THE ART AND SCIENCE OF PCB DESIGN

IAP 2025

Lecture 05 - PCB LAYOUT

PCB Layout 1/2

@ MIT

Outline

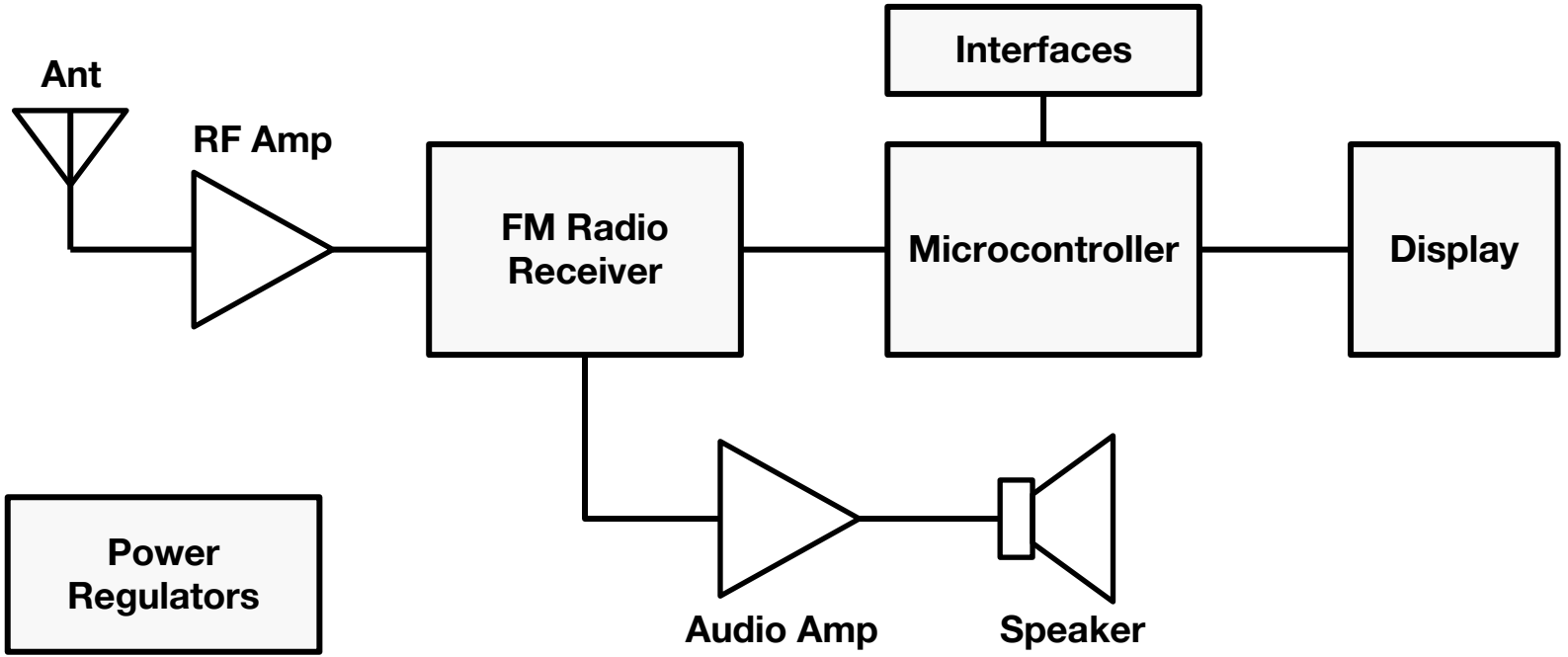
THE ART AND SCIENCE OF PCB DESIGN

IAP 2025

- Introduction
- Before the PCB
- PCB Construction
- SMT Components
- Basic PCB Layout
- Summary

Schematic

Block Diagram



System Design

Course Project

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

Identify Problem ✓

Devise Solution ✓

Identify System Parameters ✓

Create/Model System ✓

Design PCB!

FM radio receiver

Course Project!

Later topics

Fabrication → Test/Debug

@ MIT

*How were PCBs Conceived?

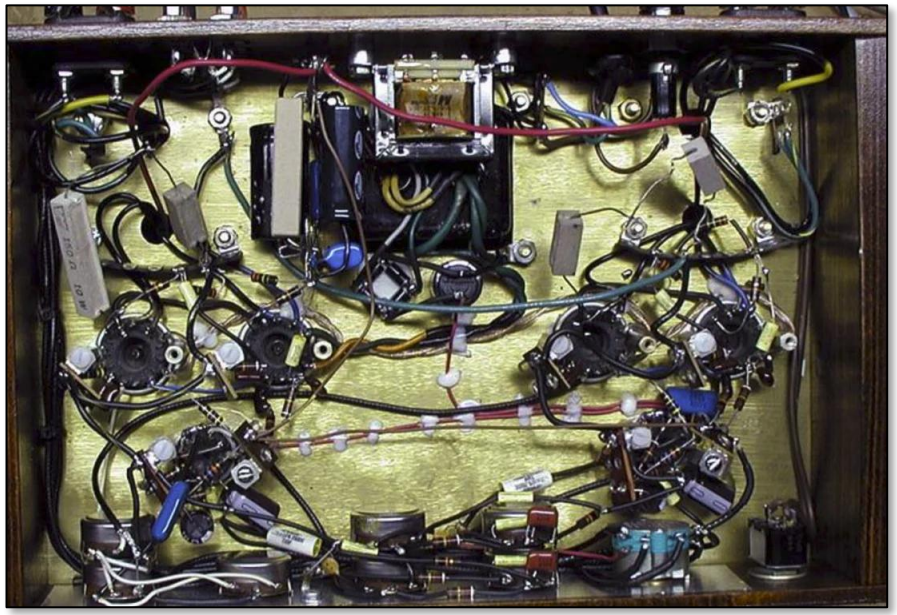
*Not an exact history

Electronics Before PCBs

Point-to-Point Connections

A common method for wiring was simple point-to-point connections

- **Components mounted to an enclosure with wires soldered between leads**
- **It works, but its messy**

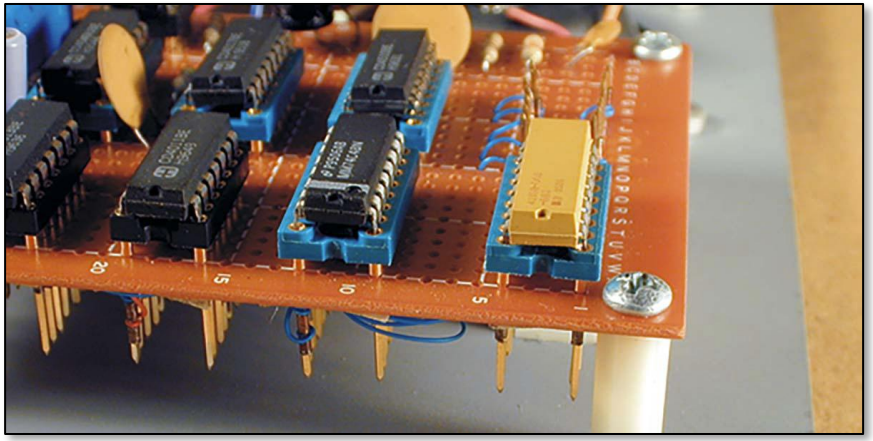


https://www.psaudio.com/blogs/pauls-posts/pcb-vs-point-to-point

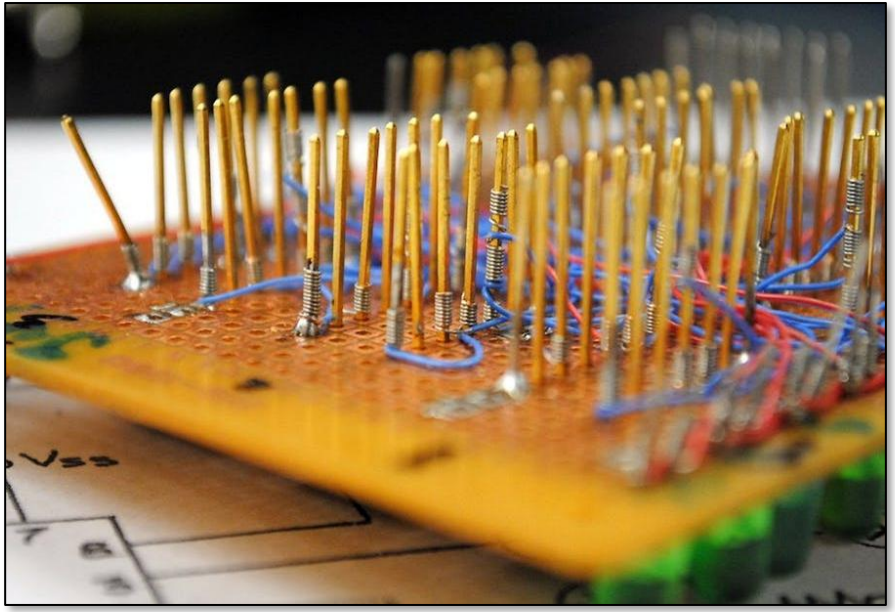
Electronics Before PCBs

Wire Wrap

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025



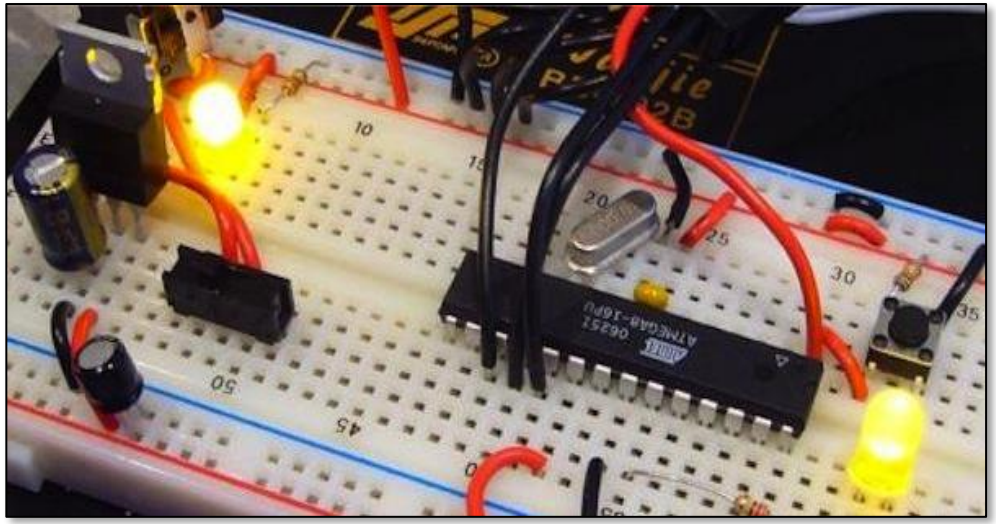
https://www.nutsvolts.com/magazine/article/wire_wrap_is_alive_and_well



