PCB Design Workshop

01: Basics of PCBs, Schematic Capture

@WPI D24 | Daniel Gorbunov

instructors

d24

- WPI ’26 (CS w/ ECE minor)
- IEEE Interim President, Projects Chair
- Worked in defense previously
- Lots of personal projects w/ custom PCBs
- Originally took a similar class at MIT

daniel gorbunov

dgorbunov@wpi.edu

instructors
d24

- WPI ’27 (CS + ECE)
- IEEE Interim VP, Events Chair
- 2 years of interning at a photonics company
- Lots of personal projects w/ custom PCBs
- Designed something going to the moon!!

hubert liu

hhliu@wpi.edu

about this class

schedule

- 3-part class **(6:00 - 7:30 PM)**
  - 03/28, 04/4 **WB 229**
  - 04/7 **boards due by midnight**
  - 04/11 movie in AK (TBD)
  - 04/18 **AK 113**
- For sponsorship, **you must commit to attend all 3 sessions**
- Course adapted from pcb.mit.edu (thank you Fischer and Adi)

about this class
materials

All class content, including:

- Lecture slides
- Lecture recordings (Echo360)
- Datasheets
- Design resources
- PCB files

Will be posted on github.com/dgorbunov/ieee_pcb_workshop

Required software: KiCad (kicad.org/download/)

about this class

support

- We will be hosting office hours in **AK 106** (IEEE Lounge)
- Schedule:
  - 7PM - 10PM MTWF *(tentative)*
  - Weekends (TBD)

- Feel free to stop by with any questions or if you need help!
- Or send us an email!

about this class

goals

- Most ECE/CS curriculums struggle to teach **practical skills**
  - *A lot of WPI's curriculum is out of date with industry*
- Most students don’t have personal projects because they **lack these skills**
- Our goal is to augment the ECE curriculum by teaching you **hardware design**
  - We want you to be careful, elegant, thoughtful designers
  - We hope to inspire you with the skills and inspiration you need to embark on your own projects

Above all, we need you to be active learners! We’re here to help you!

about this class

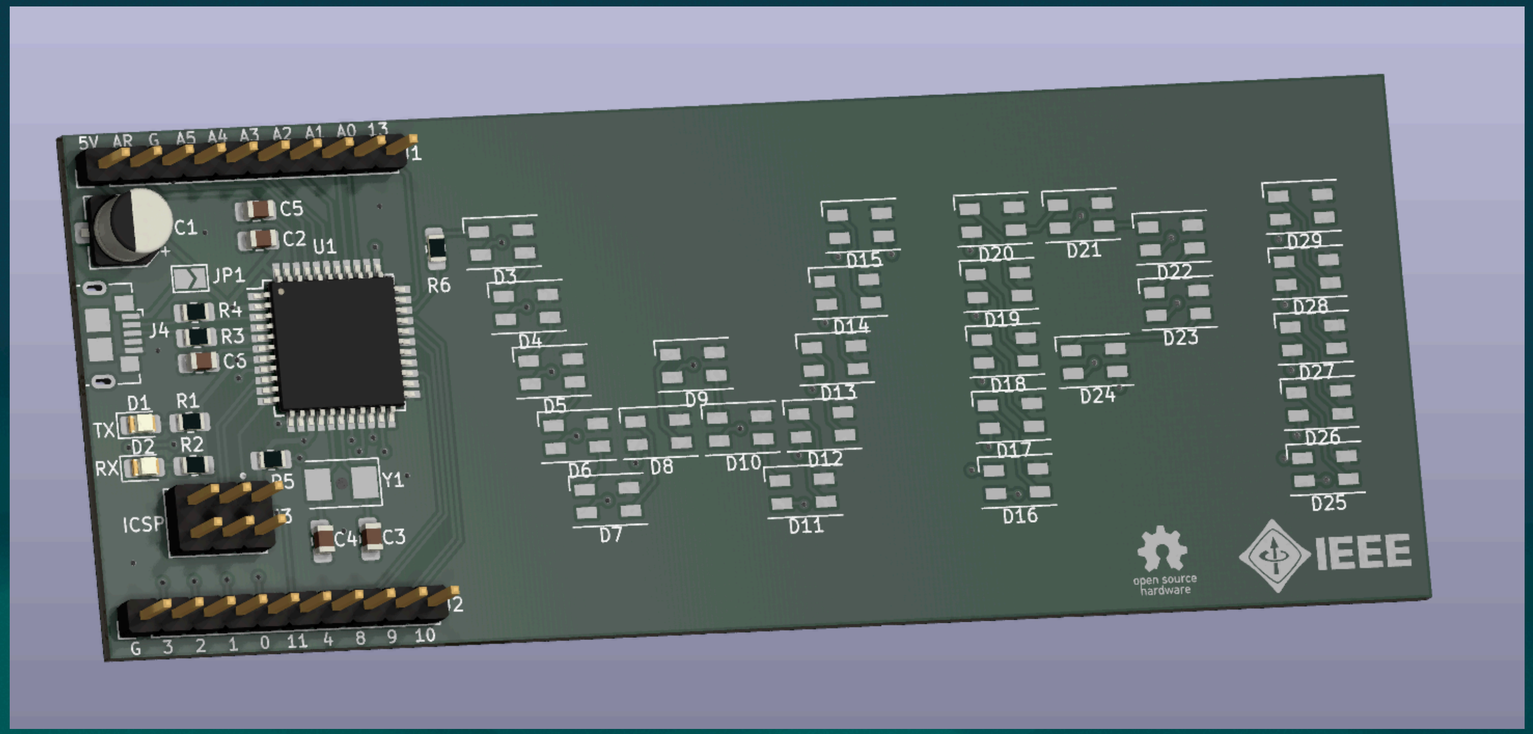
prerequisites

- We don’t have time to teach you circuit design :(
  - You should be familiar with these concepts:
    - Passive components (resistors, capacitors)
    - Microcontrollers
    - Digital signals
    - Programming in C/C++

about this class

deliverable

- You will be building a simple embedded system
- You will layout part of the left side of the board
- You will layout your own LED pattern on the right side of the board
- You will program your own LED pattern in C++ (but we will provide starter code)



We will do our best to help you!

project examples
to get you excited!

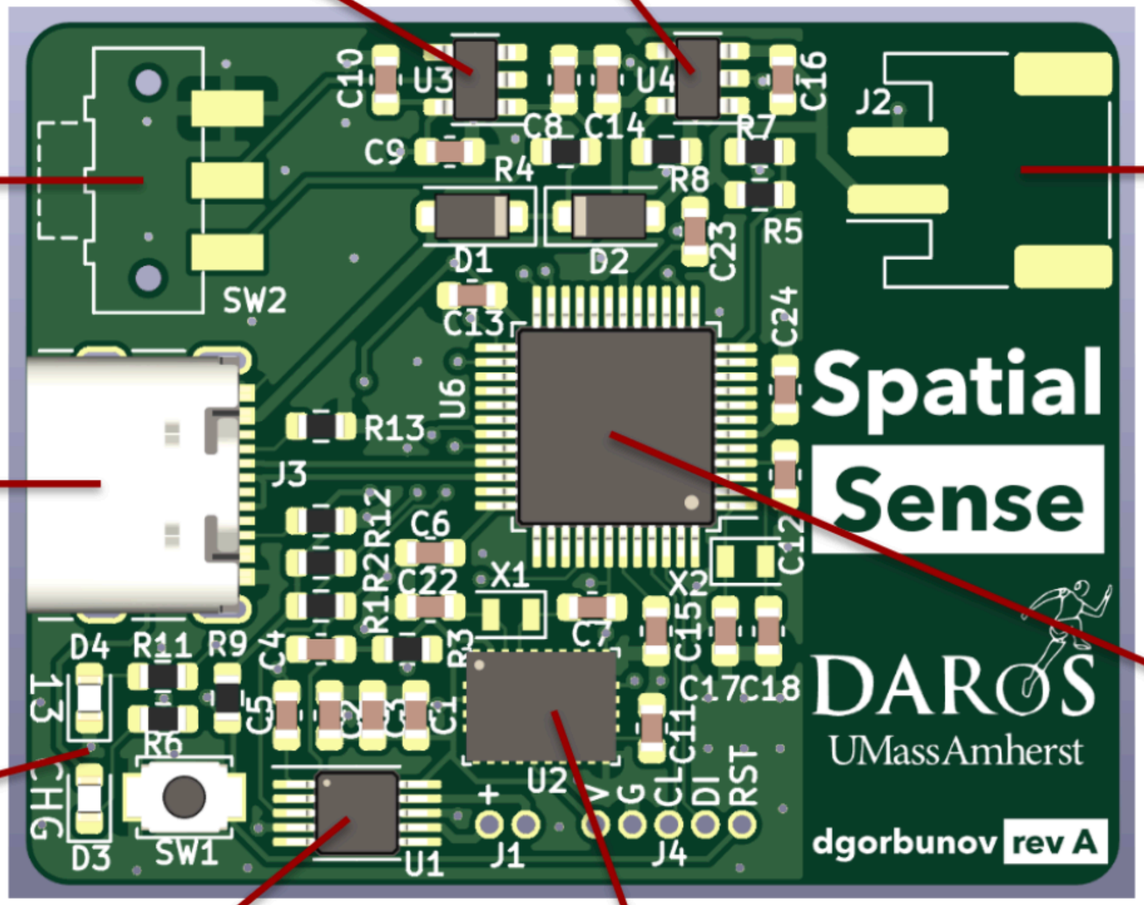
- **SpatialSense:** Wireless haptic feedback system for visually impaired people
- What are the major components that should go on the board?

project examples
to get you excited!

