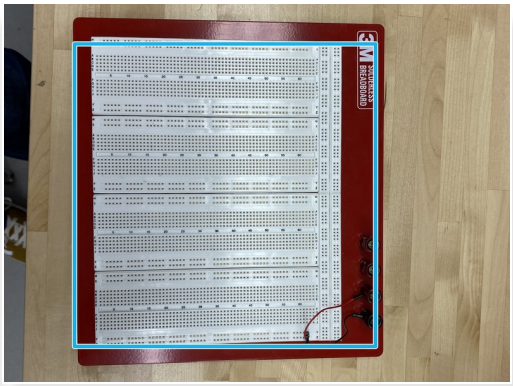
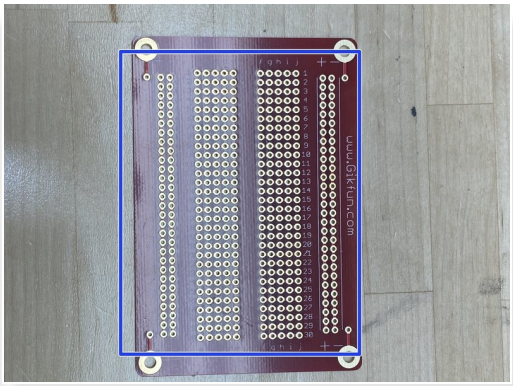




## Step 1 — What is a PCB?



- PCB Stands for Printed Circuit Board
  - PCBs can be imagined as a map or graph of the connections in an electric circuit.

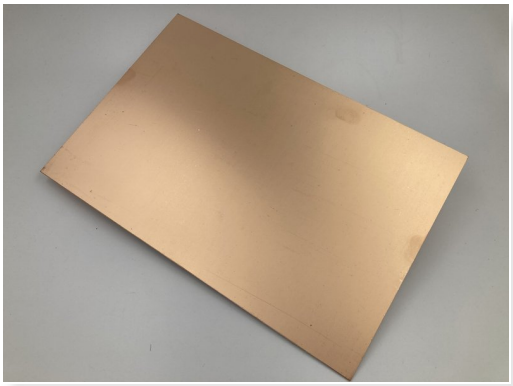
A PCB is specifically printed for each circuit to make all of the necessary electrical connections

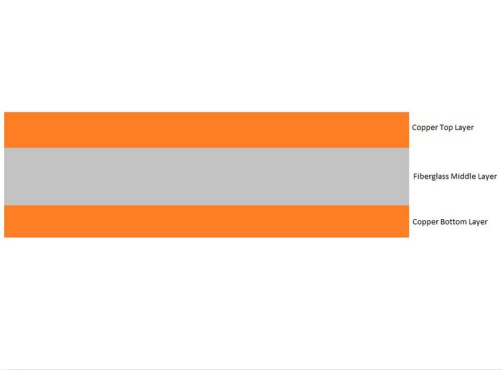
PCBs aren't the only way to accomplish this. The same result can be achieved with the following:

- Solderless Breadboards
- Prototyping Boards

Each of these solutions have benefits and downfalls. PCBs are the most professional and reliable solution. However, they are also less versatile, since a new circuit requires a whole new PCB to be made.