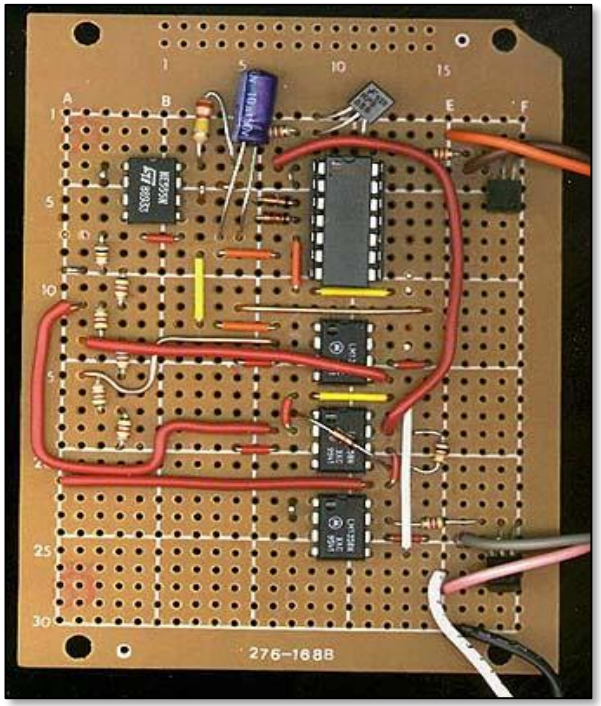
https://www.hackster.io/ss5r/wire-wrapping-not-just-a-guide-f3fc74

Electronics Before PCBs

Breadboards and Perfboards



https://learn.sparkfun.com/tutorials/how-to-use-a-breadboard/why-use-breadboards



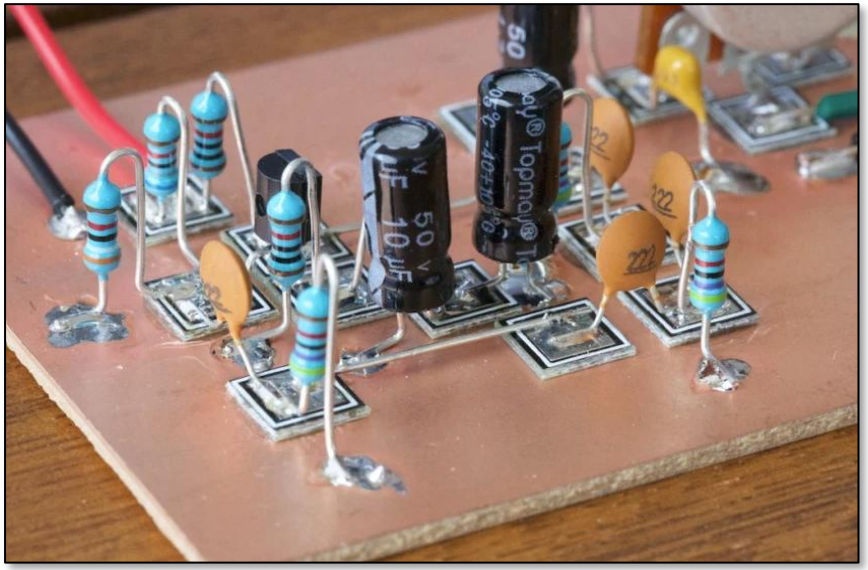
https://www.bcae1.com/circuitboardetch.htm

Electronics Before PCBs

Manhattan and Dead Bug Style

THE ART AND SCIENCE OF PCB DESIGN

IAP 2025



https://nomadic.blog/2015/08/19/initial-attempts-at-manhattan-construction-and-a-tone-generator/

https://makezine.com/article/technology/a-fine-example-of-dead-bug-style-circuit-wiring/

PCB Construction

Inventing the PCB

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

How can we make circuit construction more efficient and compact?

Generally fine to have circuits configured in a planar format and build a form-fitting enclosure around it

Have existing, well-developed lithographic processes at industrial scales

For low-power circuits (i.e., most circuits), a small conductor cross section is sufficient

Typically exists global nets that connect to may devices (e.g., GND, VCC, VDD)

PCB Construction

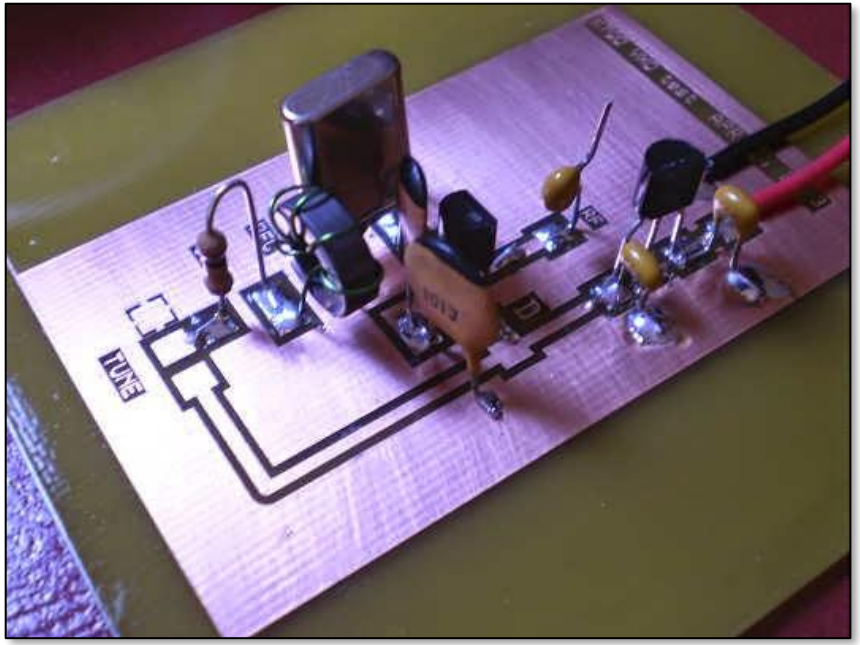
Single-Sided Board (Muppet Style)

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

Like Manhattan Style, we use a non-conductive substrate laminated with a sheet of copper. Traces are cut out to connect components together

- **We create pads for devices to be soldered to**
- **The copper regions around the traces can act as one large ground plane**

<u>We can do better!</u>



https://www.amateurradio.com/manhattan-style/

PCB Construction

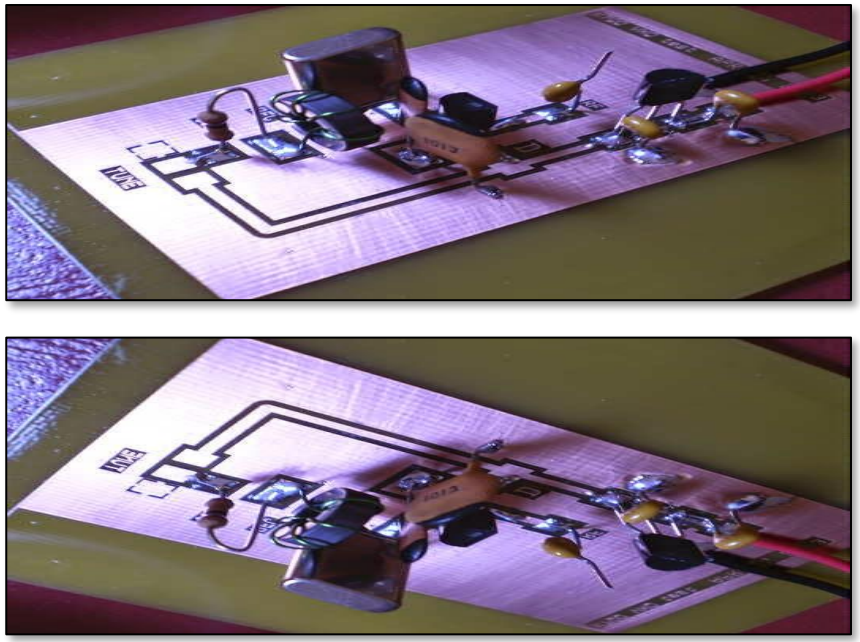
Double-Sided Board

We add a sheet of copper to the other side and cut traces

- **Adds an additional plane by which to route connections**
- **Traces can be cross each other on either side**

But this leaves us with two electrically isolated circuits.