- **SpatialSense:** Wireless haptic feedback system for visually impaired people
- Main components:
  - ARM Microcontroller (**Atmel**)
  - Haptic Motor Driver (**Texas Instruments**)
  - 6 axis IMU (**Bosch**)
  - Voltage regulator (power supply)
  - LiPo battery charging IC
  - Packet radio (915MHz)

Spatial Sense

UMassAmherst | Manning College of Information & Computer Sciences



Power Supply

Batt. Charging

Power Switch

LiPo Battery Connector

USB-C Receptacle

Atmel ATSAMD21
ARM Microcontroller

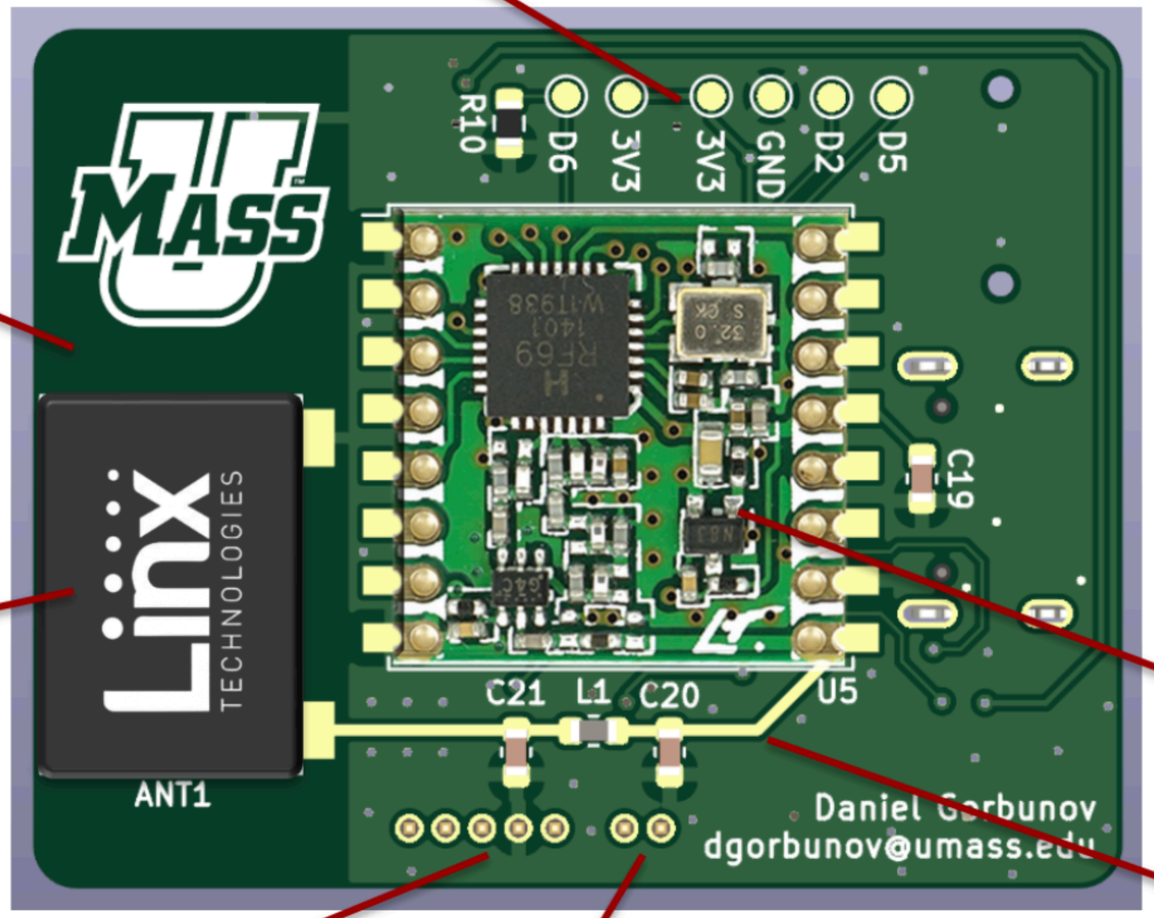
Indicator LEDs

TI DRV2605L
Haptic Motor Driver

Bosch BNO085
9DoF IMU+MCU

Spatial Sense

UMassAmherst | Manning College of Information & Computer Sciences



Breakouts (extra features)

RF Plane (no GND)

Linx Antenna
915MHz, Tuned 50Ω Impedance

RFM69HCW
915MHz Packet Radio Transceiver

RF Microstrip + Impedance Matching Network

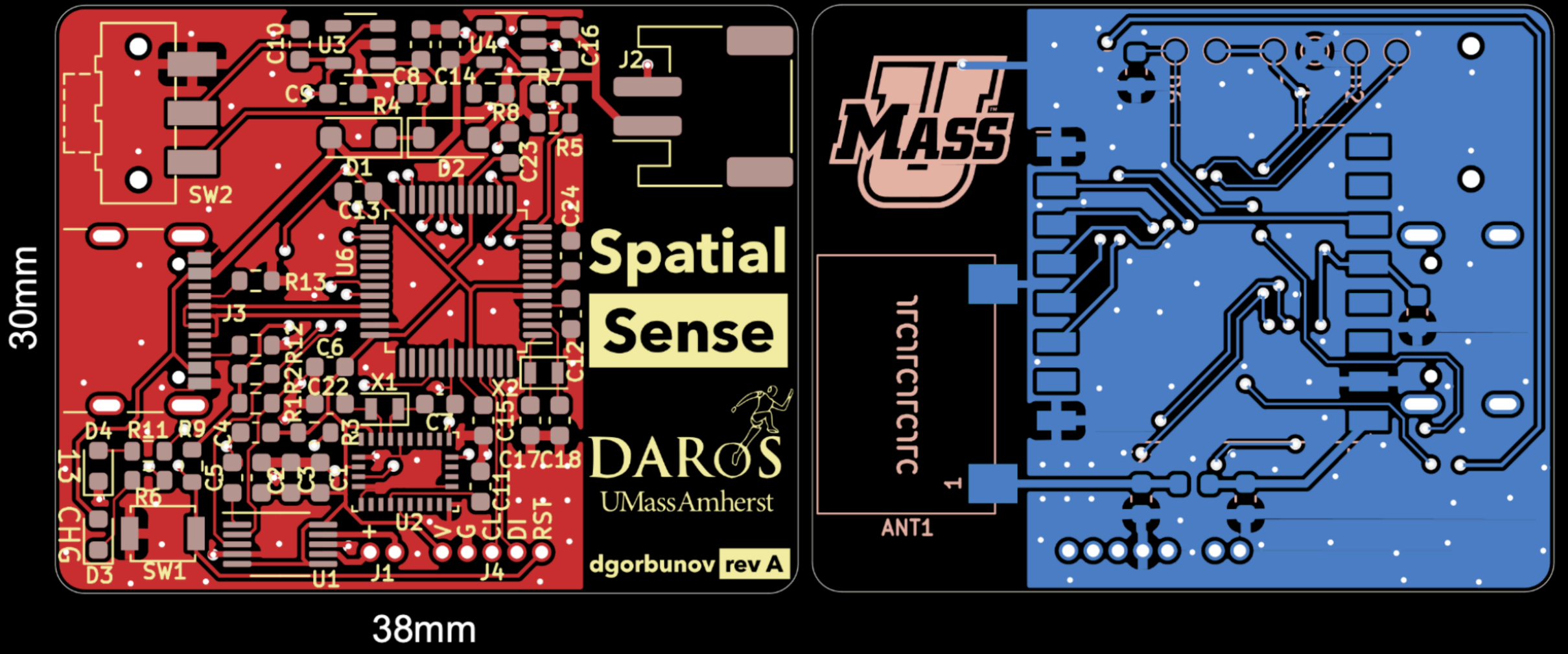
SWD Programming Interface

Vibration Motor Pins

PCB Layout

Front

Back



seems complicated?

by the end of this course…

- We want you to understand how to break down…
  - **Ideas —> schematics —> PCBs**

- Engineering and PCB design are an art…
- This methodology like this will give you superpowers…
- An idea —> functional, tangible work of art

project examples
to get you excited!

