Essentials of PCB Design

01: PCB Basics

WPI A25 | Daniel Gorbunov, William Sobral

gr-pcb@wpi.edu

instructors

a25

daniel gorbunov

dgorbunov@wpi.edu

- Senior, CS major
- Past president of IEEE at WPI
- Internships: Apple (2x), MITRE
- Lots of personal projects w/ custom PCBs
- Originally took a similar class at MIT
- github.com/dgorbunov

instructors

a25

- Senior, RBE major
- IEEE Projects Chair
- Internships: Picogrid, 3D Systems
- Lots of personal projects w/ custom circuits
- Wants to work on hardware startups

william sobral

wsobral@wpi.edu

about this class

inspiration

Course inspired by pcb.mit.edu

Originally taught by Fischer and Adi

about this class

background

WPI’s project-based curriculum doesn’t teach you project design.

Most students don’t have personal projects — they aren’t sure where to start.

about this class

goals

Our goals are to

1. To augment WPI’s ECE/CS/RBE curriculum by teaching hardware design.
2. To instill careful, elegant, and thoughtful design methodology.
3. Give you skills and inspiration to embark on your own projects.

about this class

goals

We will not teach a scientific or rigorous method for designing PCBs.

We will teach a hands-on, design-driven method for creating PCBs.

about this class

pcb design as an art

PCB design is an art form.

Your skills improve with practice and experience.

Every PCB has unique constraints which require thoughtful design.

Every PCB has a unique personal touch and attention to detail.

about this class

schedule

A-Term

Tue, Sep. 23	Basics of PCBs **6-7PM; AK 233**
Thur, Sep. 25	Designing your Project **6-7:30PM; AK 233**
Tue, Sep. 30	Layout + Routing **6-7:30PM; AK 233**
Thur, Oct. 2	Working with KiCad **6-7:30PM; AK 233**

B-Term

Mon, Oct. 20 - Fri, Oct. 24	Office Hours **TBD**
Fri, Oct. 24	Boards Due **By 11:59PM**
Tue, Nov. 4 (tentative)	Assembly **TBD**

about this class

materials

All class content, including:

- Lecture slides
- Lecture recordings (Echo360)
- Datasheets/Resources
- PCB files

Can be found at github.com/ieee-wpi/pcb

We will be using KiCad (kicad.org/download/) for design.

about this class

prerequisites

- We don’t have time to teach you circuit design 😞
- You should be have a high-level understanding of:
  - Passive components (resistors, capacitors)
  - Microcontrollers
  - Digital signals
  - Basic C++ syntax

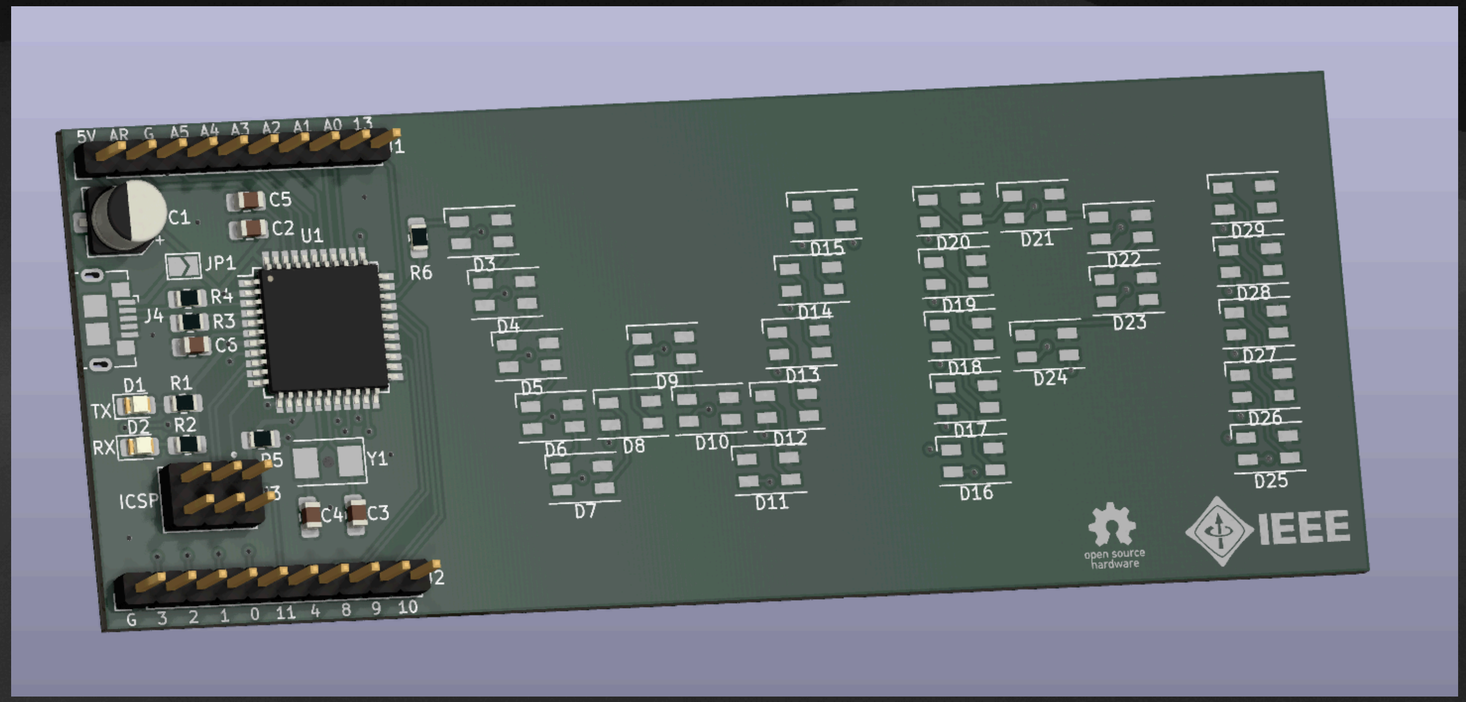
about this class

deliverable

You will be building a simple embedded system.

You'll layout the entire board, including a custom LED pattern, and peripherals of your choosing.

You'll program an LED pattern in C++ (we'll provide starter code).

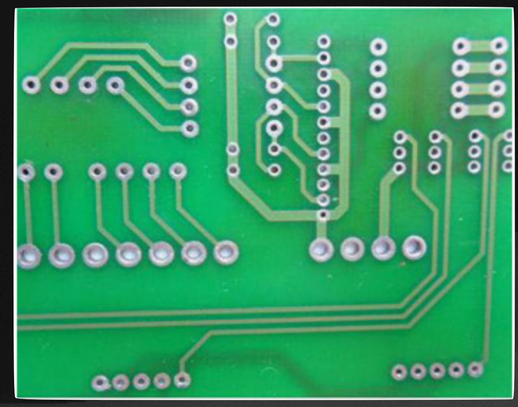


We will do our best to help you!

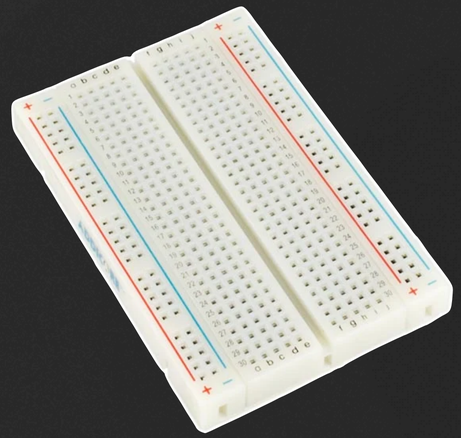
what is a PCB

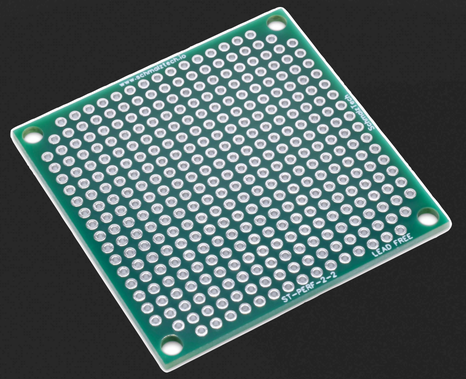
the basics

- **PCB = printed circuit board**
- **A way to connect components!**
- Functionally the same as…
  - solderless breadboards
  - solderable breadboards
  - perfboard