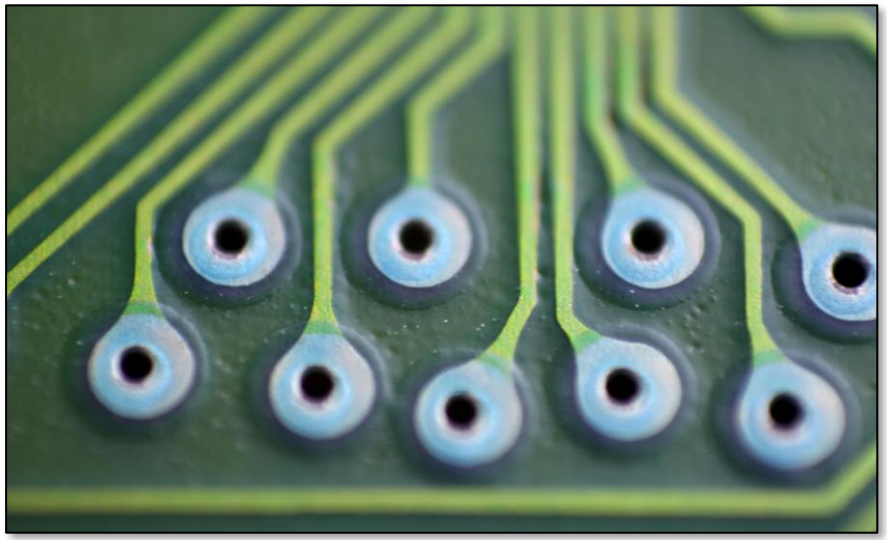
How do we connect traces from one layer to the other?



https://www.amateurradio.com/manhattan-style/

PCB Construction

Vias



https://resources.altium.com/p/changing-pcb-reference-planes-during-routing-multilayer-boards

We take the end of a trace on one side of the board and drill a hole through it and the end of another trace on the other side of the board, then plate that hole with metal

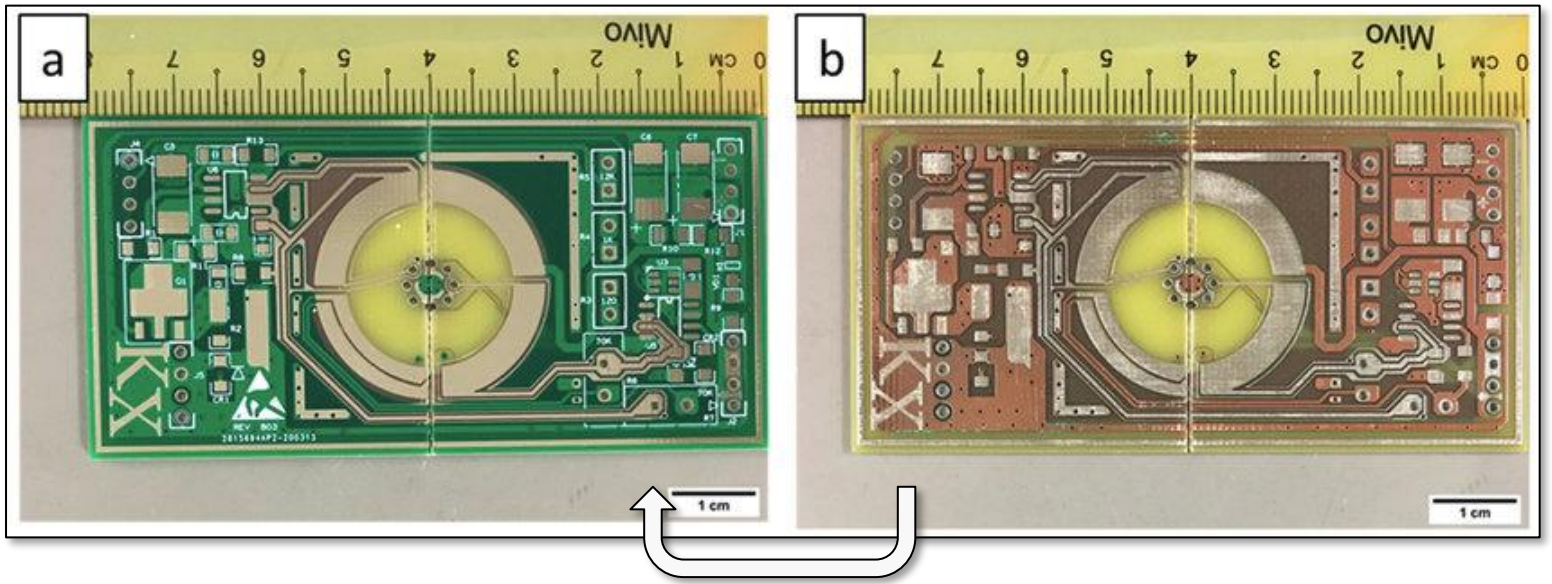
- **Creates an electrical connection that spans across layers**
- **Known as a via**
- **Large holes can be used to mount through-hole (THT) components**

PCB Construction

Soldermask

THE ART AND SCIENCE of PCB DESIGN
IAP 2025

To protect the board during assembly and from corrosion, a non-conductive coating (soldermask) is applied to copper that is not to be soldered to



https://www.researchgate.net/figure/Coarse-PCB-pieces-a-before-the-removal-of-solder-mask-and-b-after-the-removal-of_fig7_355966696

@ MIT

PCB Construction

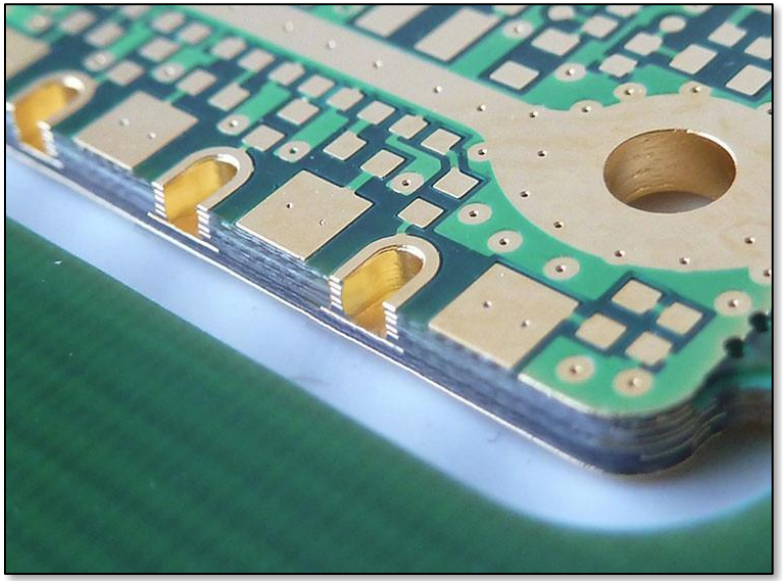
Surface Finish

THE ART AND SCIENCE OF PCB DESIGN

IAP 2025

Copper that is not covered by soldermask (usually pads that will be soldered to later), is plated with metal to prevent corrosion and improve solderability. Common plating types:

- **ENIG (gold plating)**
- **HASL (solder plating)**
- **ENEPIG (palladium/gold plating)**
- **ImAg (silver plating)**
- **ImSn (tin plating)**
- **OSP (organic cover)**



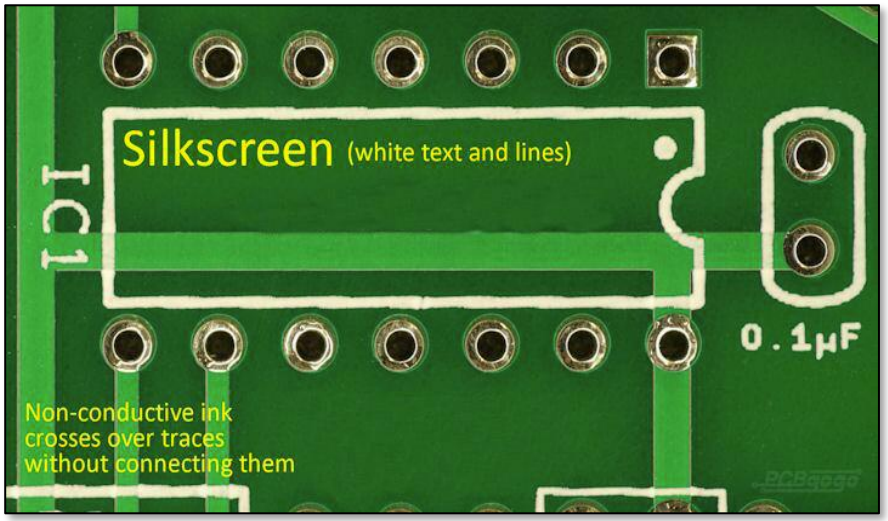
https://www.setgmbh.de/en/technology/plated-through-slots-and-sideplating