- Reactive LED Matrix
  - Demo!

what is a PCB

good question

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

what goes on a PCB

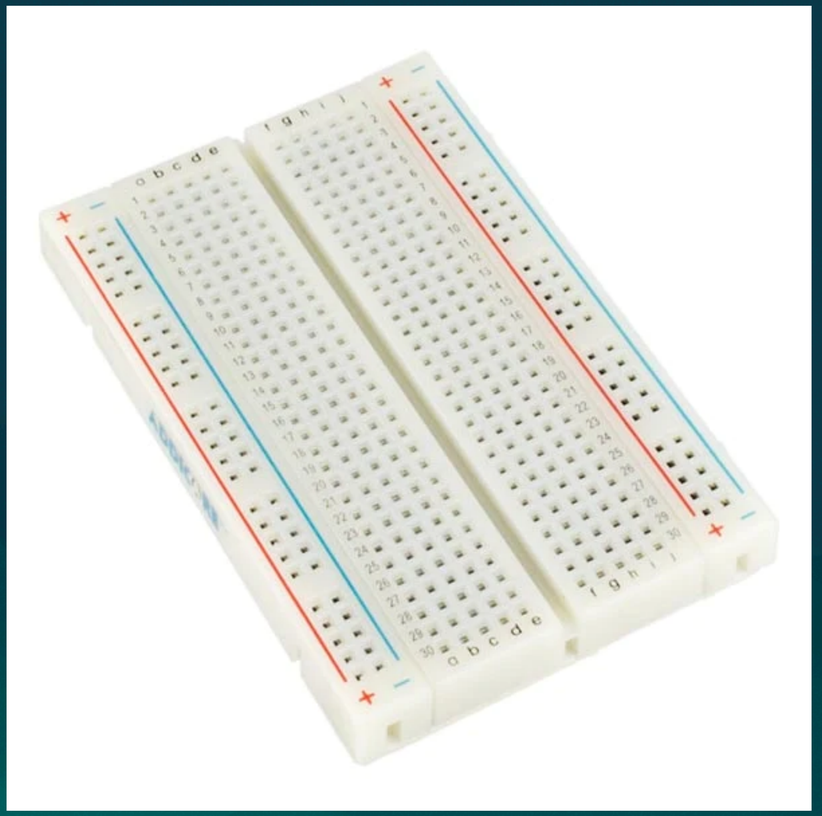
components

- **Passive components**
  - Resistors, capacitors, inductors, crystals
- **Active components** *(anything with silicon)*
  - Transistors, diodes, integrated circuits (ICs)
- **Mechanical components**
  - Switches, connectors

types of components

tht vs smt

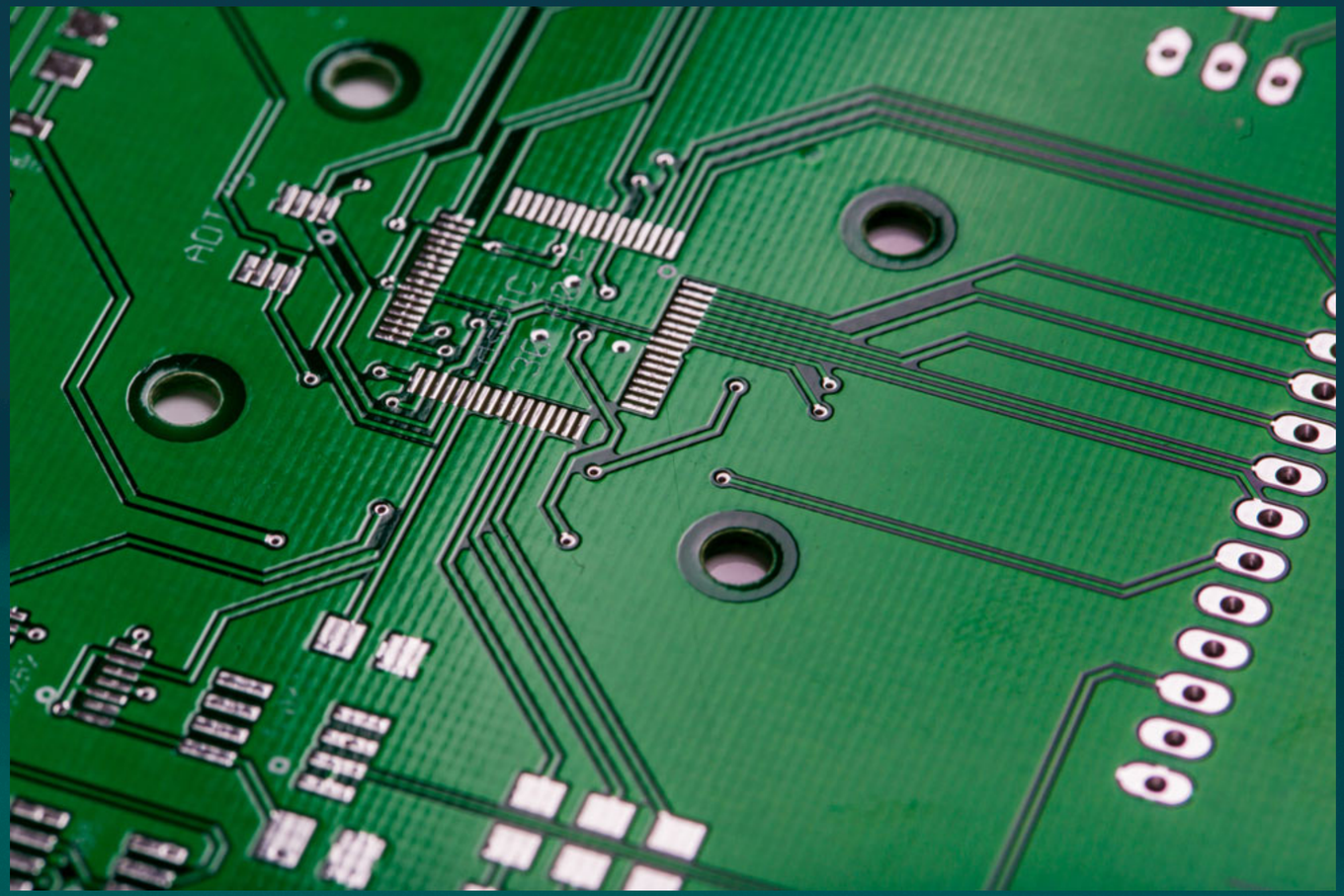
- **THT = through hole technology**
  - Component legs in holes drilled through entire PCB
  - Easy to solder by hand, larger components
- **SMT = surface mount technology** *(anything with silicon)*
  - Component legs solder onto pads on PCB surface
  - Easier to solder by a machine, smaller components



pads

the things that components go on

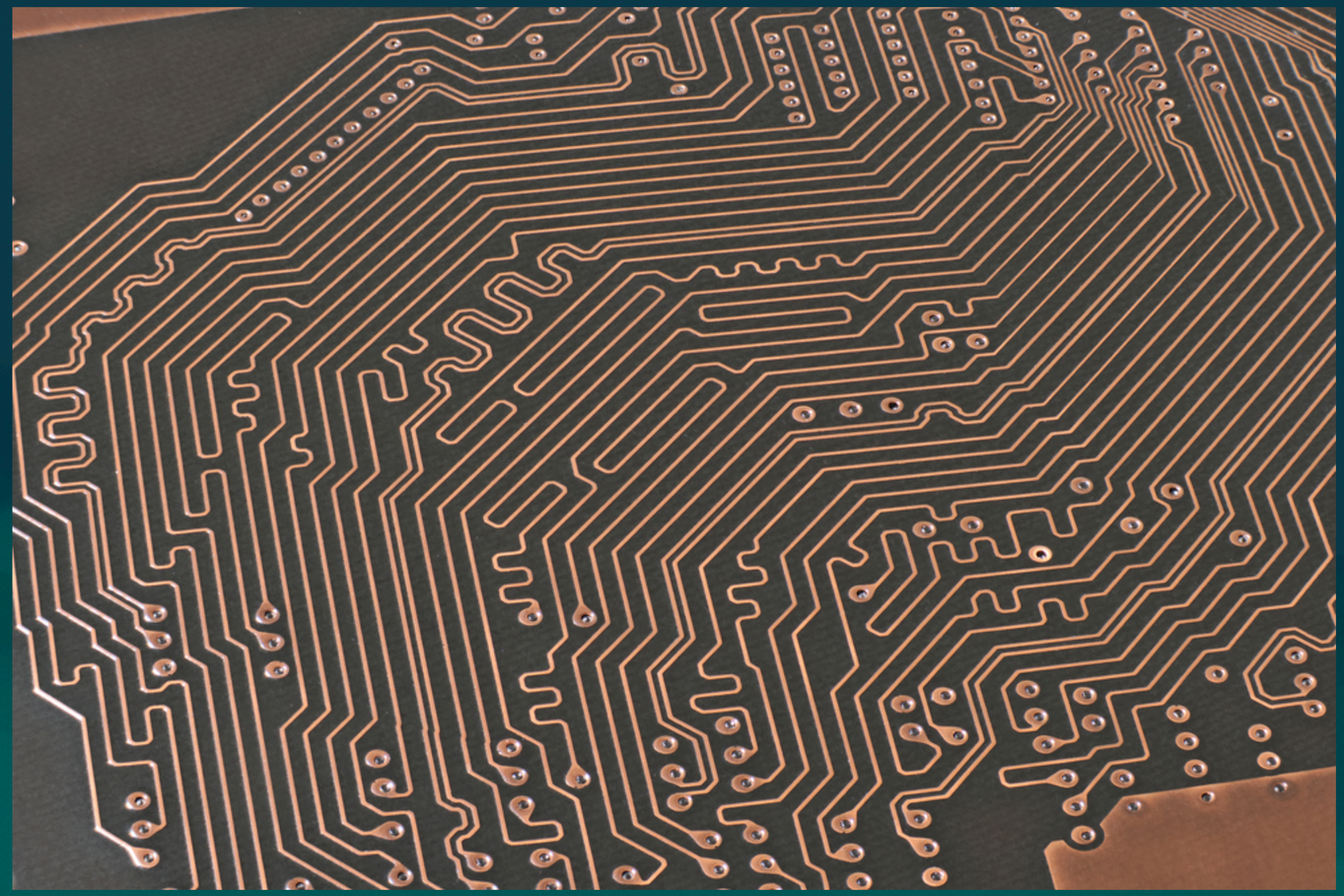
- The thing the component connects to
- The goal of a PCB is to connect traces to each other as defined by the schematic
- Come in all shapes and sizes depending on component