## Step 2 — Construction & Layers



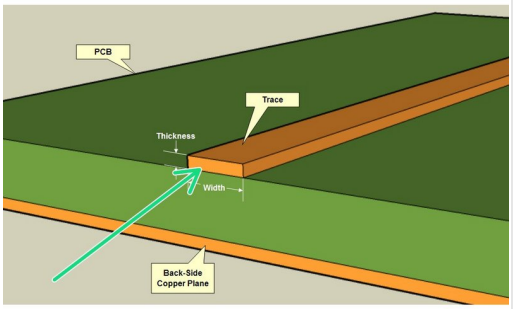


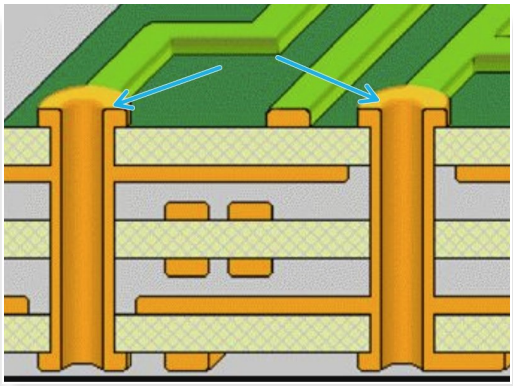


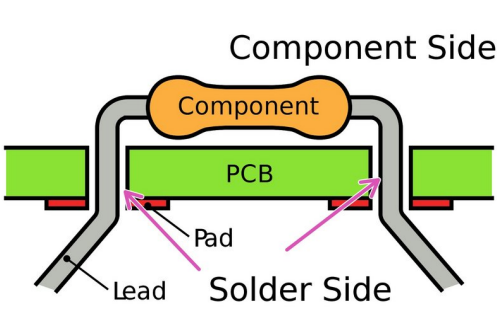
- The base of a PCB is called FR-4, which is a fiberglass core with layers of copper on either side
  - This base PCB has 2 Layers. A layer is a sheet of conductive material (copper) that can carry an electric signal.

    PCBs with more layers are created by putting alternating layers of insulator and conductor into a hydraulic press.
- Most basic PCBs will have two layers, but more complex PCBs can have up to 10 or even more!

## Step 3 — Transmitting Signals

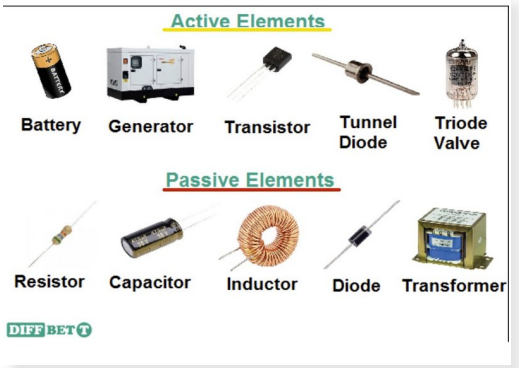


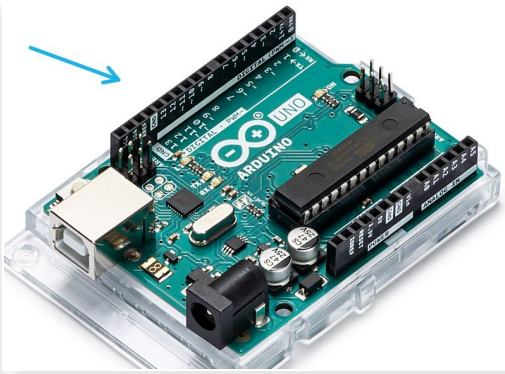



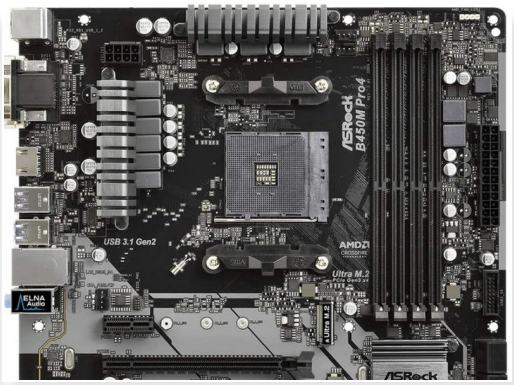



- How is the electricity transmitted?
  - ⓘ As we saw, each layer can transmit a signal across it. However, we still need a way to transmit a signal across layers and to place multiple signals onto a single layer.
- Traces: Thin strips of copper on a layer of the PCB. These link components on the same layer of the PCB.
  - ⓘ Conductive material left over after a PCB layer has been milled out.
- Vias: These are small holes that transmit a signal across different layers on the PCB
  - ⓘ Small holes that are drilled out and then electroplated with a conductor
- Through-Holes: Holes where components are soldered in. The solder electrically connects all of the layers that it passes through.
  - ⓘ Large holes that are drilled out and the electroplated with a conductor

## Step 4 — What goes on PCBs?




Components are any electrical devices that will be directly connected to a PCB. There is an incredibly varied selection of components that can be used, but they fall into two main categories:

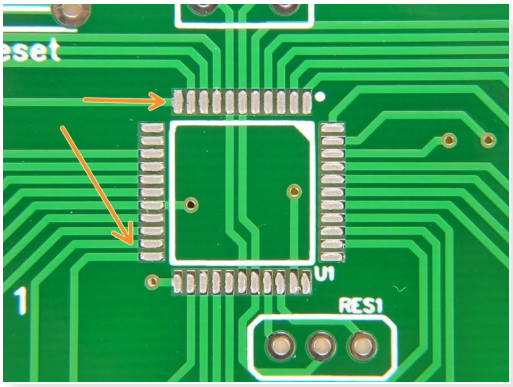
- Active Circuit Elements: Small chips with multiple input and output pins to interface with the circuit inside. Includes logic gates, op-amps, ROMs, microprocessors, and more
- Passive Circuit Elements: These are components with two terminals with specific electrical properties like a resistor, inductor, capacitor, or diode