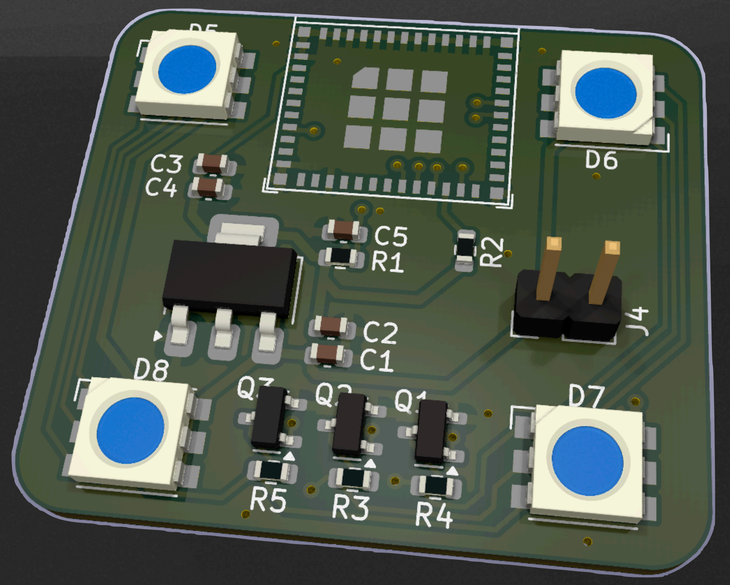
≈





why make a PCB?
the basics

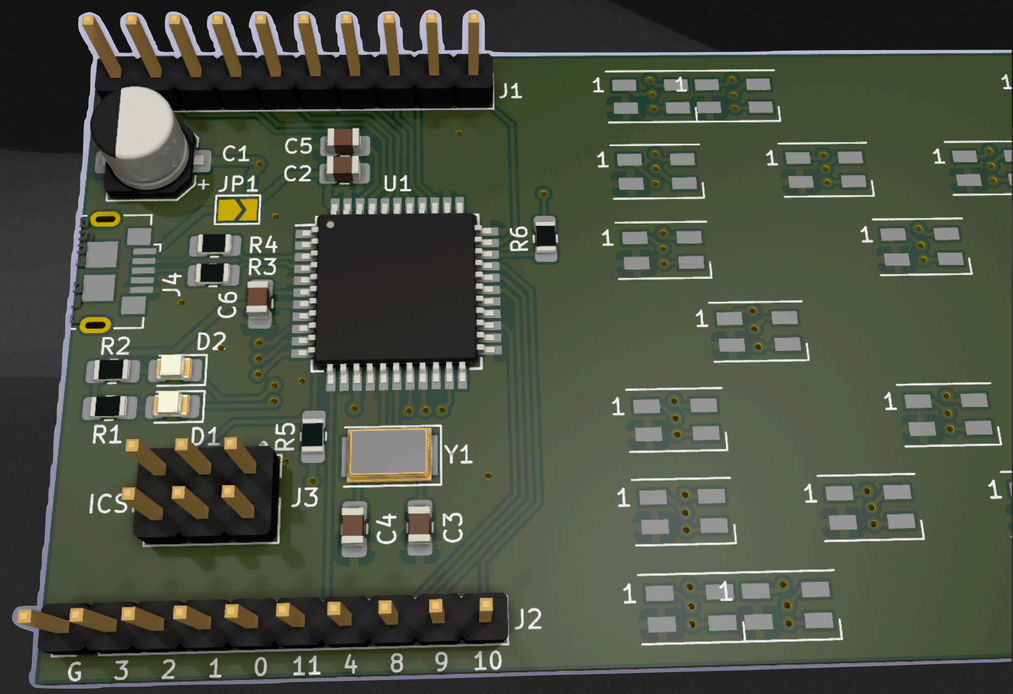
- **Transform an idea from a breadboard prototype into a refined product**
- Increased part availability
- Scalable for mass production
- Allows for complicated connections
- Denser than other platforms
- Entire process can be automated



what goes on a PCB

components

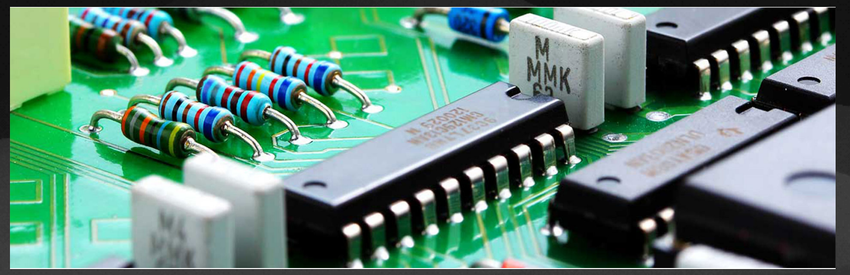
- **Passive parts**
  - Resistors, capacitors, inductors, crystals
- **Active parts** *(anything that requires power)*
  - Transistors, diodes, integrated circuits, voltage regulators
- **Mechanical parts**
  - Switches, connectors, mounting holes



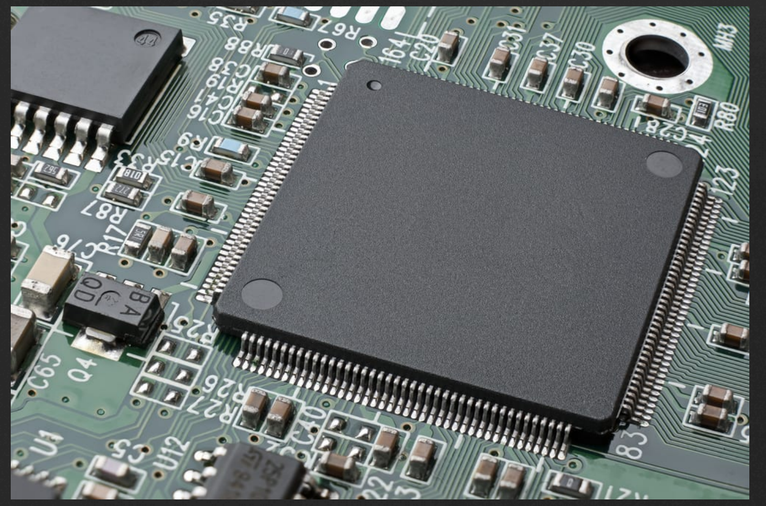
types of components

tht vs smt

- **THT = through hole technology**
  - Component legs put in holes drilled through PCB
  - Easy to solder by hand, larger components

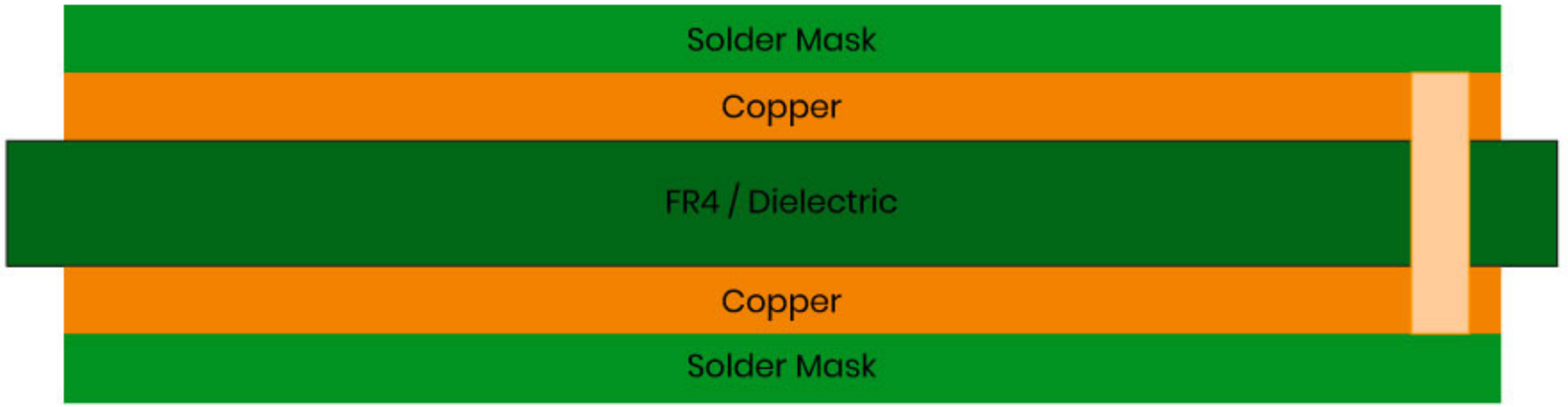


- **SMT = surface mount technology**
  - Component legs solder onto pads on PCB surface
  - Easy for machines to solder, smaller components



2 LAYER RIGID PCB STACK-UP

LAYERS	THICKNESS	LAYER DEFINITION
	0.80	Mask
L1	2.00	Layer 1
	59.00	
L2	2.00	Layer 2
	0.80	Mask