traces

the things that connect pads

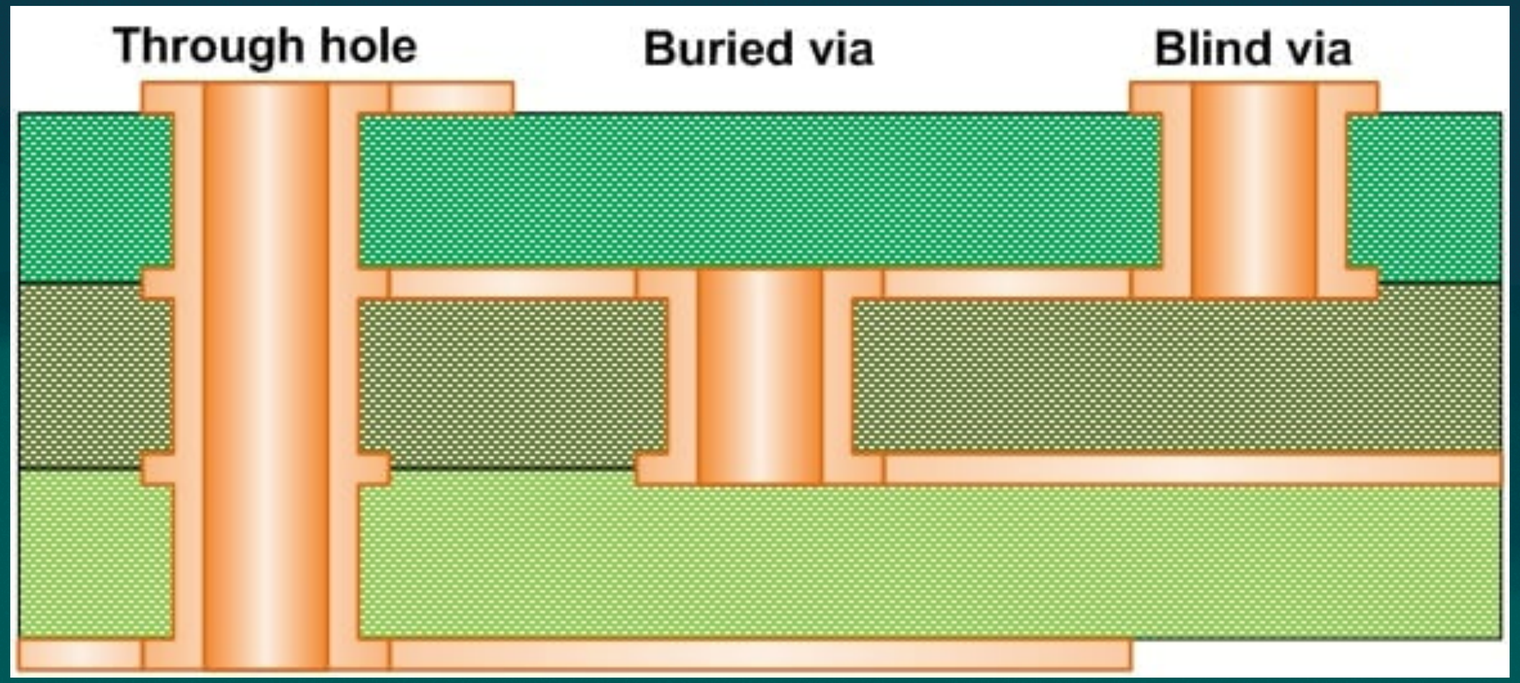
- This is what makes PCBs so useful!
- Usually straight lines with 45 or 135 degree bends
  - Why not 90 degrees?
- Wider trace width = **less** resistance
- Lower trace length = **less** resistance



vias

the things that connect layers

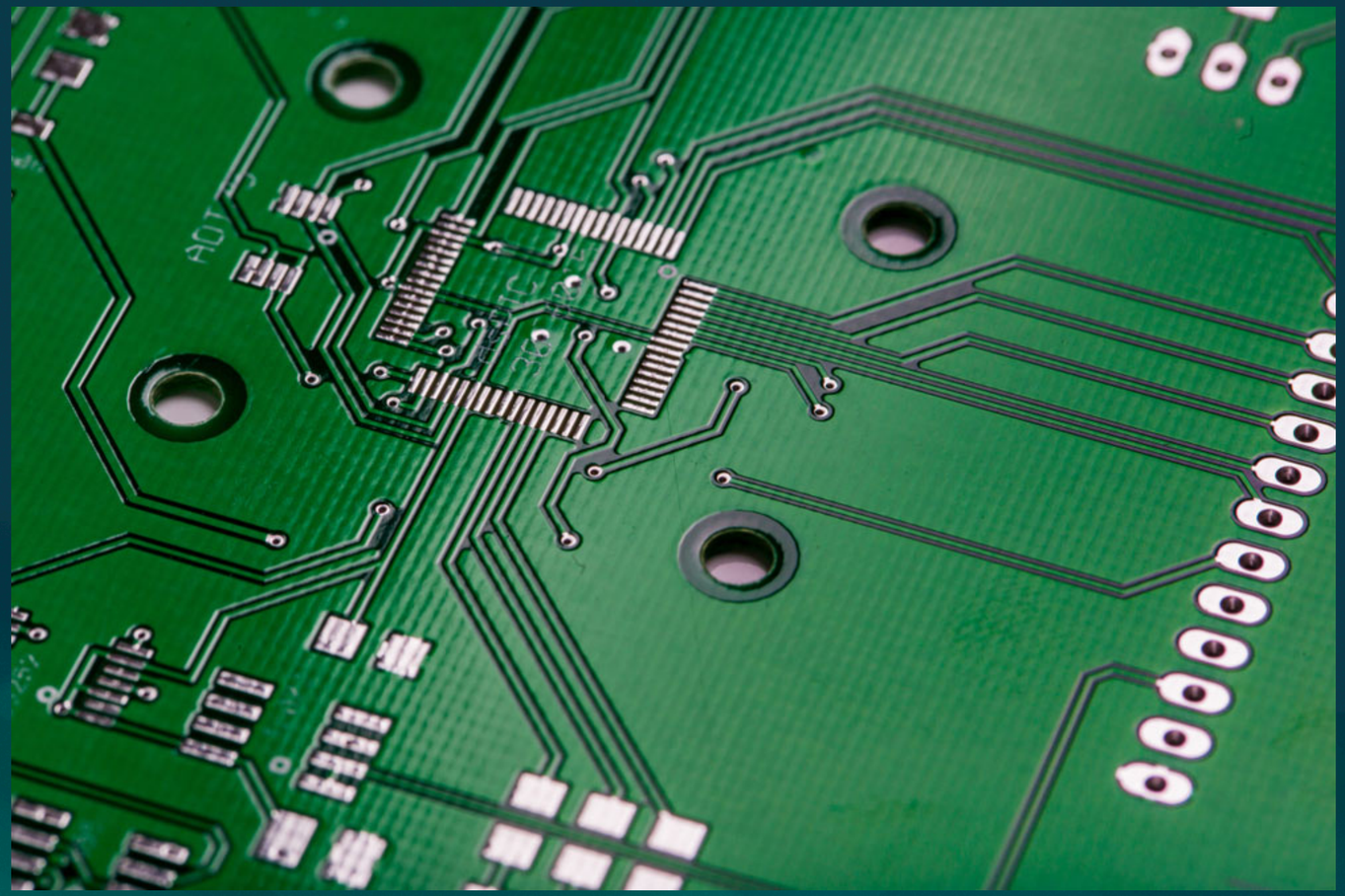
- Connect traces on different layers
- Through holes (normal) vias
- Blind vias ($)
- Buried vias ($$$)