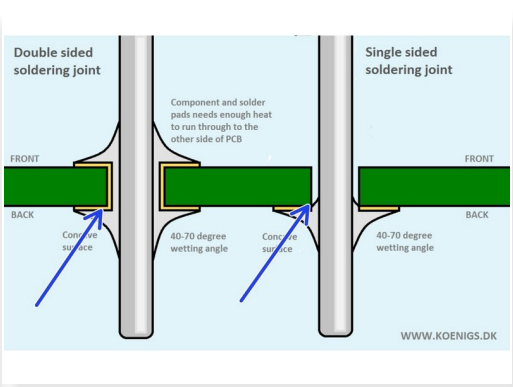
⚠ These groups are a huge simplification of the components that can actually be on a PCB, but they provide a wide foundation. Don't feel constrained by these groups if you are looking to create a PCB.

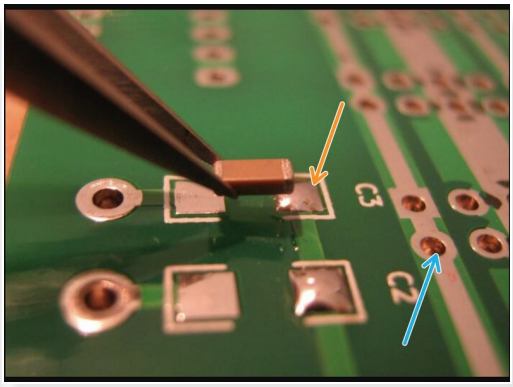
On a PCB, there can be hundreds or even thousands of connections. When circuits get this large, a PCB is the best way to package them

- Try to spot some of the different Passive and Active circuit elements on the Arduino and Motherboard

## Step 5 — Surface-Mount vs. Through-Hole



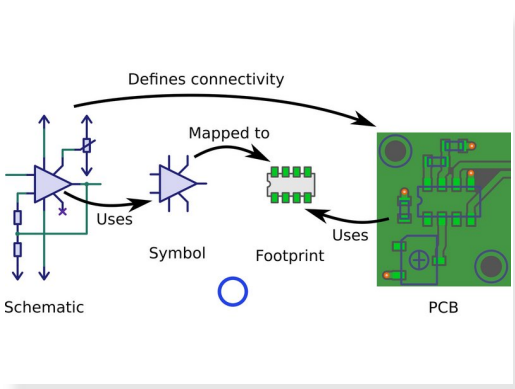



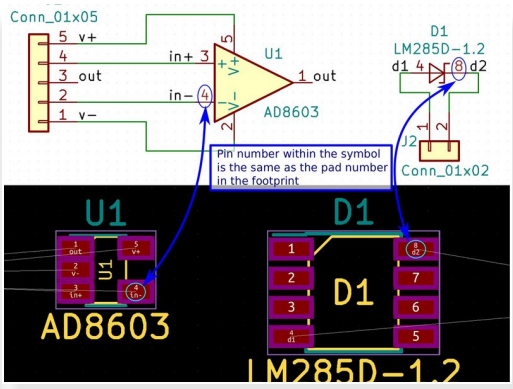
- Surface-Mount footprints (SMD) consist of solder "pads", which are small, flat, exposed pieces of copper that components can be soldered onto
  - Soldered with solder paste and a reflow oven
- Through-Hole (THT) footprints consist of through-holes that go all the way through the PCB
  - Soldered with wire solder and a soldering iron

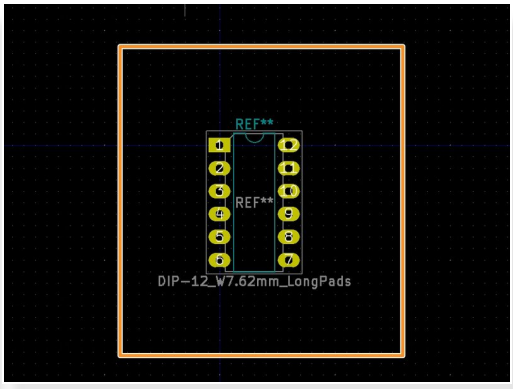
Why choose one component type over the other?