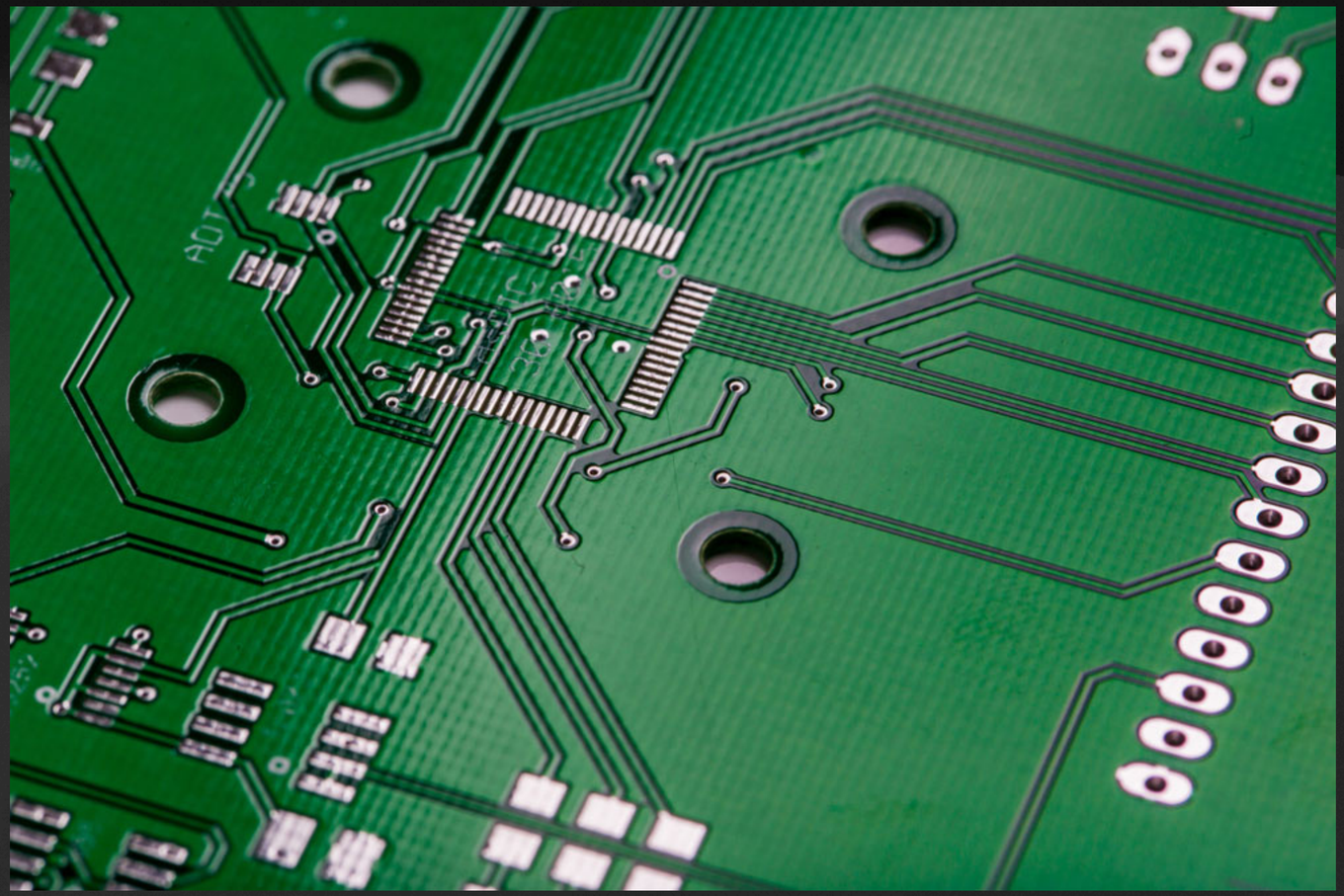
TOTAL THICKNESS WITH MASK: 63.800 mils

Notes: Outer layer copper thickness is 1/2 oz before plating. Finished outer layer CU thickness is approximately 1.4 to 2.2 mils after plating.

pads

the things that components go on

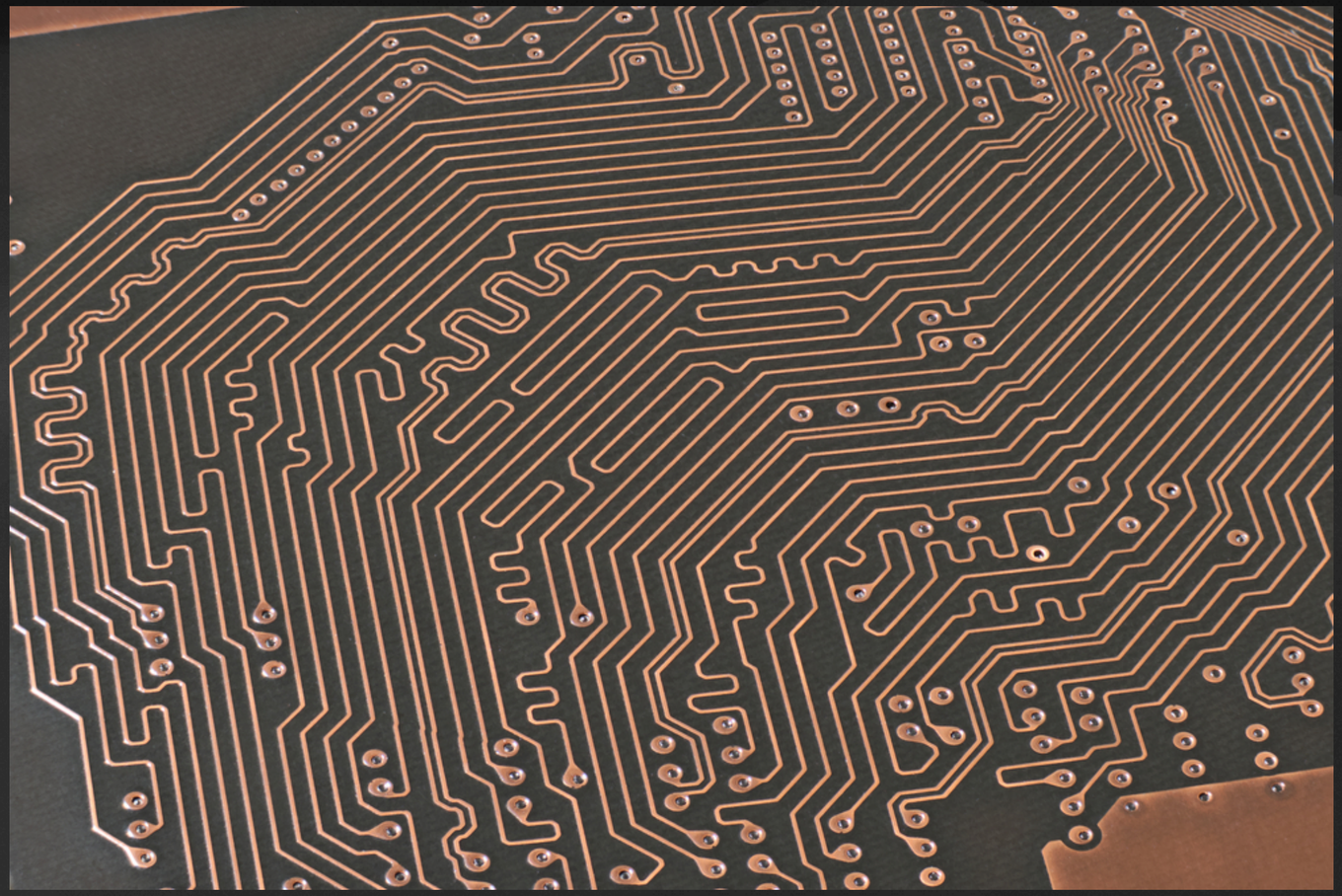
- The thing the component connects to
- The goal of a PCB is to connect traces to each other as defined by the schematic
- Come in all shapes and sizes depending on component



traces

they connect pads

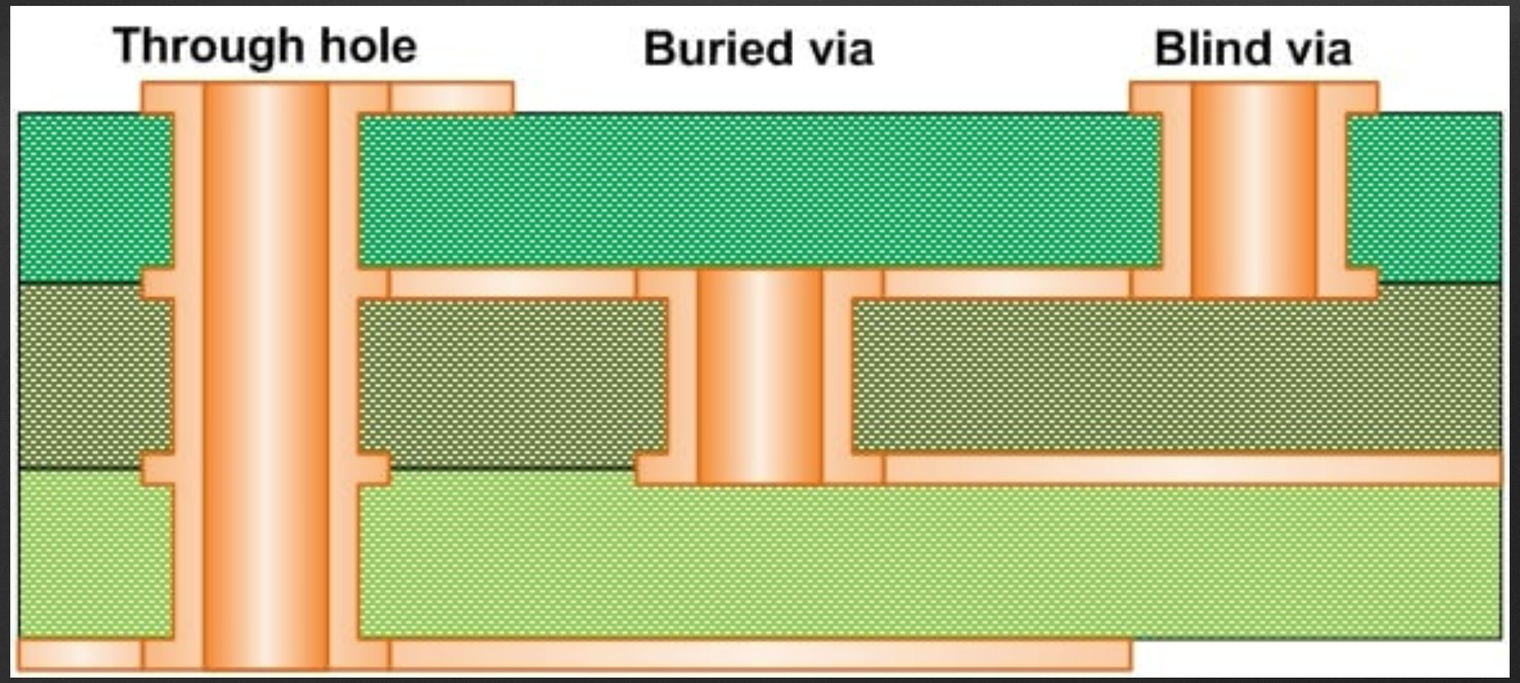
- This is what makes PCBs so useful!
- Usually straight lines with 45 or 135 degree bends
  - Why not 90 degrees?
- Wider trace width = **less** resistance
- Lower trace length = **less** resistance