planes

big areas of copper

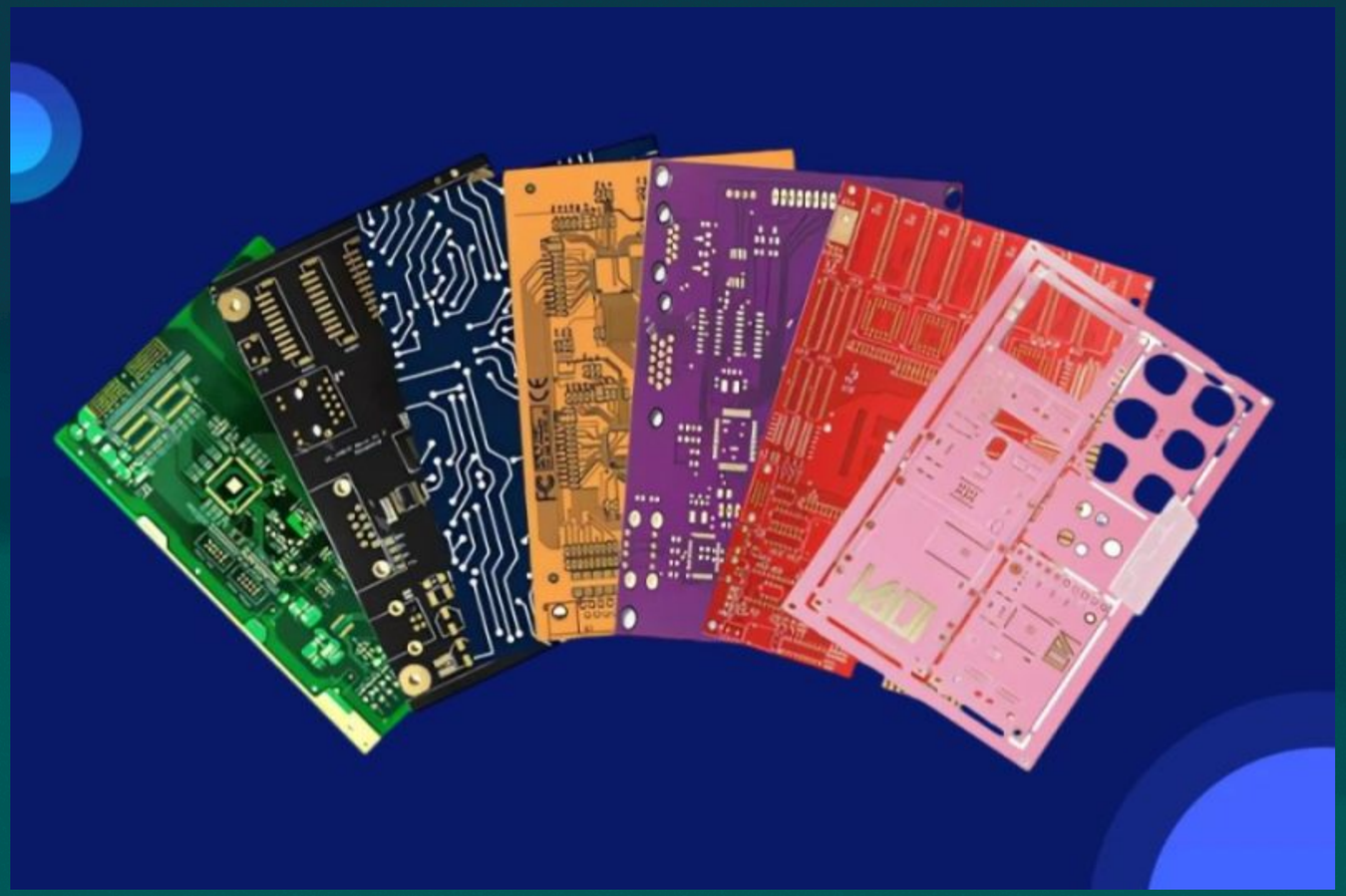
- Good practice to have a ground plane to reduce noise
- Nice to have a power plane
- Signal planes are nice to have! Usually not used in 2-layer pcbs
- Planes make routing much easier!



solder mask

the thing that keeps solder from going everywhere

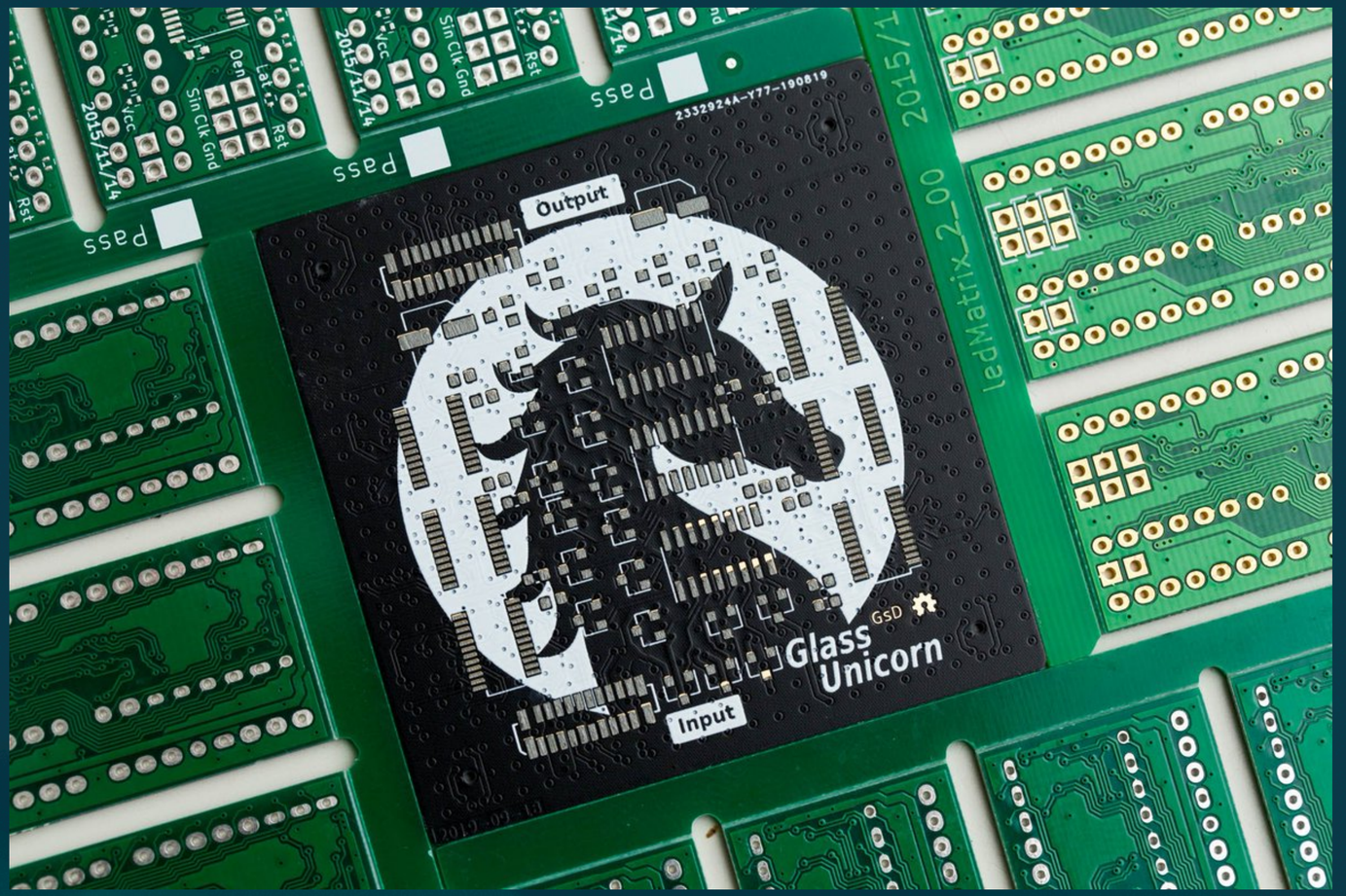
- Protects your board
- Prevents trace oxidation
- Prevents solder riding
- ‘Painted’ on
- COLORS

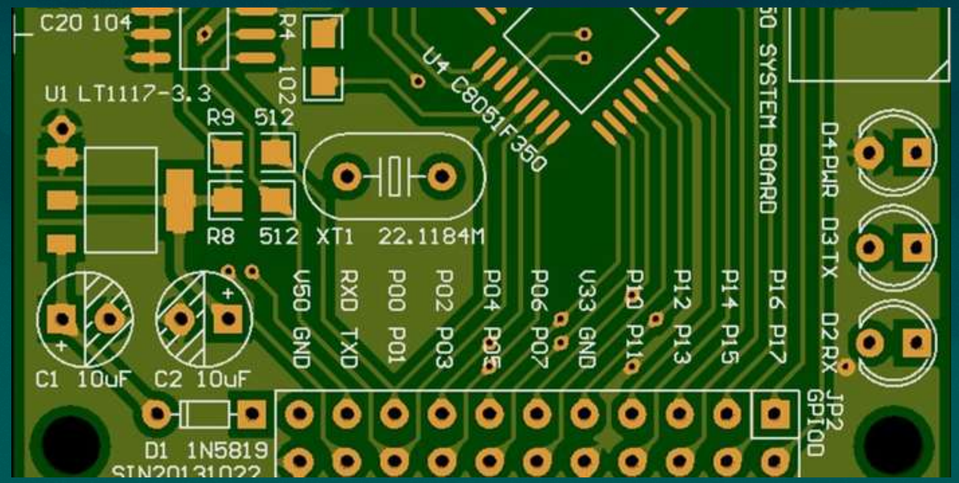


silkscreen

labels and art!