PCB Construction

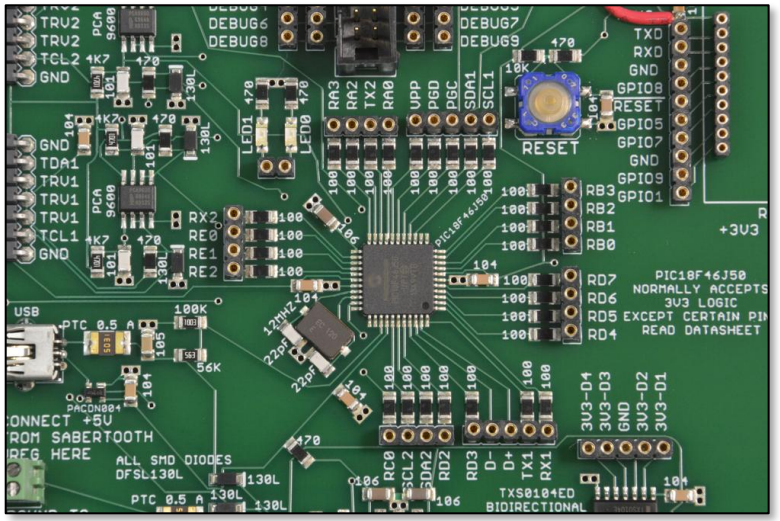
Silkscreen

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

Text and markings can be added on top of soldermask with non-conductive ink (silkscreen). Critical for marking component locations and orientations



https://www.pcbgogo.com/blog/What_Is_Silkscreen_On_a_PCB_.html



https://resources.altium.com/p/your-guide-pcb-silkscreen

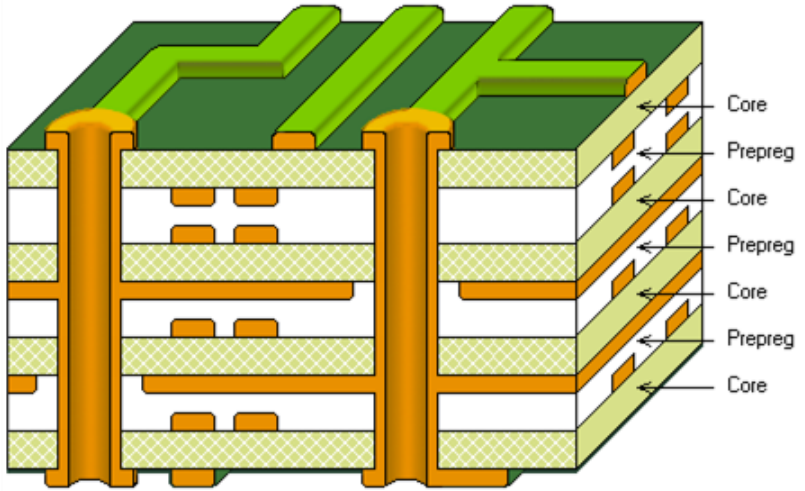
PCB Construction

Putting it all Together

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

Substrates and their copper laminates can be stacked together to create multilayer PCBs, with layers connected to another using vias.

- **Typically 2-20 conductive layers**
- **The non-conductive substrates are known as dielectrics which can range in materials depending on the application**
- **Soldermask/surface finish/silkscreen on top and bottom**



https://morepcb.com/8-layer-pcb-stack-up/

SMT Components

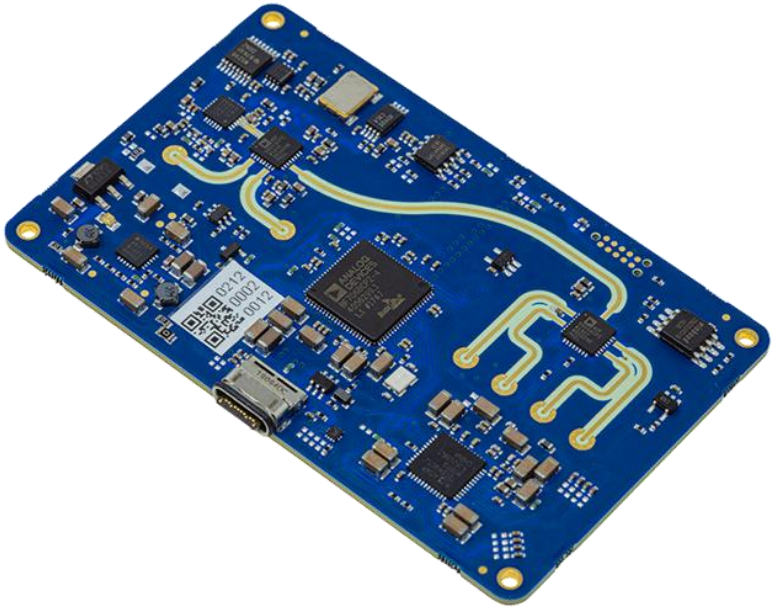
Surface Mount Technology

THE ART AND SCIENCE of PCB DESIGN

IAP 2025

PCB components are typically made to be mounted on top of PCBs using Surface Mount Technology (SMT)

https://electronicslovers.com/2018/10/surface-mount-electronic-components-and-their-types.html



https://www.analog.com/en/design-center/evaluation-hardware-and-software/evaluation-boards-kits/eval-tinyrad.html#eb-overview

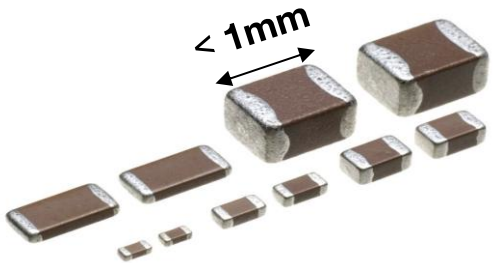
SMT Components

Two-Lead Passive Chip Components

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

Capacitors, inductors, and resistors come in standard sizes

- **Each size defines a standard copper pad design (footprint) that can be used on the board**



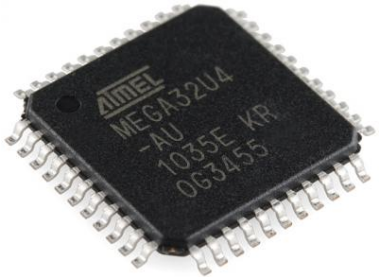
https://www.doeeet.com/content/eee-components/passives/five-not-so-general-purpose-mlcc/

Metric Code (0.1mm)	Imperial Code (0.01 in)
0603	0201
1005	0402
1608	0603
2012	0805

First two digits: approx. length
Last two digits: approx. width

SMT Components

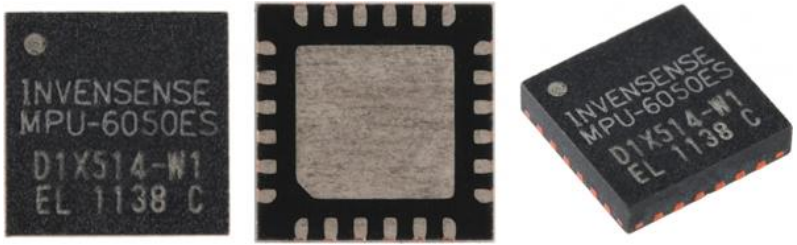
Multi-lead Integrated Circuits

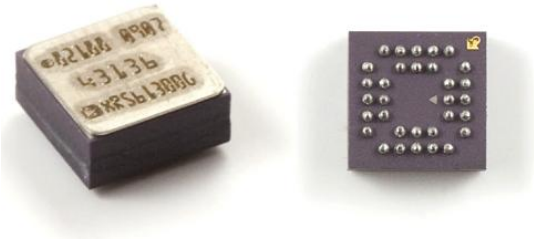


THE ART AND SCIENCE of PCB DESIGN
IAP 2025

Come in many different packages:

- **SOP, SOIC, TSSOP, QFP have splayed pins that extend beyond the package**
- **QFN, DFN, MLF have pins on the side of the package, but they do not protrude**
- **BGA, WLP, LGA, CSP have pins located and soldered on the bottom of the package**





https://learn.sparkfun.com/tutorials/integrated-circuits/all

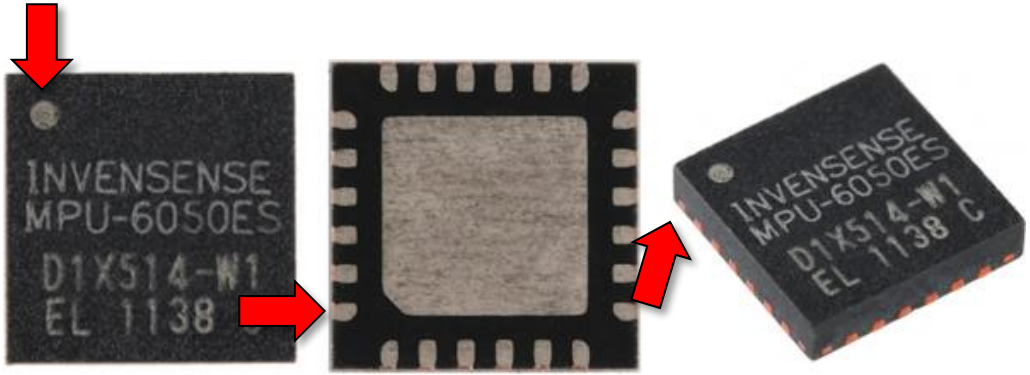
@ MIT

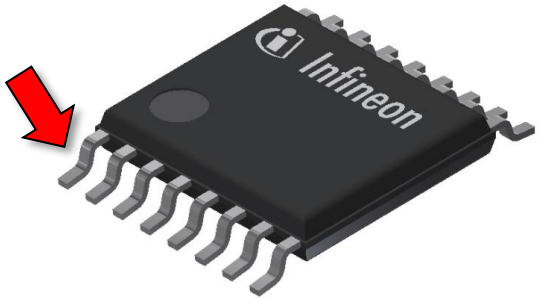
SMT Components

Multi-lead Integrated Circuits

THE ART AND SCIENCE of PCB DESIGN
IAP 2025

Dot or notch on the corner of IC refers to pin 1





https://learn.sparkfun.com/tutorials/integrated-circuits/all

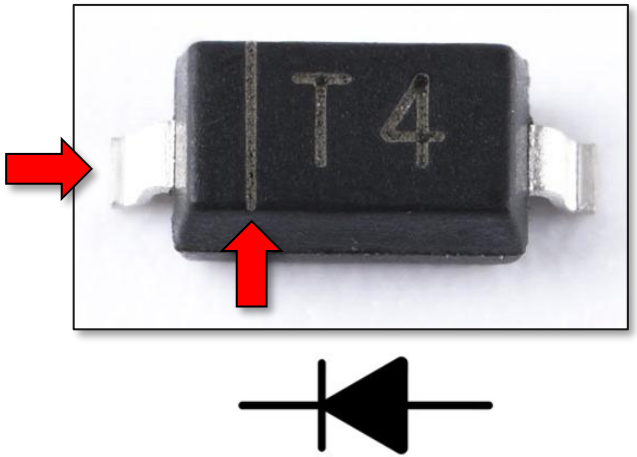
https://www.infineon.com/cms/en/product/packages/PG-TSSOP/PG-TSSOP-16-8/