vias

they connect layers

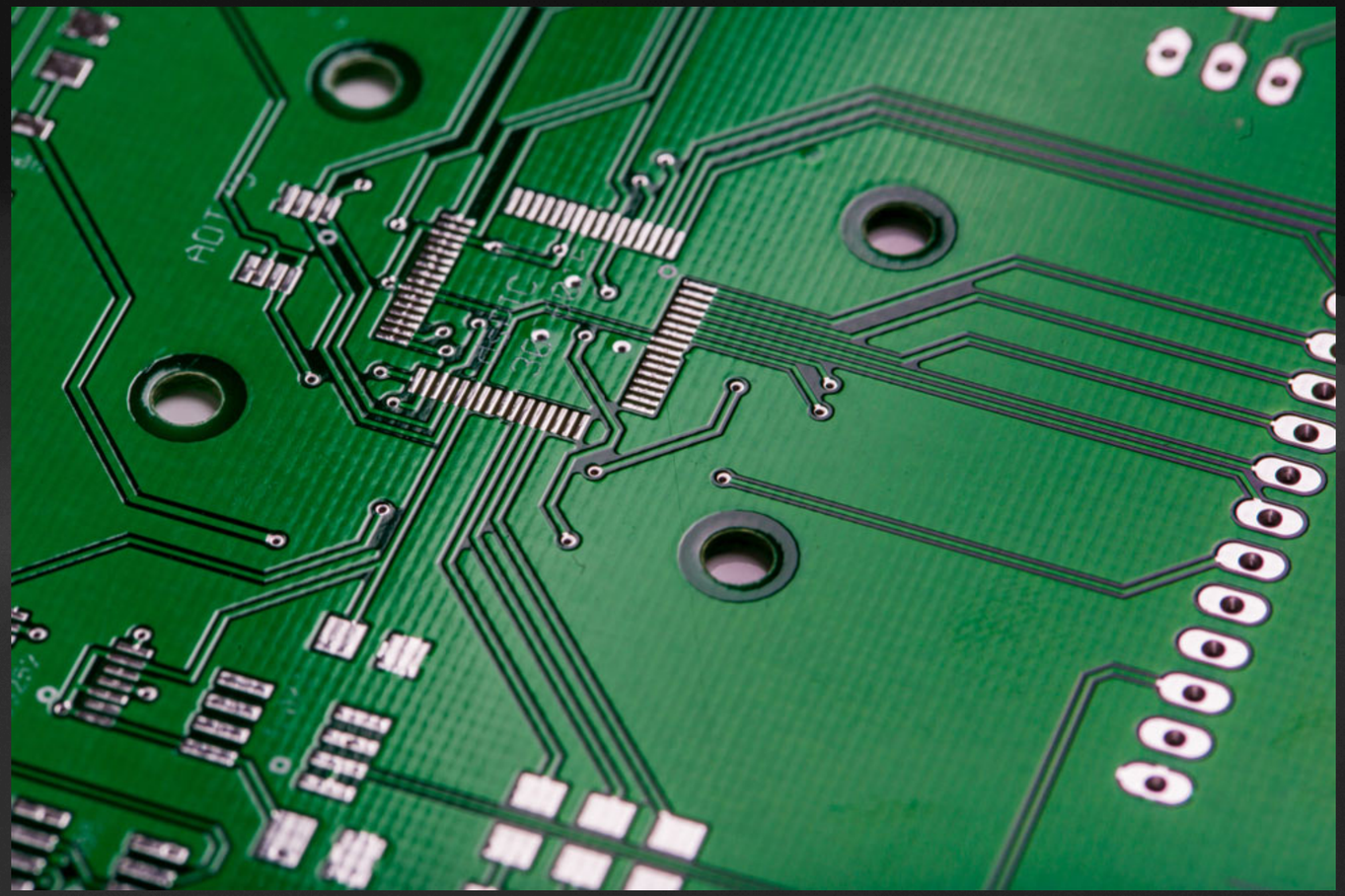
- Connect traces on different layers
- Through holes (normal) vias
- Blind vias ($)
- Buried vias ($$$)



planes

big areas of copper

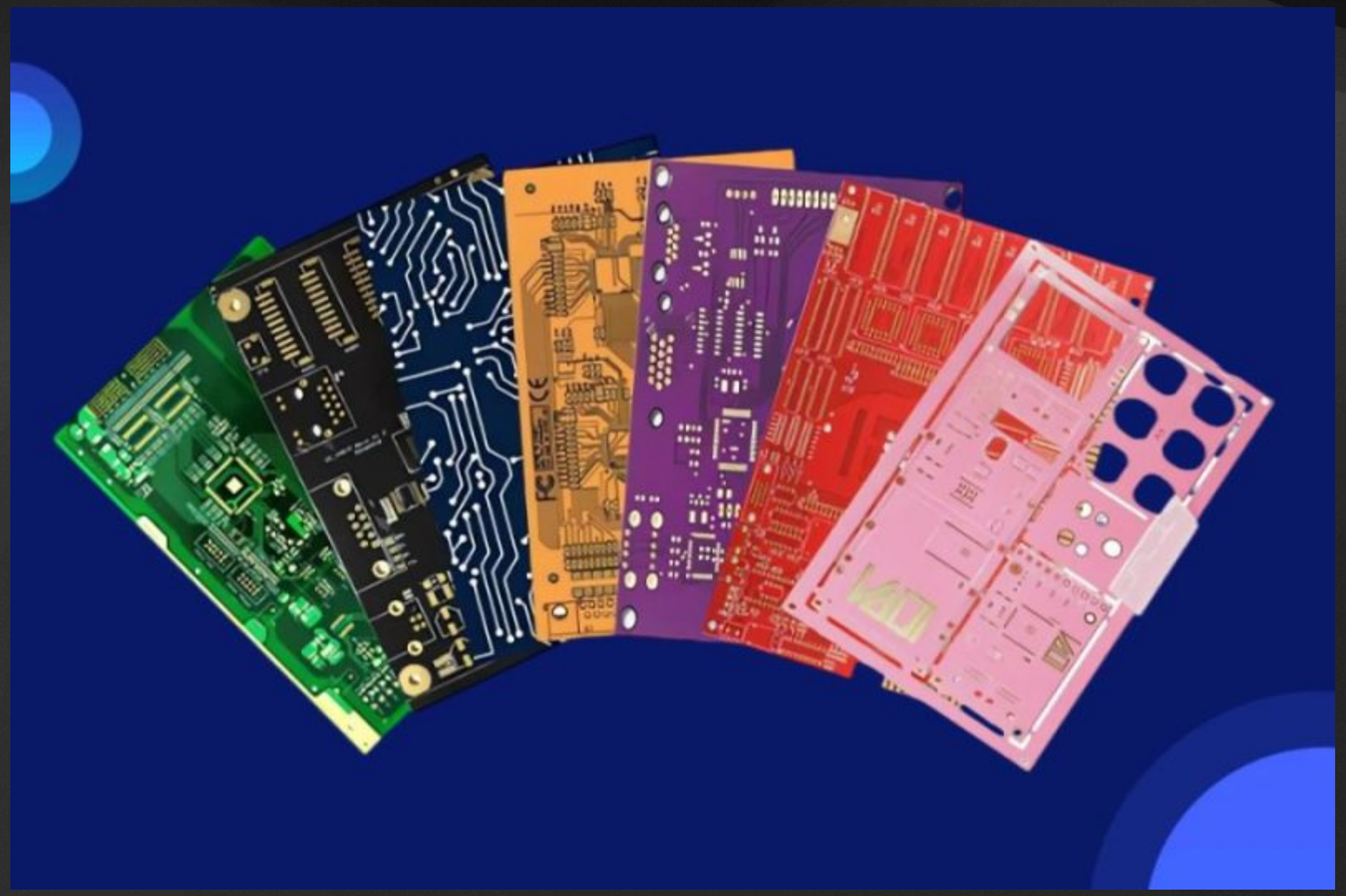
- Good practice to have a ground plane to reduce noise
- Nice to have a power plane
- Signal planes are nice to have! Usually not used in 2-layer pcbs
- Planes make routing much easier!



solder mask

the thing that keeps solder from going everywhere

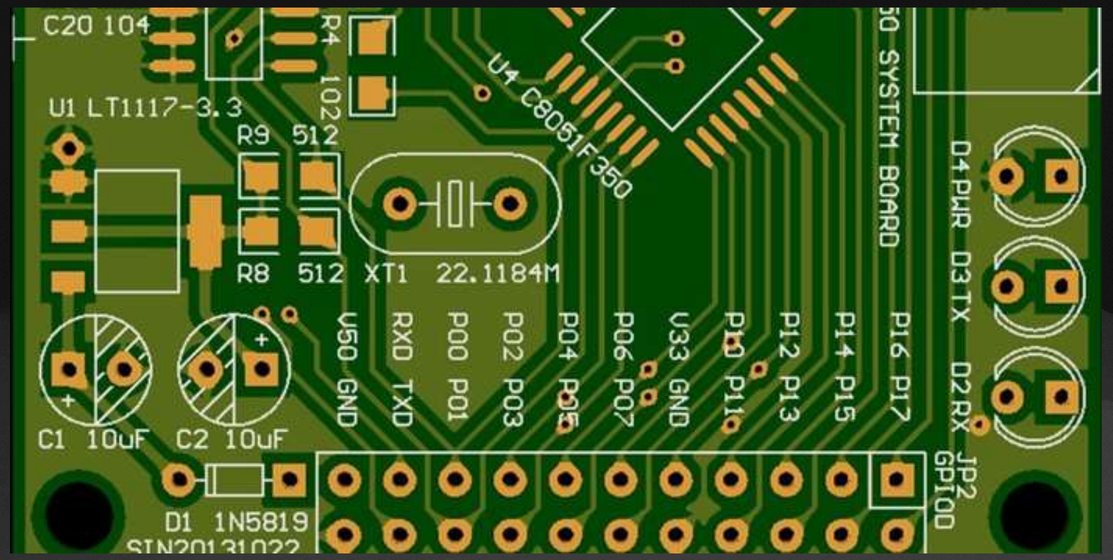
- Protects your board
- Prevents trace oxidation
- Prevents solder riding
- ‘Painted’ on
- COLORS