- SMD components are significantly more compact and allow for more complex PCBs to be made smaller. However, they are also more difficult to solder and require specialized equipment
- THT components are large and connect to all layers of a multi-layer PCB. This makes them worse for creating more compact PCBs, but are still a good option for larger components like connectors and pinouts

## Step 6 — What's a Footprint?

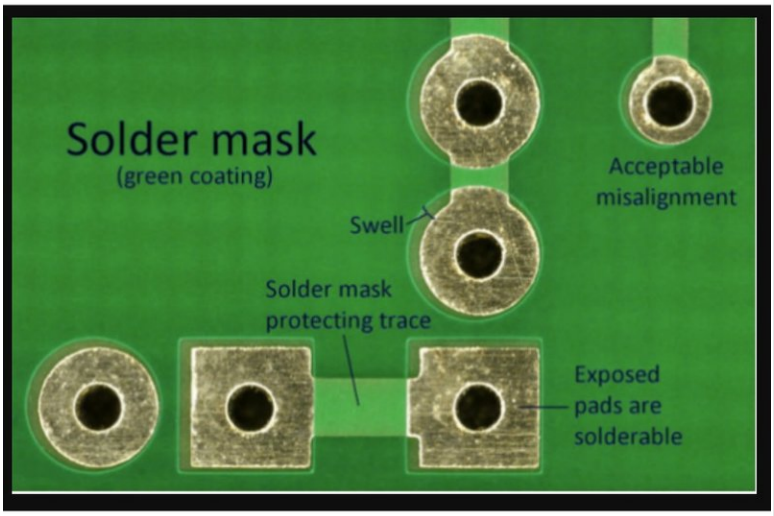


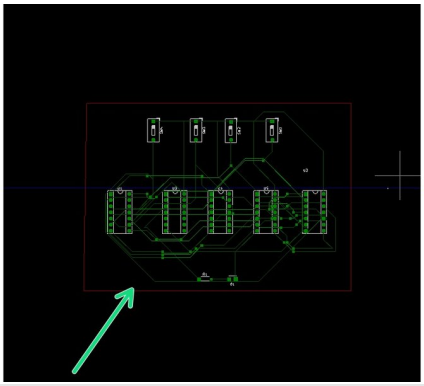

- A PCB interacts with a component through its *Footprint*.
  - A footprint maps the schematic symbol to the physical solder pads and through-holes that it will use to interact with the PCB
  - Active Components usually have footprints that consist of connections arranged in an array to interface with the circuit inside
  - Passive Elements typically have less complex footprints with 2-3 connections.
- Both types of components can use SMD or THT mounting. The footprints will be different for the SMD and THT versions of the same device.
- Footprints can also change based on the component's size. SMD components, in particular, have many different sizes for the same components.

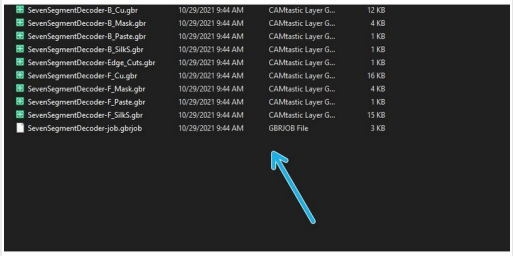
## Step 7 — Why are PCBs Green?!?!?




- During production, PCBs are covered with a layer of "Solder Mask," which is a green polymer coating that:
  - Repels Solder
    - Solder is made to stick to exposed metal, which improves it's electrical connection to a circuit. However, we only want solder on the solder pads. Therefore, Solder mask is made to repel solder to ensure that it doesn't stick and conduct anywhere that we don't want it to.
  - Removes Conductivity
    - The outer surface of a PCB is conductive without solder mask. If two unprotected PCBs touch, they could interfere with each other's operation.
  - Protects the Top and Bottom Surfaces
    - The outer surface of a PCB is just a sheet of copper. If this gets scratched or damaged, it can interfere with the signal quality.

## Step 8 — What data format are PCBs made with?







PCBs are exported and created using Gerber files (.gbr or .gb)

- These files contain many different layers, each representing one part of the PCB
  - Some examples are: top copper, bottom copper, drill holes, silkscreen, and solder mask
- These files are given to complex machinery that reads them and does the processing necessary to create the finished PCB

The process is a collection of drilling, milling, and electroplating steps, which you will learn about in a future guide