SMT Components

Diodes

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

Notched side of an SMT diode refers to its cathode



https://etstore.in/index.php/product/1n4148w-t4-smd-fast-switching-diode-sod-123-75v-150ma/

SMT Components

Footprints

All SMT components will require a footprint in order to be mounted onto a PCB. This will contain:

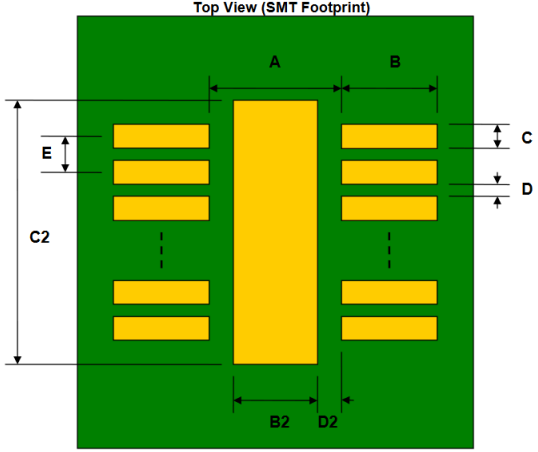
1. **Cutouts for pins to be soldered to**
2. **Markings to show orientation (i.e. pin 1 dot)**
3. **Thermal pad/vias**
4. **Other labels/designators**

These specifications are typically defined in manufacturer's datasheet

DFN-8/MLP-8 to DIP-12 SMT Adapter (1.27 mm pitch, 6 x 8 mm body)

REF.	DIMENSIONS					
	mm			inches		
	MIN.	TYP.	MAX.	MIN.	TYP.	MAX.
A		7.00				
B		0.80				
C		0.50				
B2		3.40				
C2		4.30				
D		0.77				
D2		1.80				
E		1.27				
F					0.600	
G					0.100	
H					0.025	

Top View (SMT Footprint)

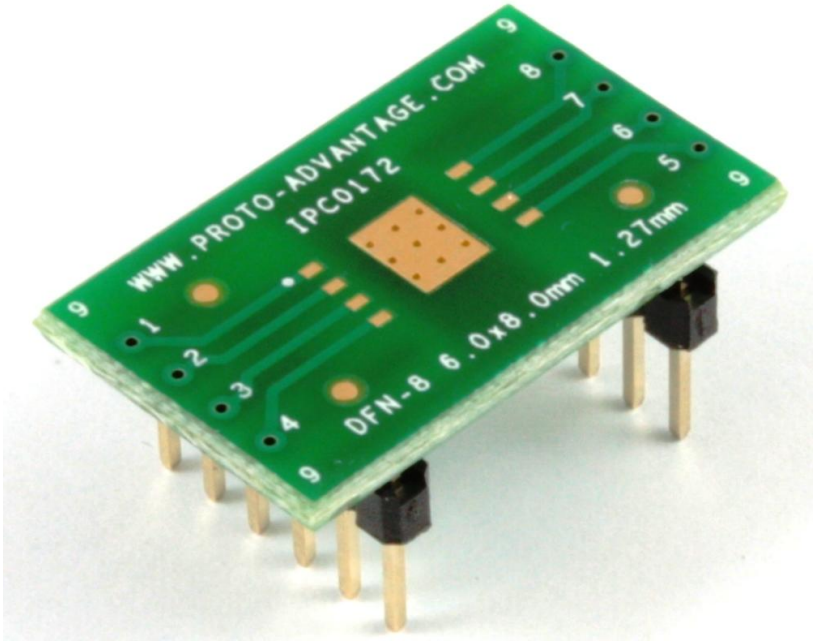


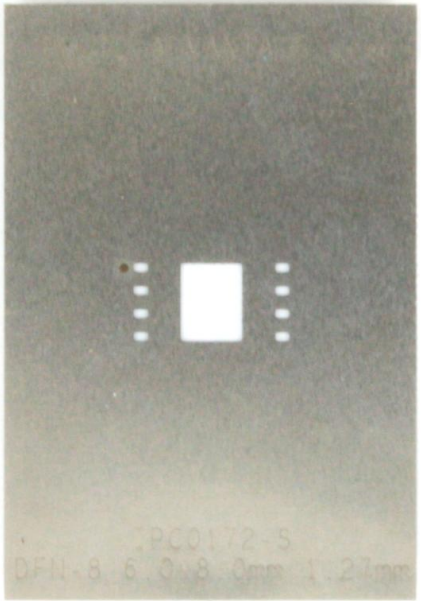
(Representative drawing only - not to scale)

SMT Components

Footprints

Footprints also allow us to specify solder stencils





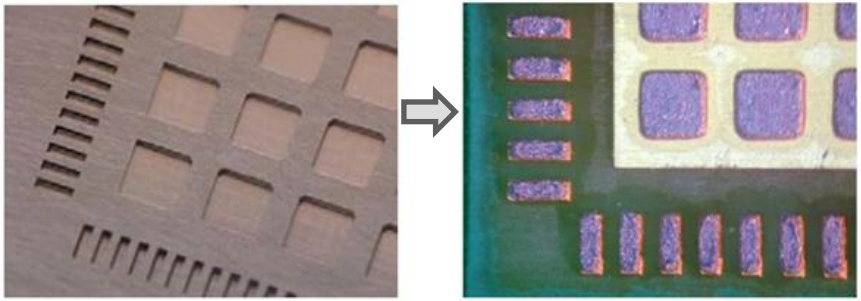
https://www.chipquik.com/store/product_info.php?products_id=3100182

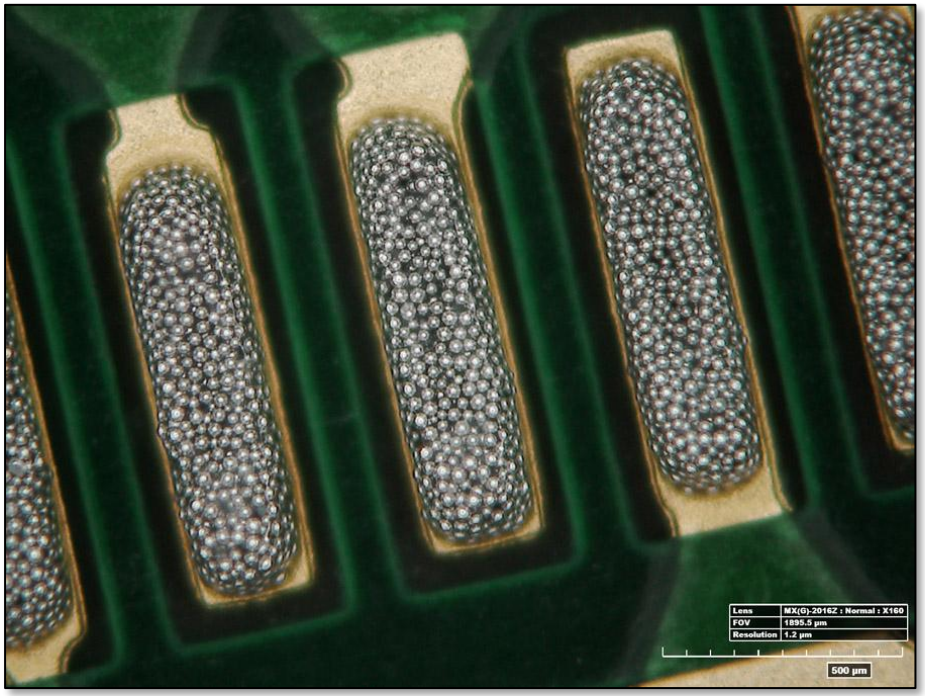
SMT Components

Solder Paste

Contains small beads of solder suspended in a liquid flux carrier.