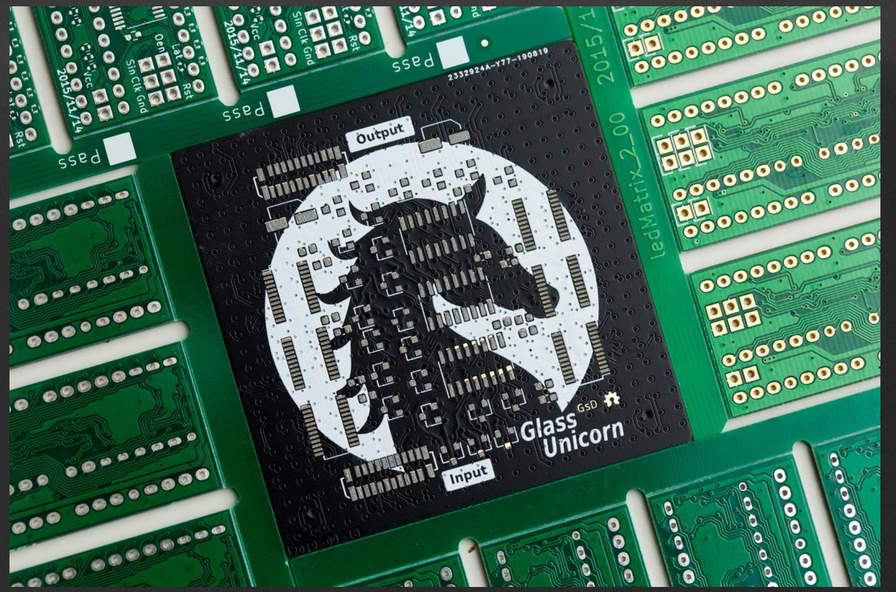


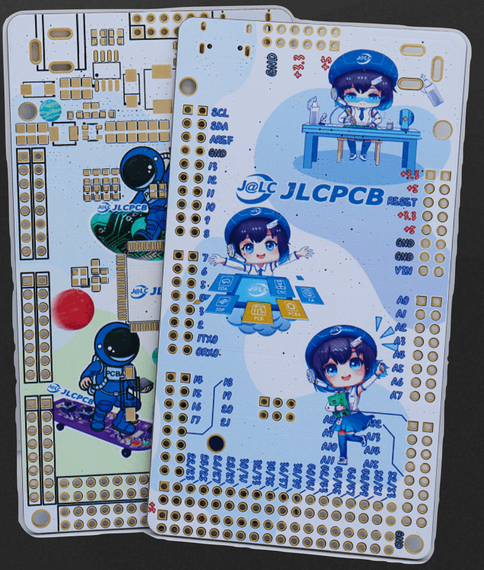
silkscreen

labels and art!

- Typically printed on with inkjet style printed
- You can do cool art!







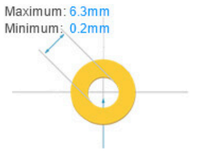
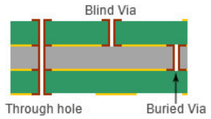
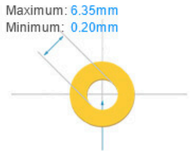
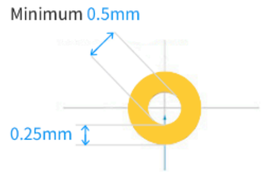
tolerances

the stuff the manufacturer cares about

- The boring stuff: layer count, max dimensions
- Via diameters
- Clearances
- Trace width and spacings
- Solder mask
- Silkscreen

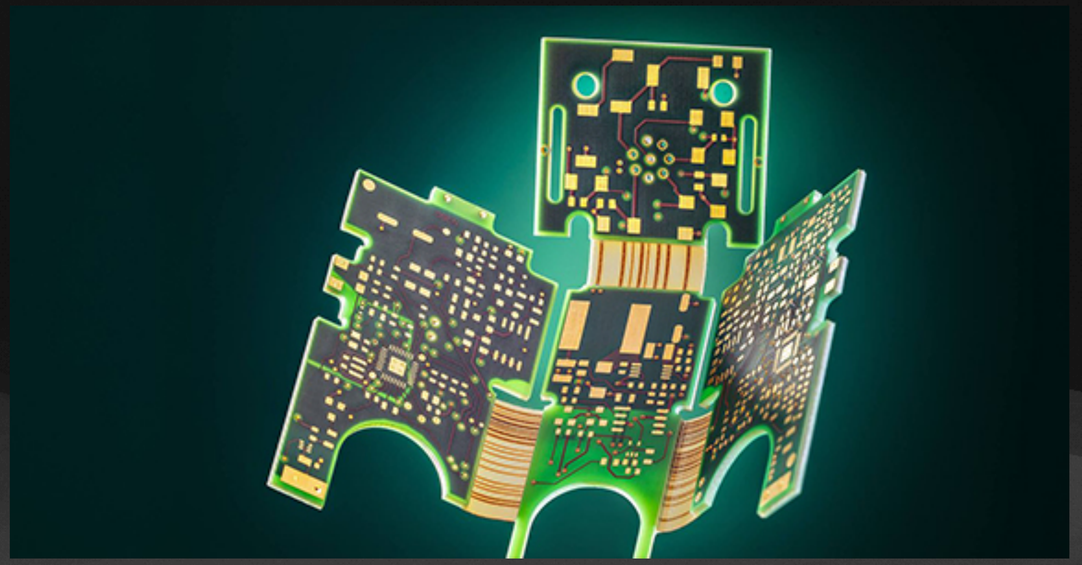
tolerances

from jlcpcb

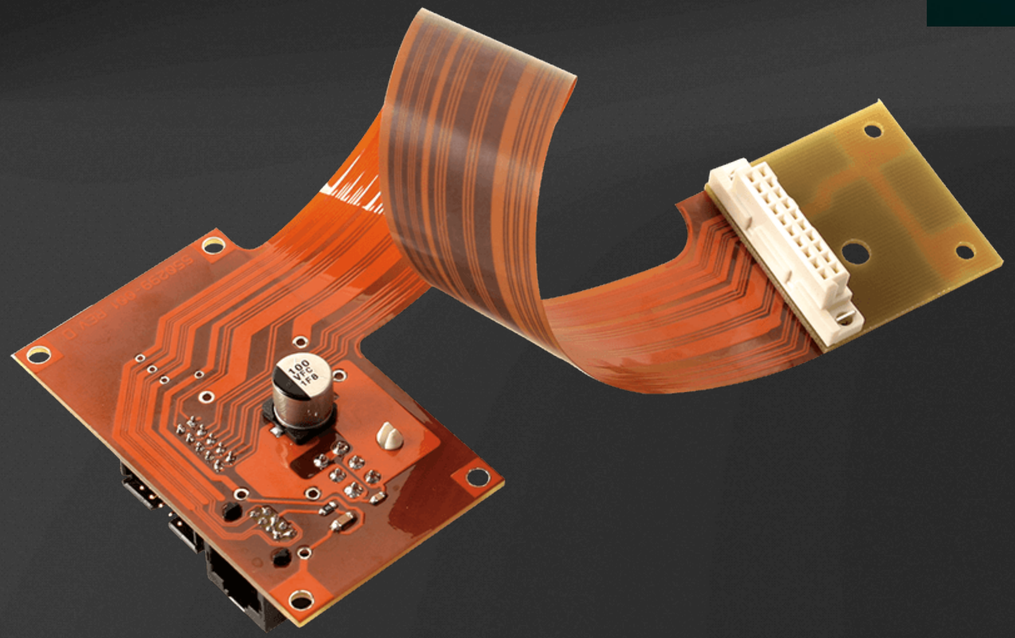
Features	Capability	Notes	Patterns
Drill Hole Size	0.15mm - 6.30mm	1 & 2 Layer PCB: 0.3 - 6.3mm Multi-Layer PCB: 0.15 - 6.3mm (0.15mm more costly)	
Drill Hole Size Tolerance	+0.13/-0.08mm	e.g. for the 0.6mm hole size, the finished hole size between 0.52mm to 0.73mm is acceptable.	
Blind/Buried Vias	Don't support	Currently we don't support Blind/Buried Vias, only make through holes.	
Min. Via hole size/diameter	0.15mm / 0.25mm	- 1 & 2 Layer PCB: 0.3mm(hole size) / 0.5mm(diameter) - Multi-Layer PCB: 0.15mm(Via hole size) / 0.25mm(Via diameter) ① Via diameter should be 0.1mm(0.15mm preferred) larger than Via hole size ② Preferred Min. Via hole size: 0.2mm	
PTH hole Size	0.20mm - 6.35mm	The annular ring size will be enlarged to 0.15mm in production.	
Pad Size	Minimum 1.0mm	The pad size will be enlarged by 0.5mm than the hole size. The minimum size of annular ring around plated through hole pads is 0.25mm. If the recommended sizes are not respected then the pad will not be produced properly.	