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





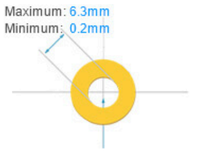
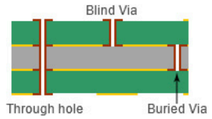
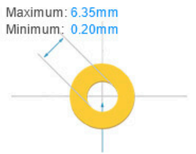
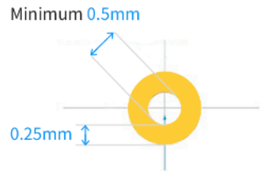
tolerances

the stuff the manufacturer cares about

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

tolerances

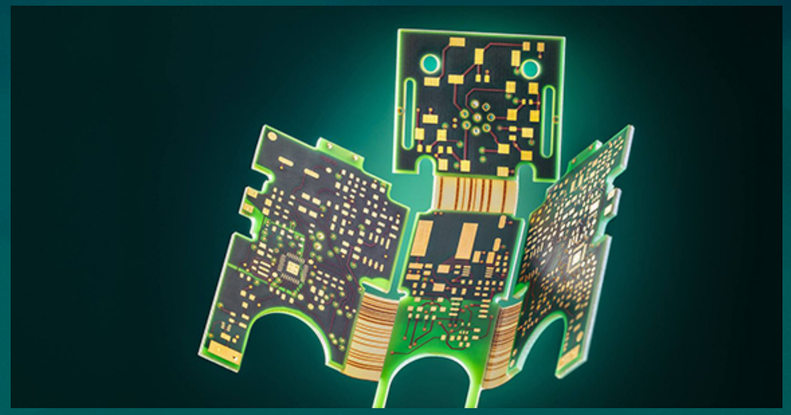
from the manufacturer

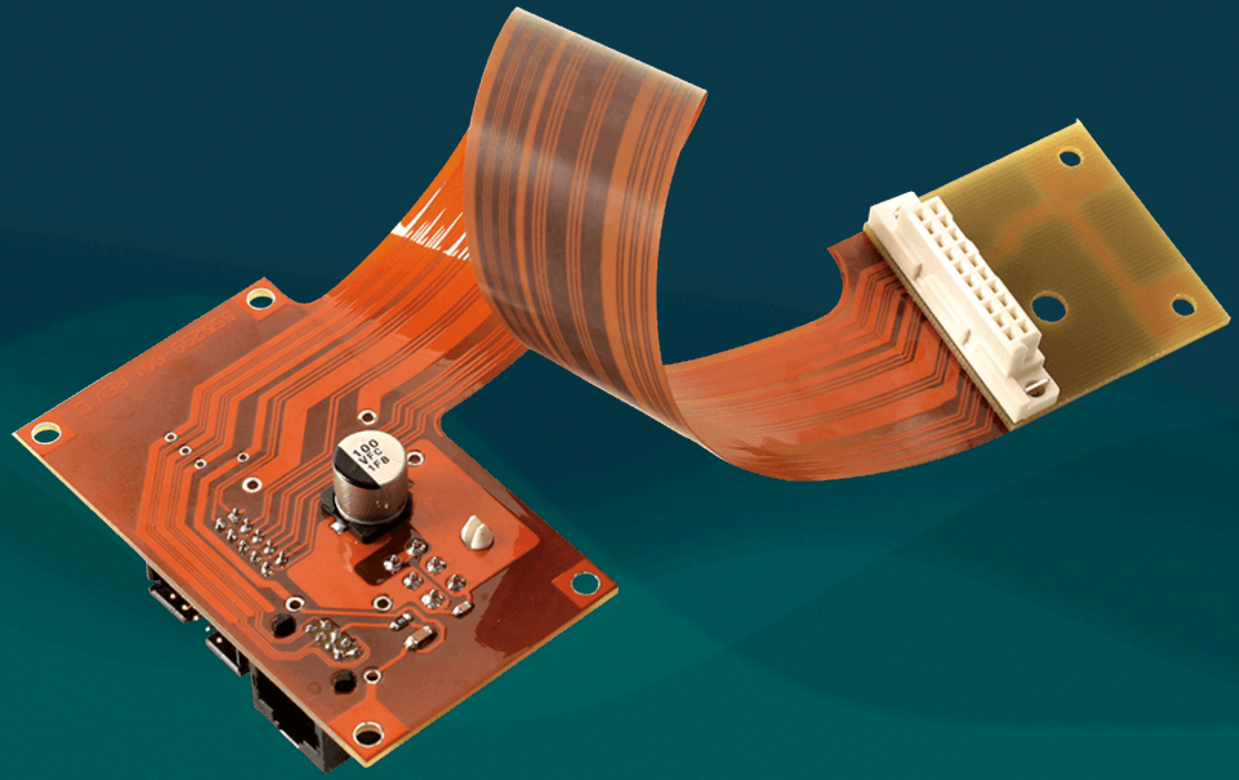
Features	Capability	Notes	Patterns
Drill Hole Size	0.15mm - 6.30mm	1 & 2 Layer PCB: 0.3 - 6.3mm Multi-Layer PCB: 0.15 - 6.3mm (0.15mm more costly)	
Drill Hole Size Tolerance	+0.13/-0.08mm	e.g. for the 0.6mm hole size, the finished hole size between 0.52mm to 0.73mm is acceptable.	
Blind/Buried Vias	Don't 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

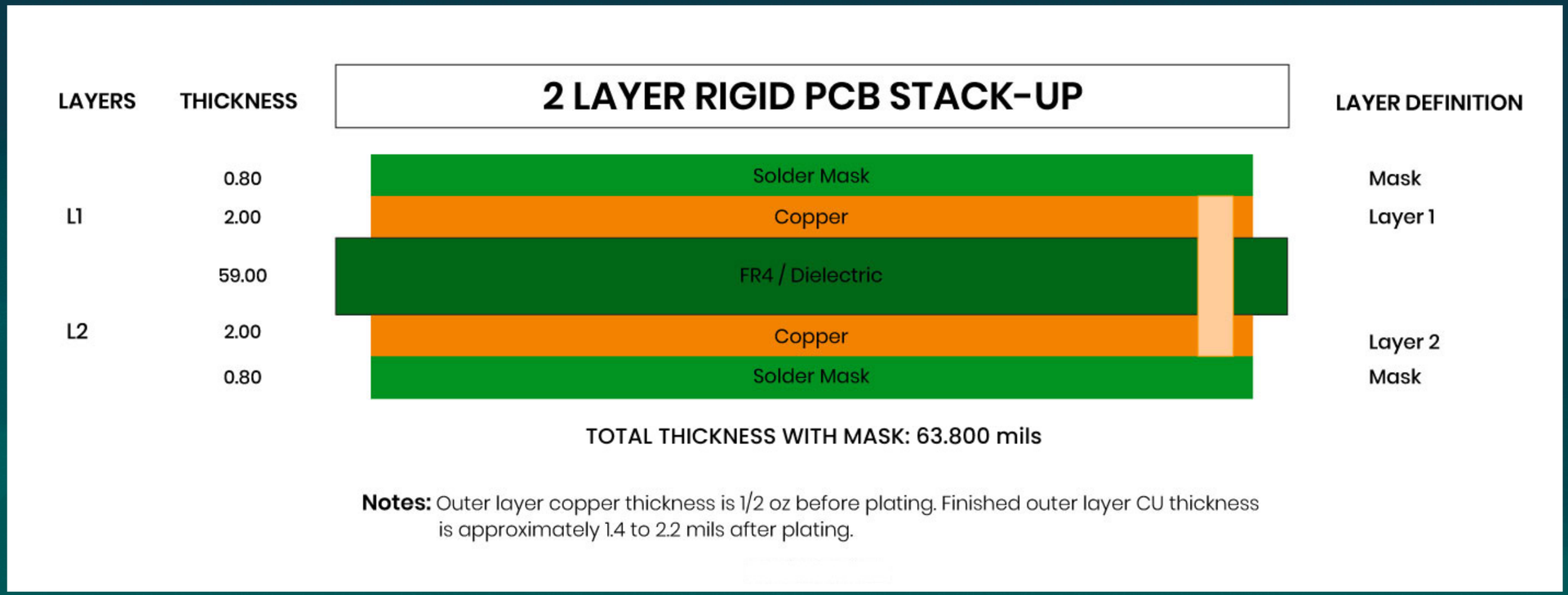
- FR-4 (we will be using this)
- Flex
- Rigid flex
- Others