When heated, solder melts onto pads while flux burns away

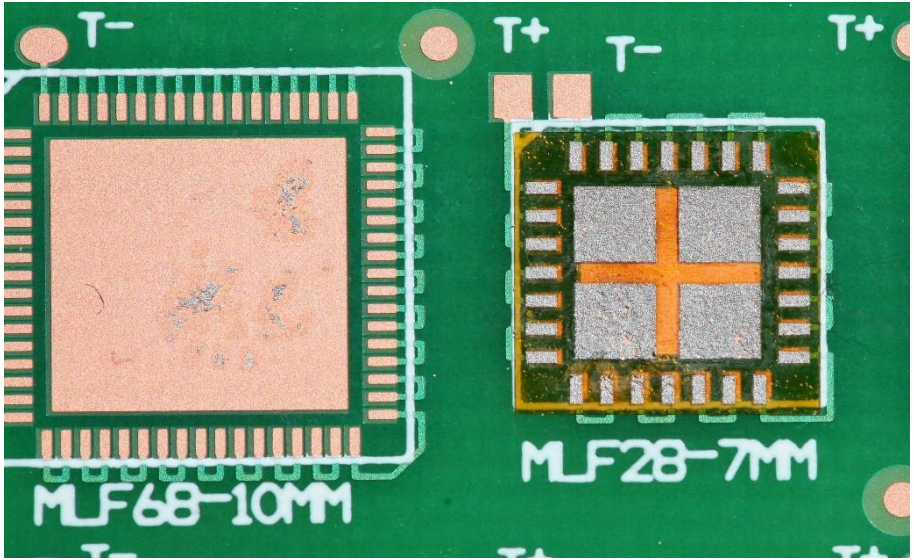




https://bltcircuitservices.co.uk/product/hg1-no-clean/

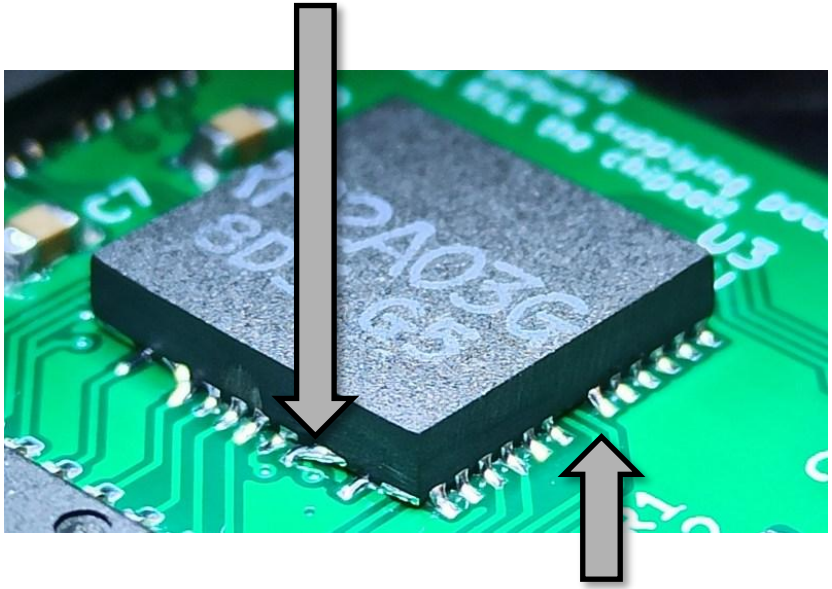
SMT Components

Solder Paste



https://www.solder.net/products/stencilmate-leadless-device-rework-stencils/

Bad reflow



Good reflow

https://hackaday.com/2022/10/02/making-a-handheld-nes-by-turning-dip-chips-into-qfn/

SMT Components

Reflow and Soldering Process

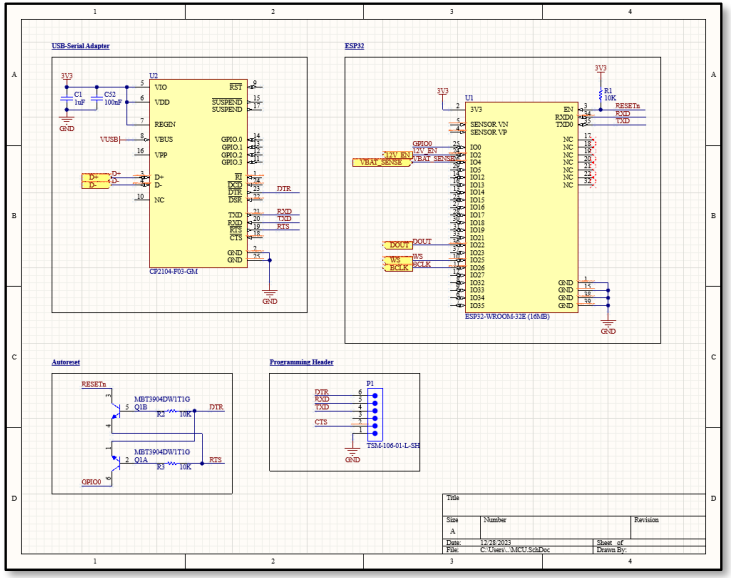
THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

1. PCB is laid flat and cleaned
2. Stencil is secured on top and aligned with pads
3. Solder paste is applied and spread over the stencil
4. Stencil is removed
5. SMT components are placed on their respective pads
6. PCB is reflowed (oven, hot plate, hot air)
7. Through-hole components are manually placed and soldered by hand

How do we layout PCBs?

Basic PCB Layout

Layout Tools



Given a schematic, how do we convert it into a board that fulfills our requirements?

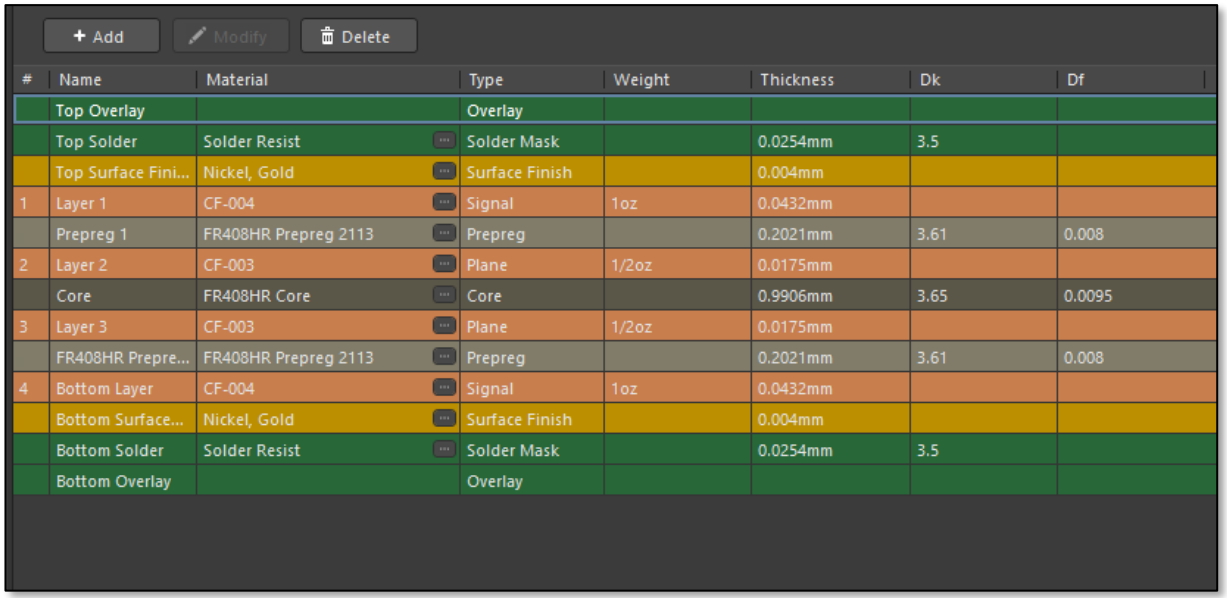
Basic PCB Layout

Layout Tools

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

Define board manufacturing parameters including:

- **Number of layers**
- **Thickness**
- **Substrate dielectric type**
- **Surface finishes and soldermask**



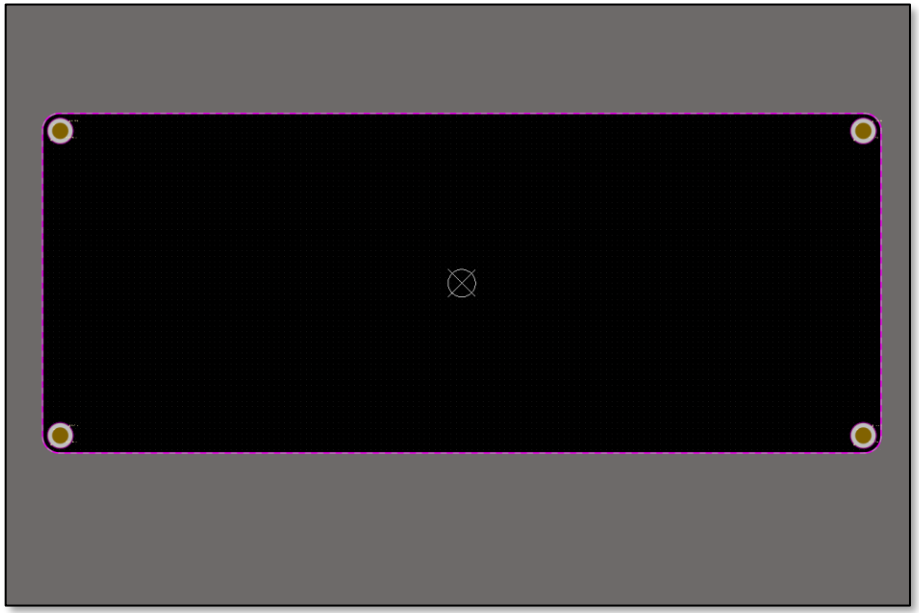
#	Name	Material	Type	Weight	Thickness	Dk	Df
	Top Overlay		Overlay				
	Top Solder	Solder Resist	Solder Mask		0.0254mm	3.5	
	Top Surface Fini...	Nickel, Gold	Surface Finish		0.004mm		
1	Layer 1	CF-004	Signal	1oz	0.0432mm		
	Prepreg 1	FR408HR Prepreg 2113	Prepreg		0.2021mm	3.61	0.008
2	Layer 2	CF-003	Plane	1/2oz	0.0175mm		
	Core	FR408HR Core	Core		0.9906mm	3.65	0.0095
3	Layer 3	CF-003	Plane	1/2oz	0.0175mm		
	FR408HR Prepre...	FR408HR Prepreg 2113	Prepreg		0.2021mm	3.61	0.008
4	Bottom Layer	CF-004	Signal	1oz	0.0432mm		
	Bottom Surface...	Nickel, Gold	Surface Finish		0.004mm		
	Bottom Solder	Solder Resist	Solder Mask		0.0254mm	3.5	
	Bottom Overlay		Overlay				