base materials

the stuff that everything is built on



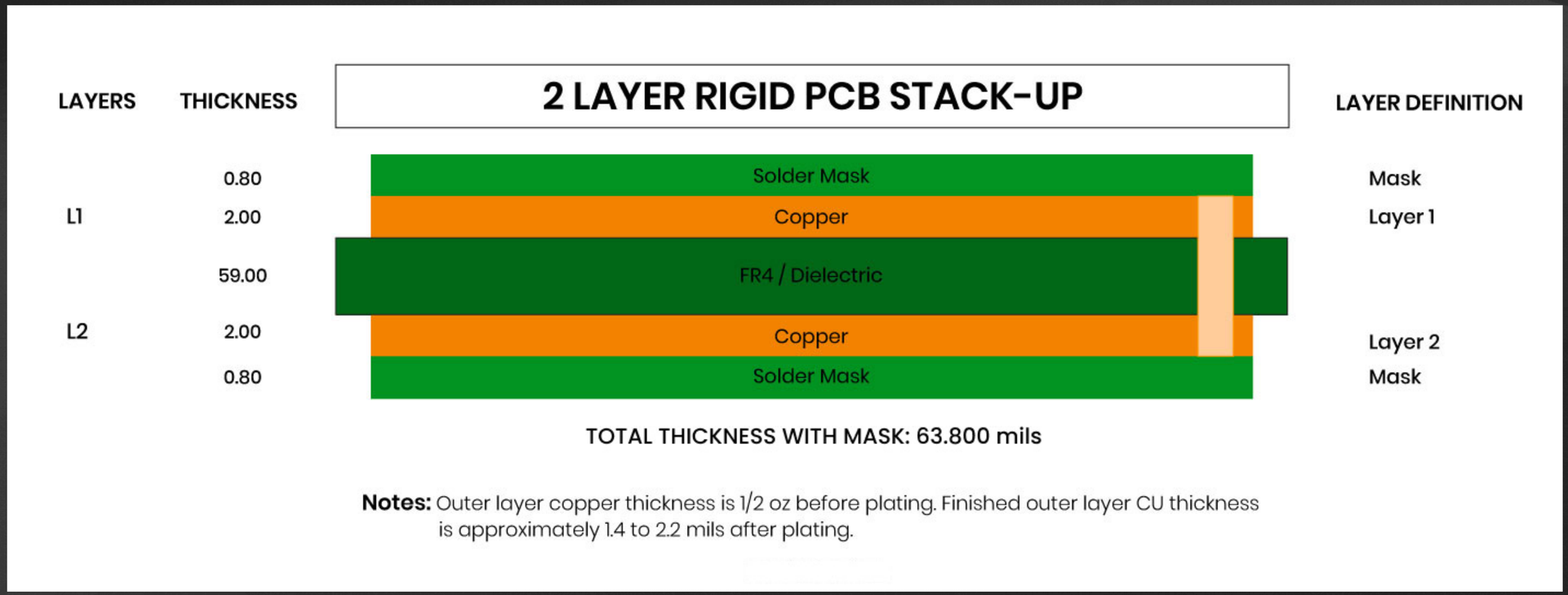
- FR-4 (we will be using this)
- Flex
- Rigid flex
- Others