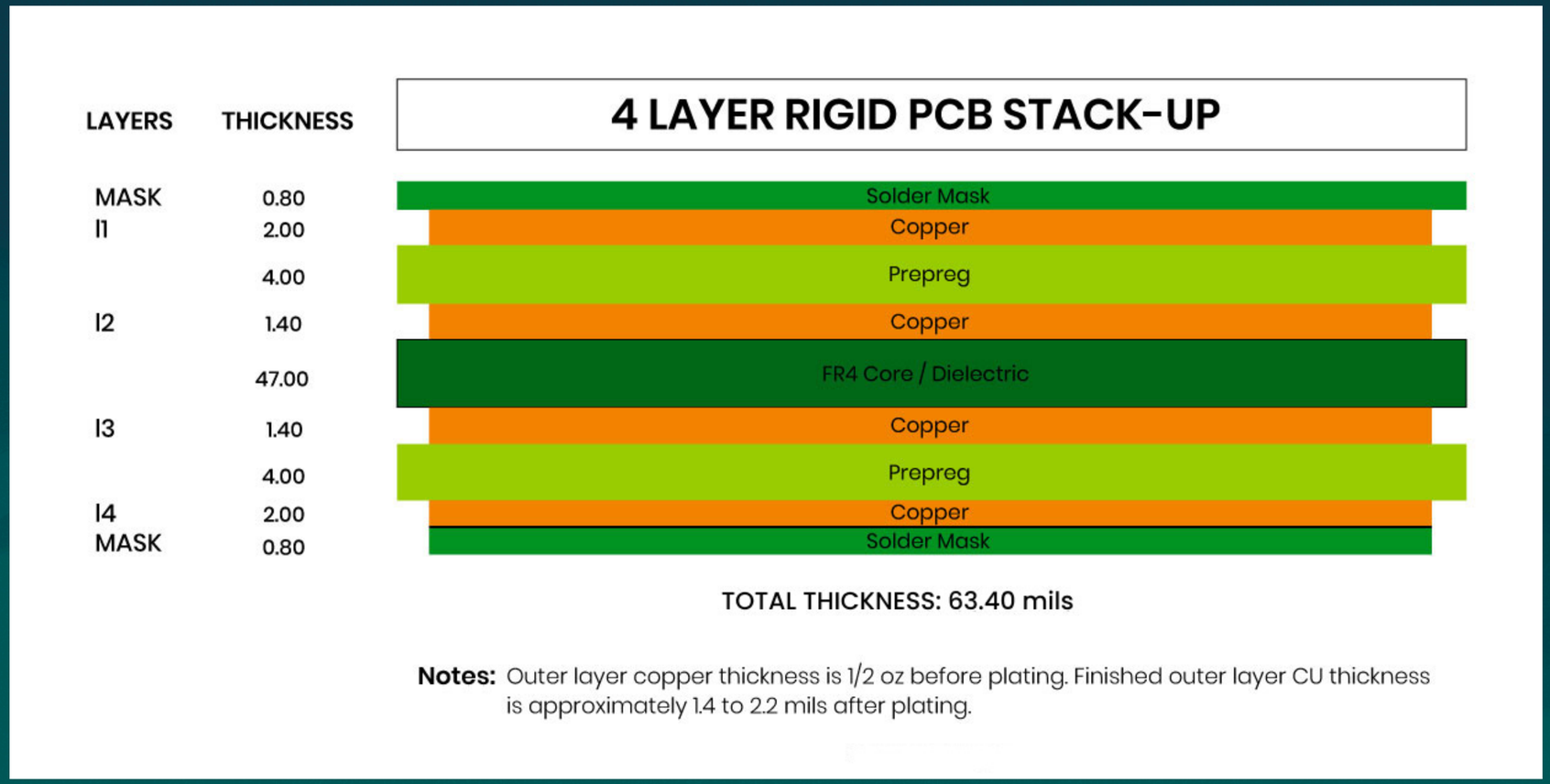
types of PCBs
there are many

- 2 layer PCB



types of PCBs
there are many

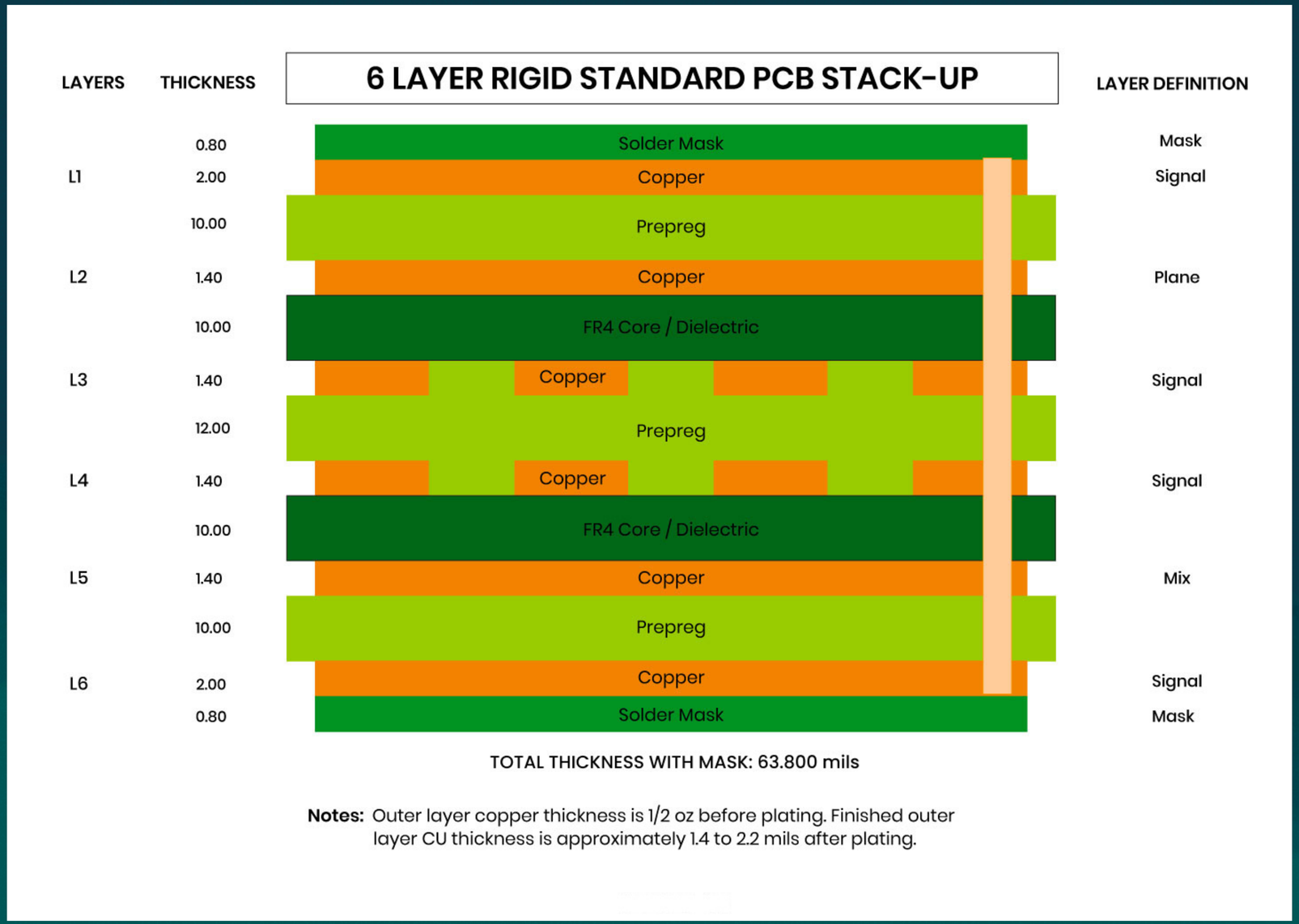
- 4 layer PCB



types of PCBs

there are many

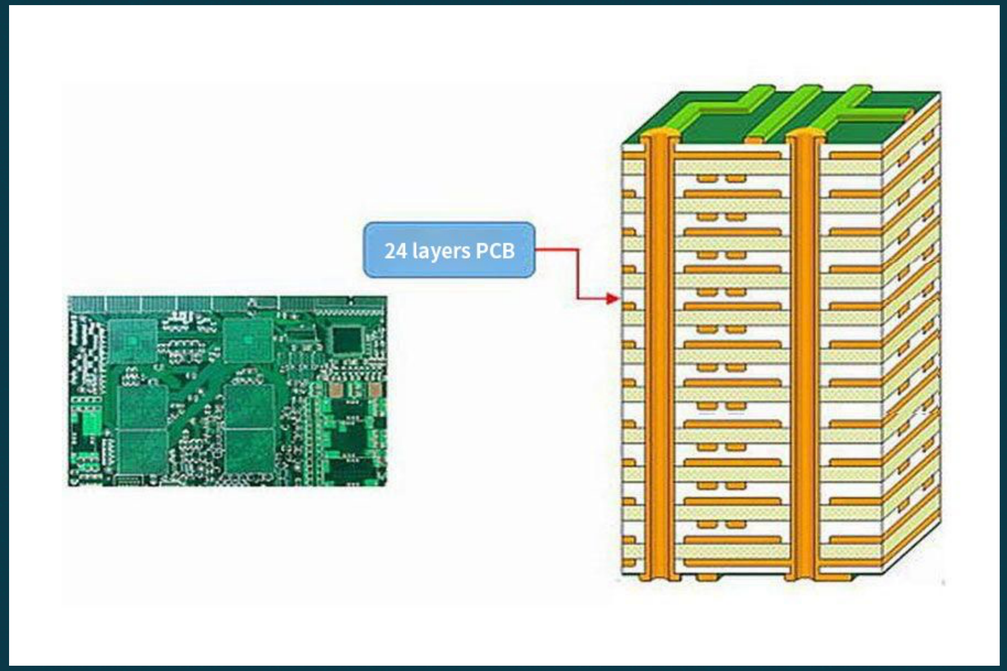
- 6 layer PCB

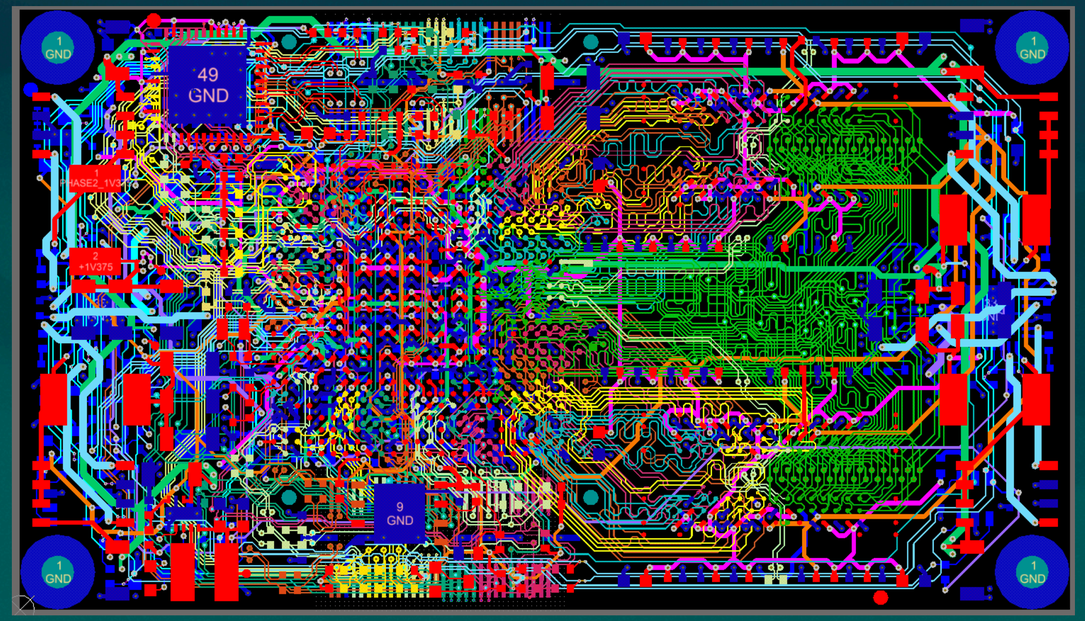


types of PCBs

there are many

- 20+ layer PCBs
- Computing hardware





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

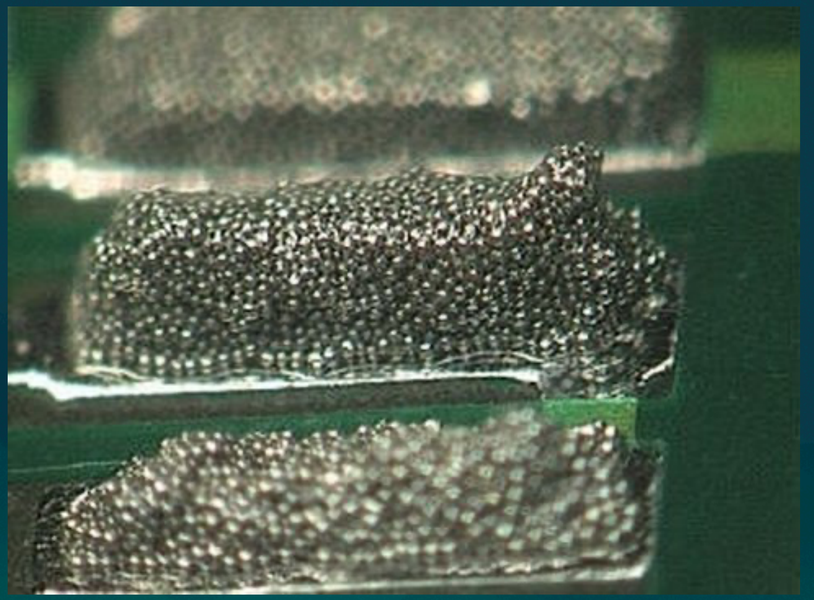