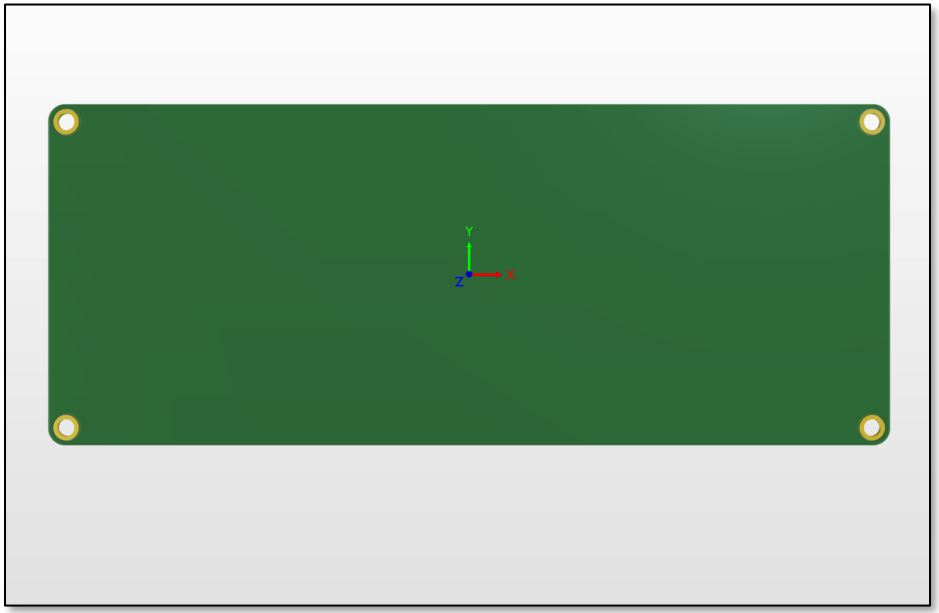
@ MIT

Basic PCB Layout

Board Shape

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025



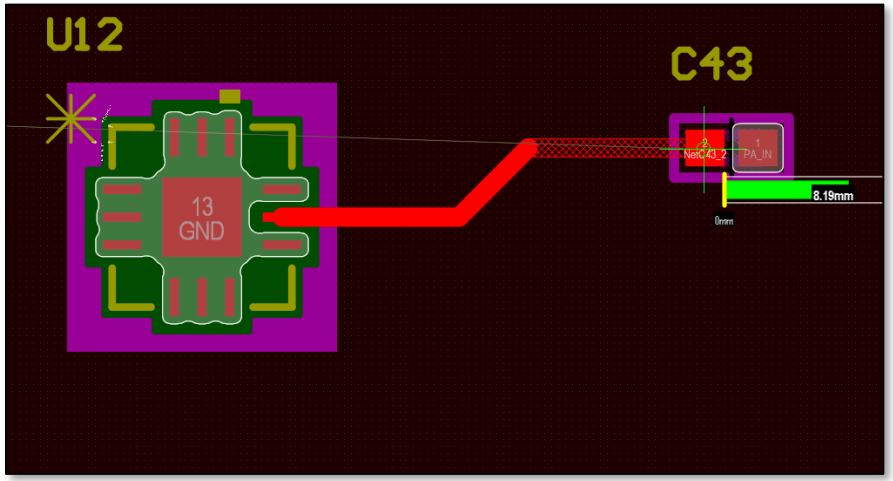


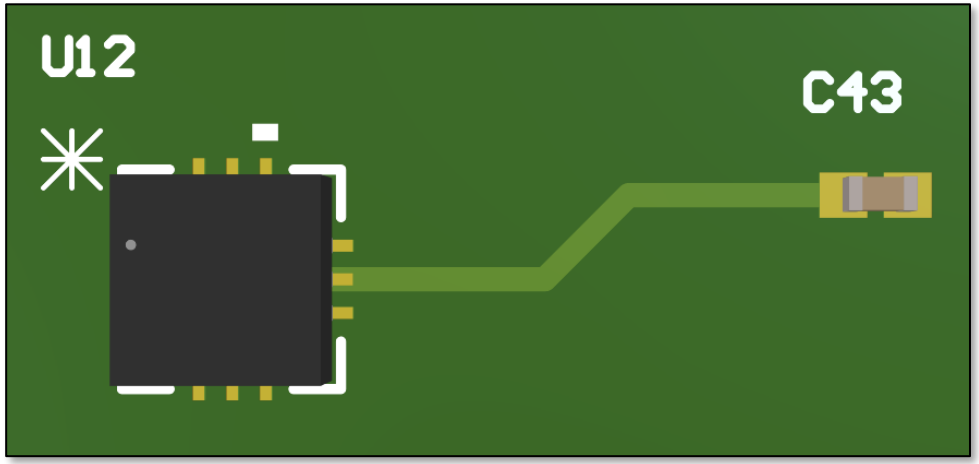
Basic PCB Layout

Routing

Red – Top Layer
Blue – Bottom Layer
Yellow - Silkscreen

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025





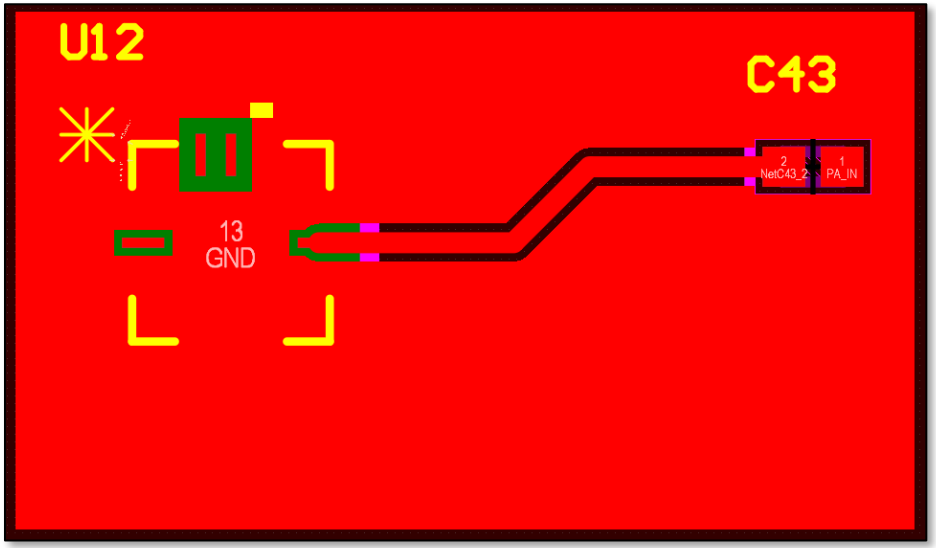
@ MIT

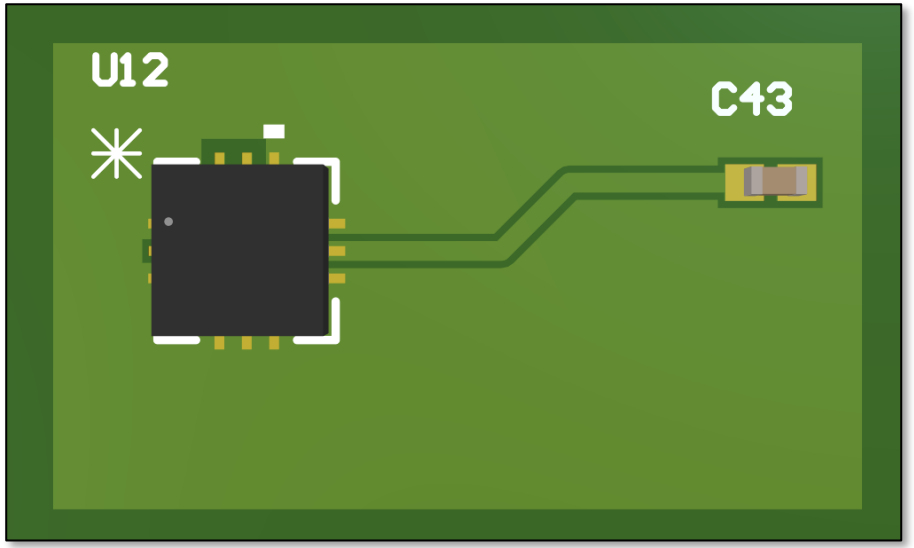
Basic PCB Layout

Copper Pours

Red – Top Layer
Blue – Bottom Layer
Yellow - Silkscreen

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025





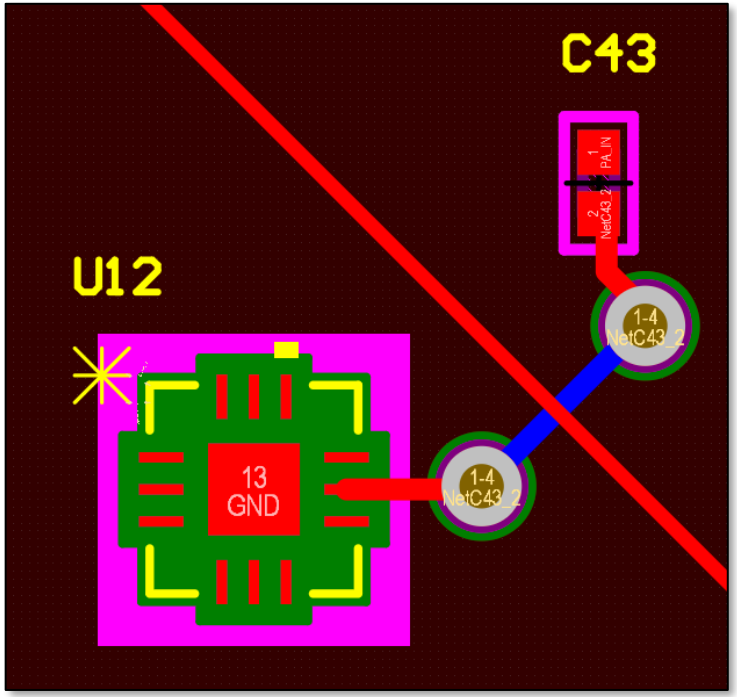
@ MIT

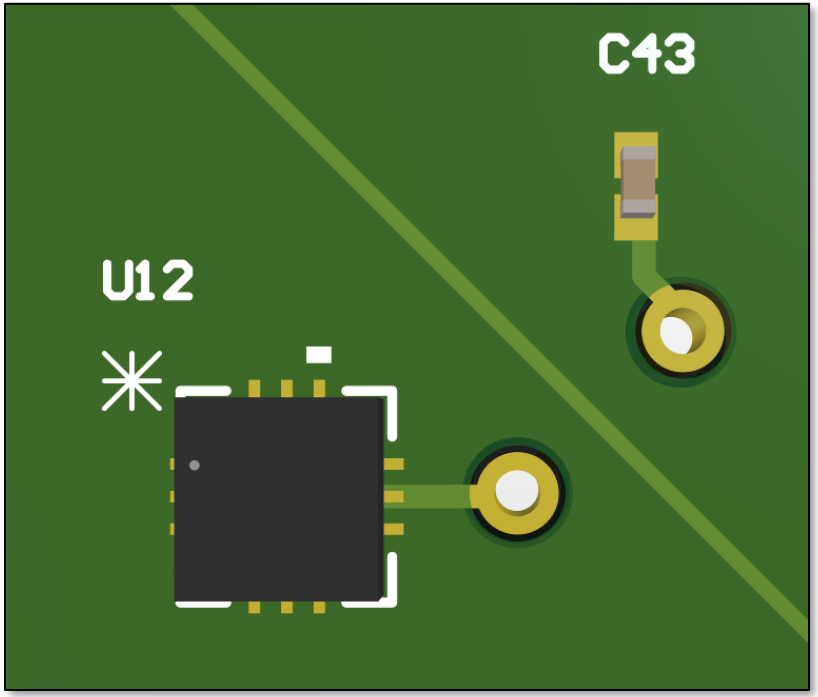
Basic PCB Layout

Vias

Red – Top Layer
Blue – Bottom Layer
Yellow - Silkscreen

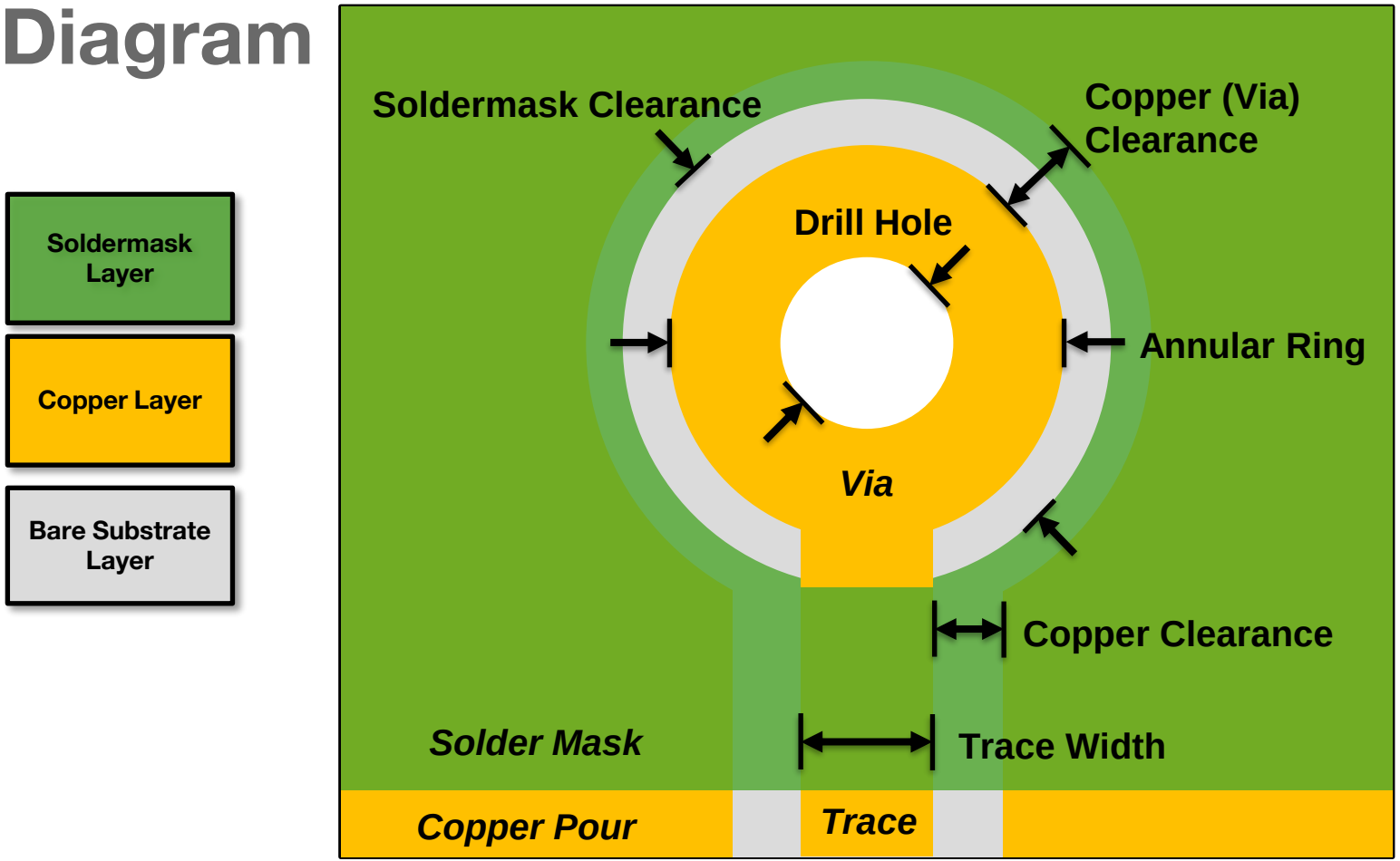
THE ART AND SCIENCE OF PCB DESIGN
IAP 2025





@ MIT

Diagram



THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

@ MIT

Basic PCB Layout

Design Rules Check