types of PCBs

there are many

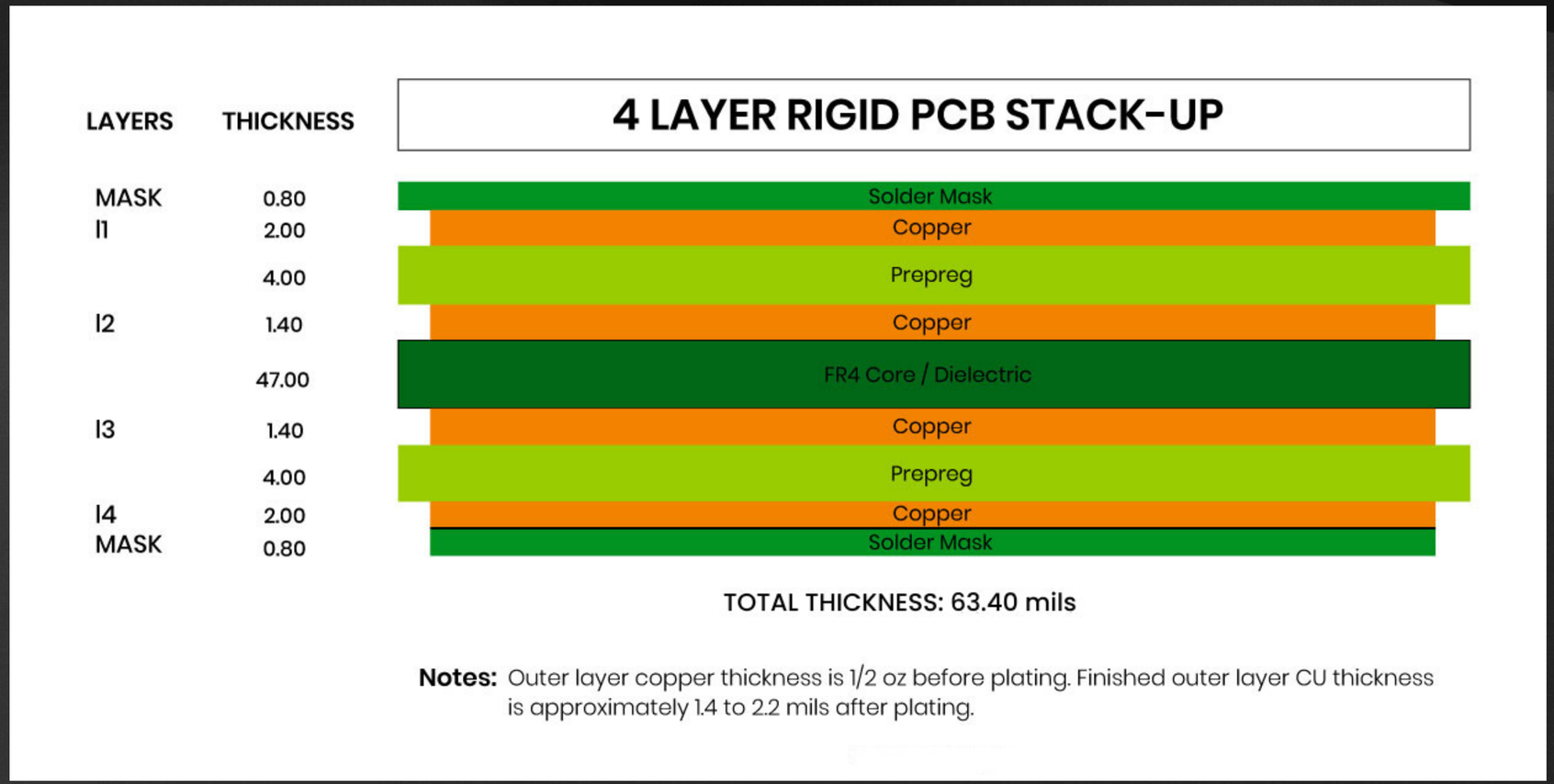
- 2 layer PCB



types of PCBs

there are many

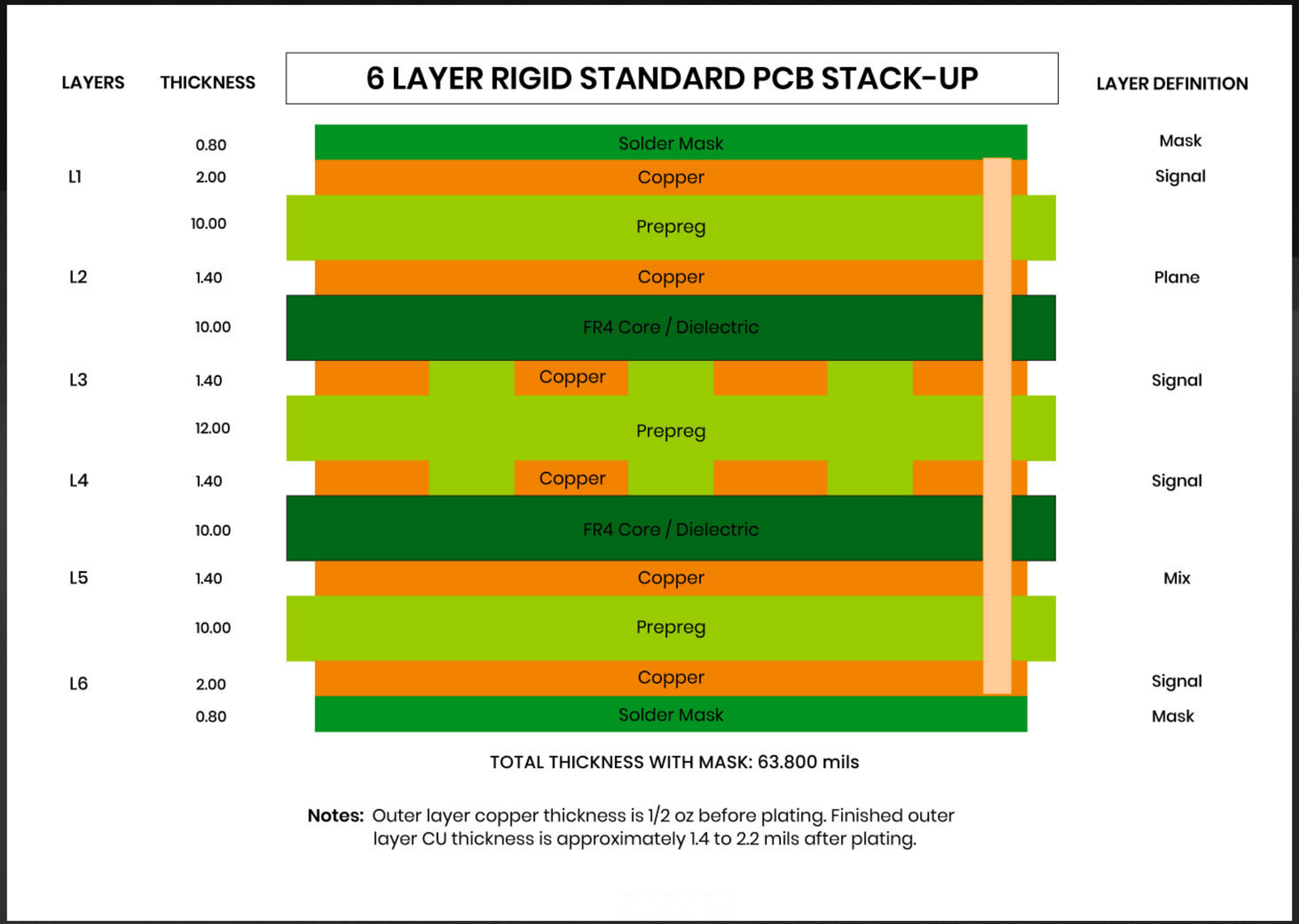
- 4 layer PCB



types of PCBs

there are many

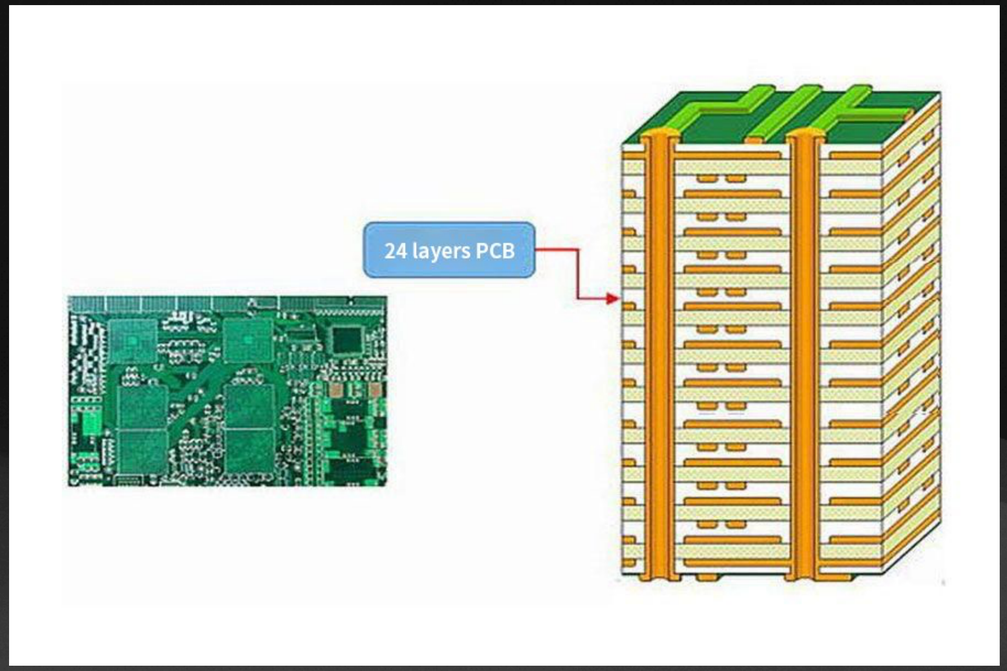
- 6 layer PCB

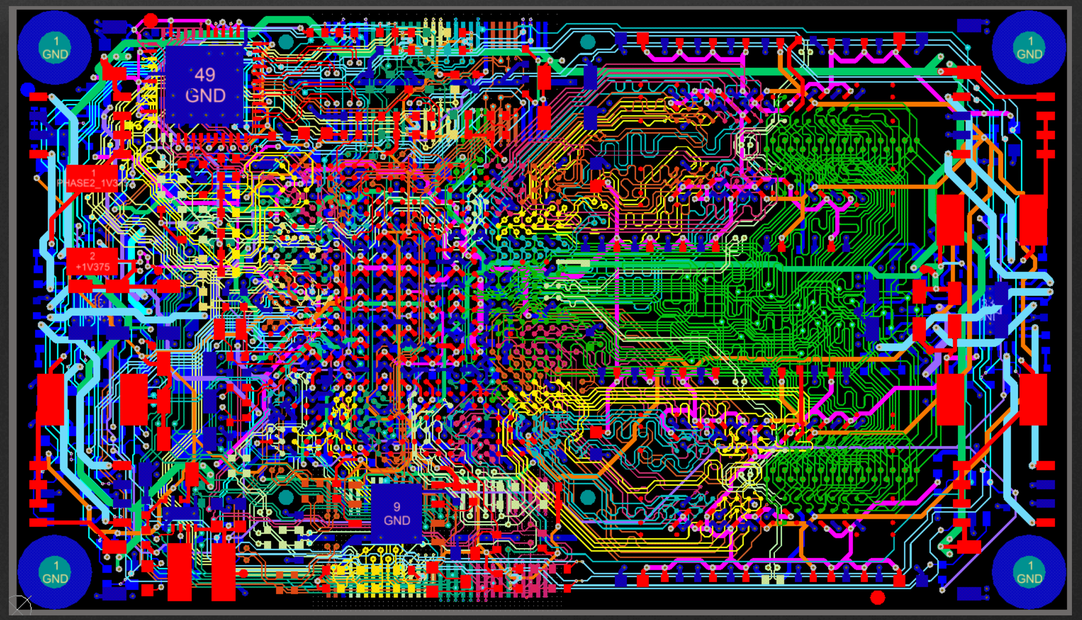


types of PCBs

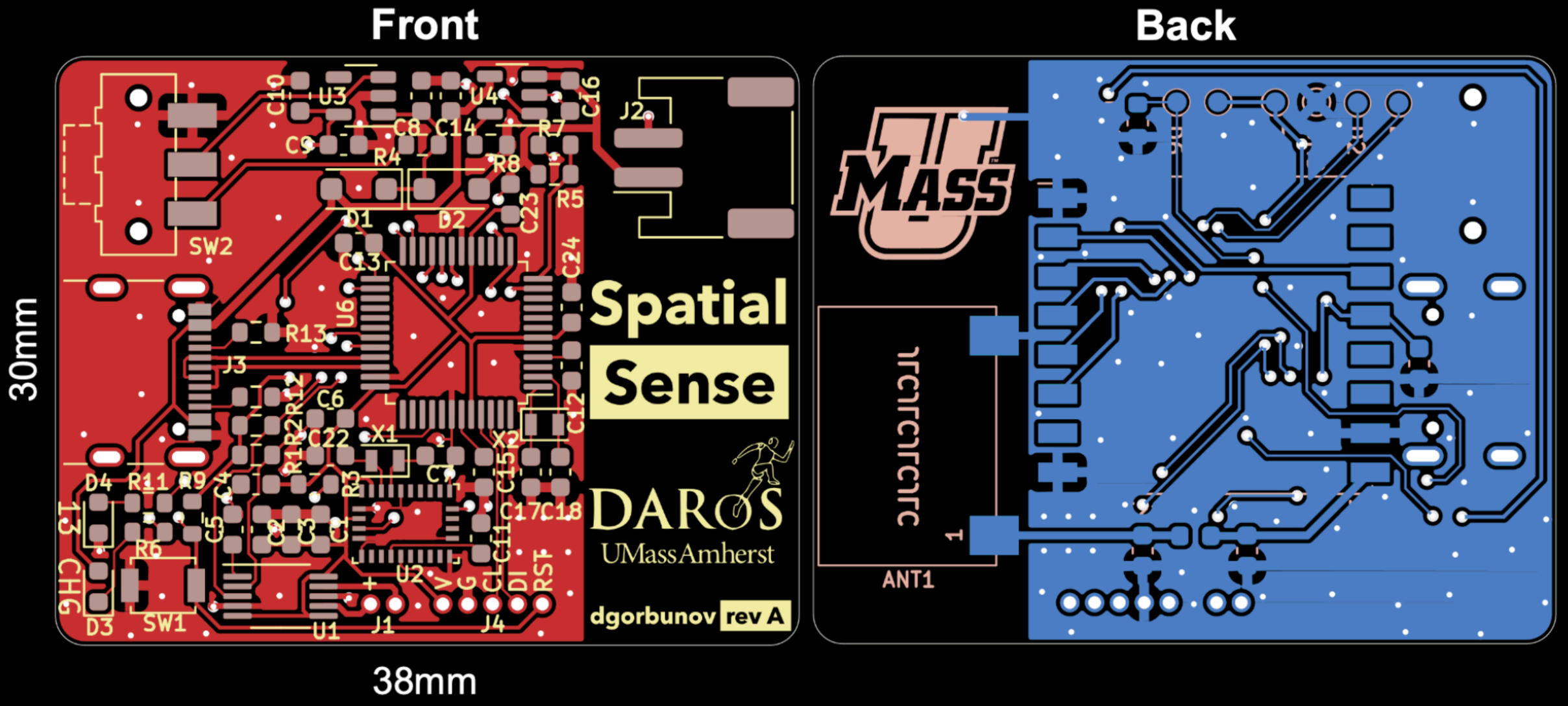
there are many

- 20+ layer PCBs
- Computing hardware





example board



who needs layers?

the problem of intersections

- THT wastes valuable space by drilling through all layers
- SMT components allow traces to be routed on reverse side
- Traces cannot intersect! More layers gives us more options.
- Higher **board complexity = More layers**