how is the board made

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

assembly

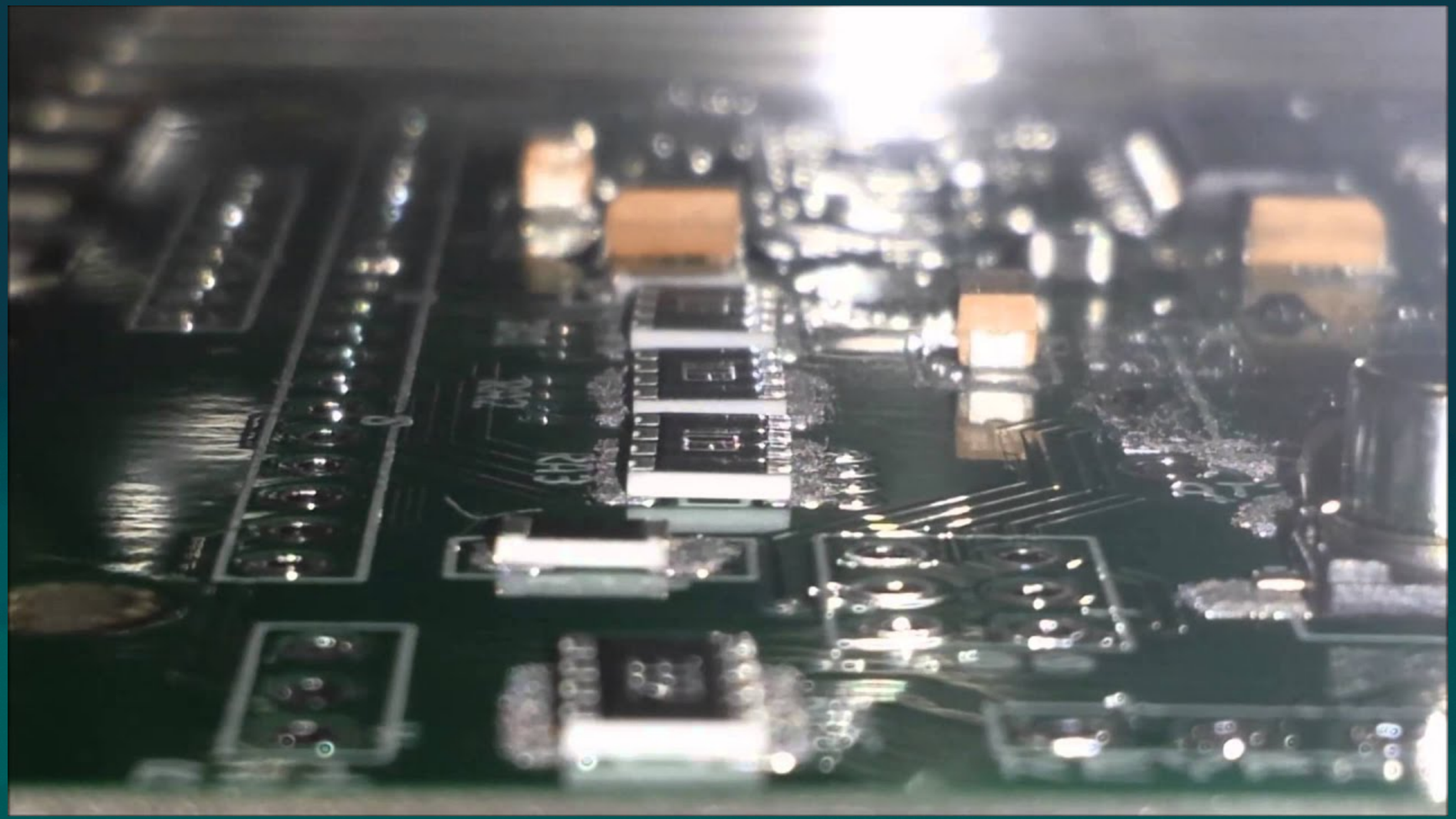
how to put components on

- **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

what you will be doing!



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

about our project

what you will be making

- **Steps of PCB Design:**
- Circuit Design
- Schematic Capture
- Layout
- Routing
- Order
- Assembly