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

The aforementioned layout features are combined with Design Rule Checks (DRC) to define a board layout

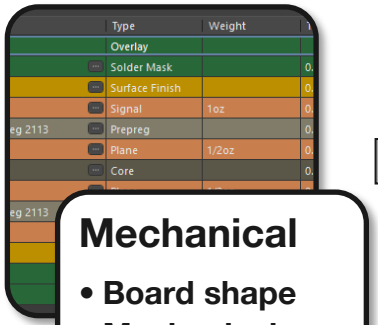
DRC provides verification during layout that certain rules are met as defined by the user. These include:

- **Minimum clearance between traces, components, and board edge**
- **Trace width bounds**
- **Pads are connected to the correct nets**

Warning messages are provided in a user-generated report or actions can be outright prevented during a DRC violation

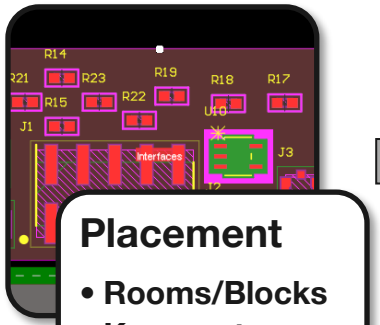
Basic PCB Layout

Design Flow



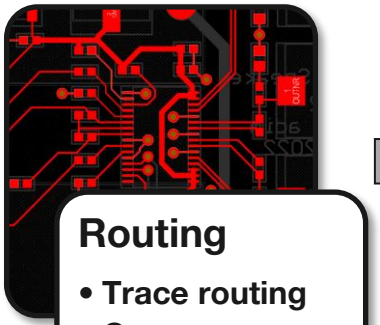
Mechanical

- Board shape
- Mechanical Features
- Stackup



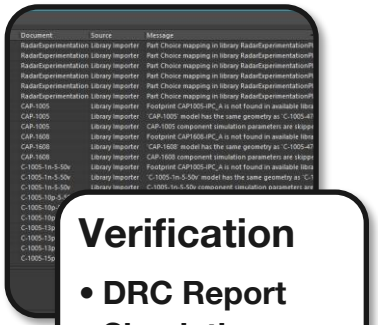
Placement

- Rooms/Blocks
- Keep-out zones
- Individual component arrangement



Routing

- Trace routing
- Copper pours



Verification

- DRC Report
- Simulation

Repeat as needed

THE ART AND SCIENCE OF PCB DESIGN

IAP 2025

Questions?