manufacturing

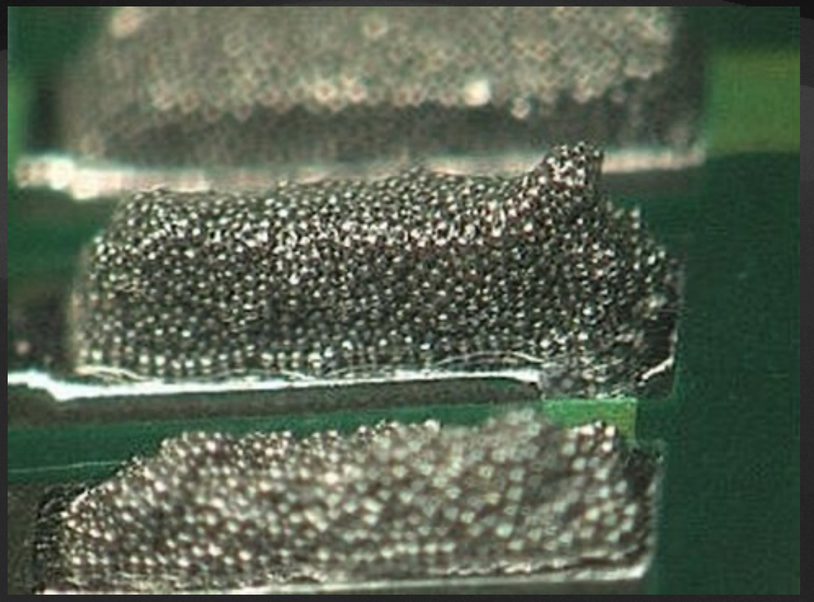
how the board is made

- China has brought consumer PCBs down to $2/5 boards

assembly

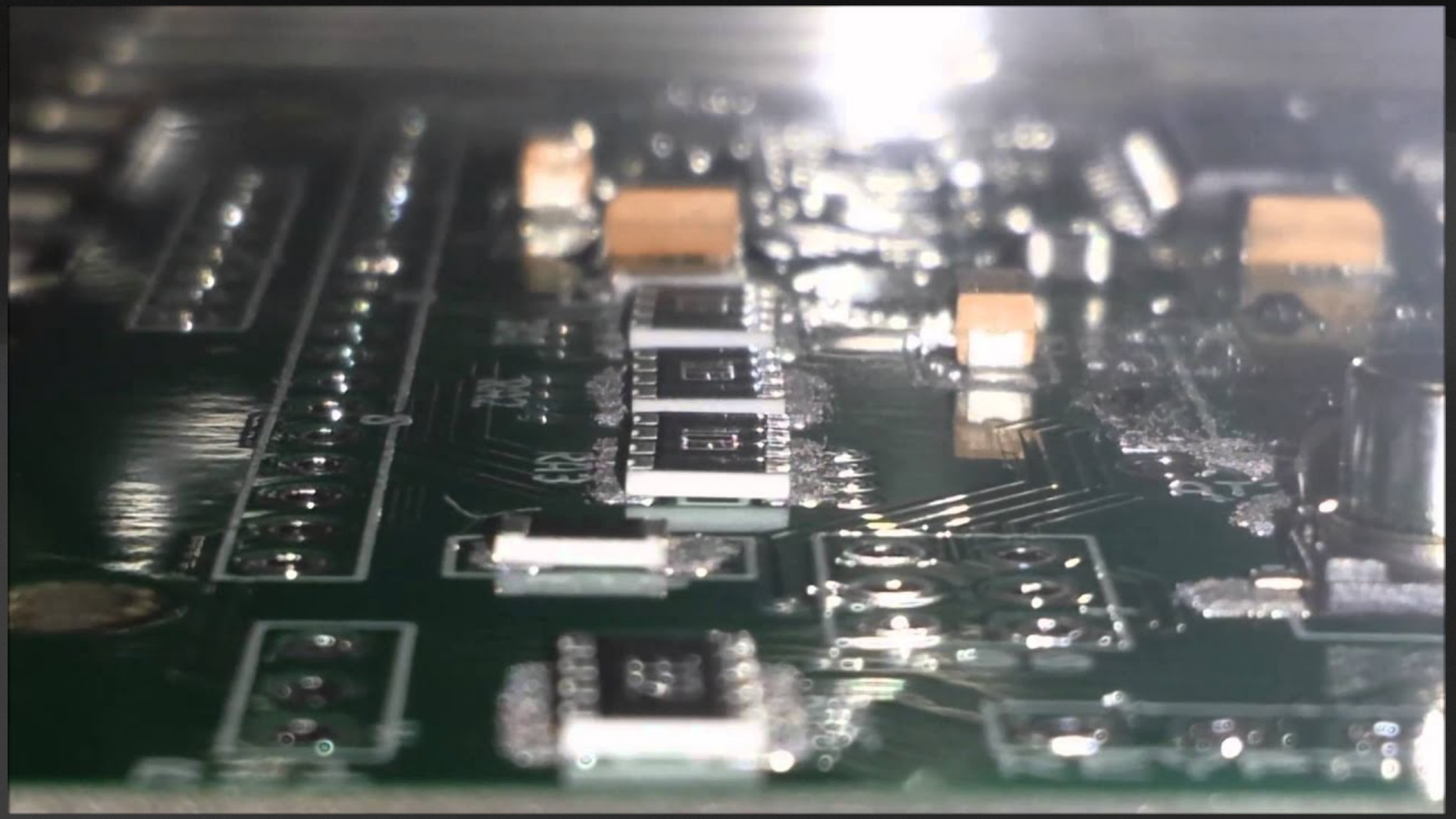
how to put components on

- **THT** components usually soldered **by hand** (even in factories!)
  - Good, old fashioned soldering iron
- To place **SMT** components, we use **solder paste**
  - Balls of solder powder and flux
  - Flux allows solder to flow onto pads
  - Components placed on solder paste
  - High temp. —> solder flows between pads and comp. legs



reflow soldering

you will be doing this!



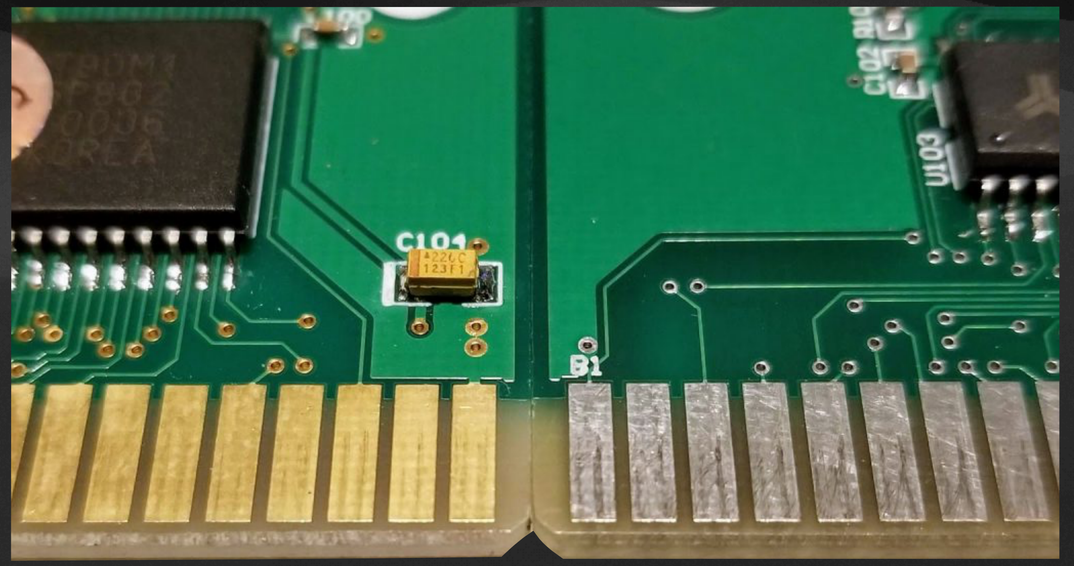
automated assembly

pick and place, reflow oven

finishes

the stuff that makes soldering easier

- **HASL** = hot air surface leveling
  - dip board in molten solder, level pads with hot air
  - pads can have uneven surfaces
- **ENIG** = electroless nickel with immersion gold ($)
  - apply thin layer of electroless nickel, then immerse board in gold solution
  - pads don’t have a thickness to them

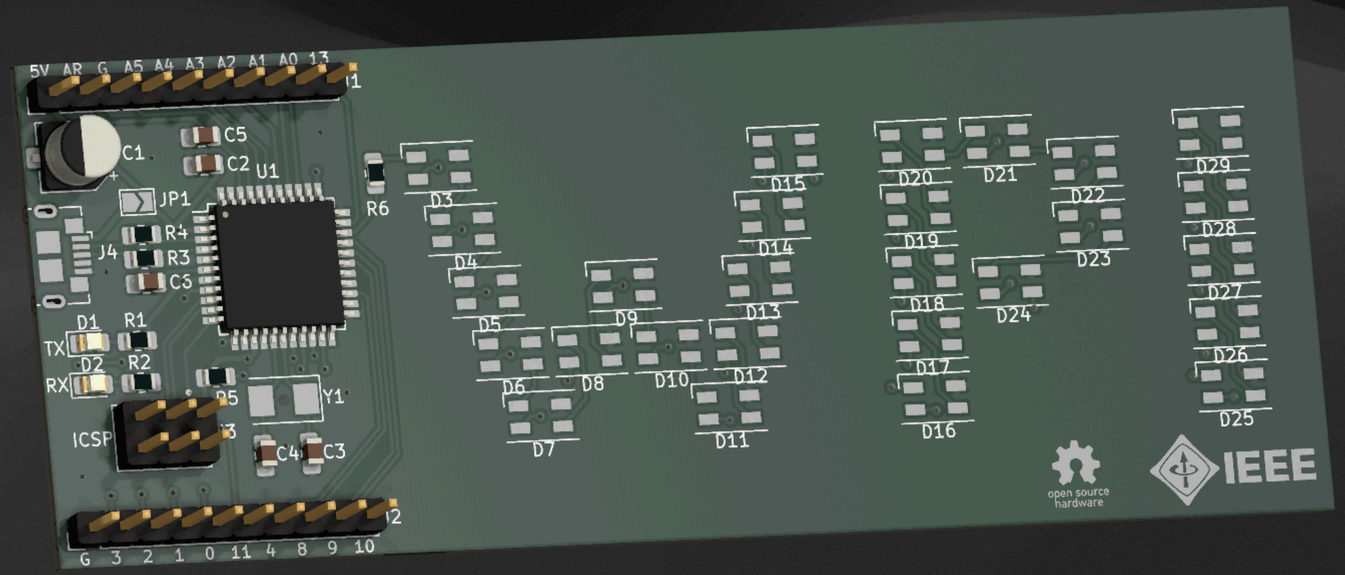


class PCB

what you will be making

Steps of PCB Design:

- Circuit Design
- Schematic Capture
- Layout
- Routing
- Order
- Assembly




IT'S ABOUT
TO GOAT


JLCPCB
JLCPCB.COM


WORK