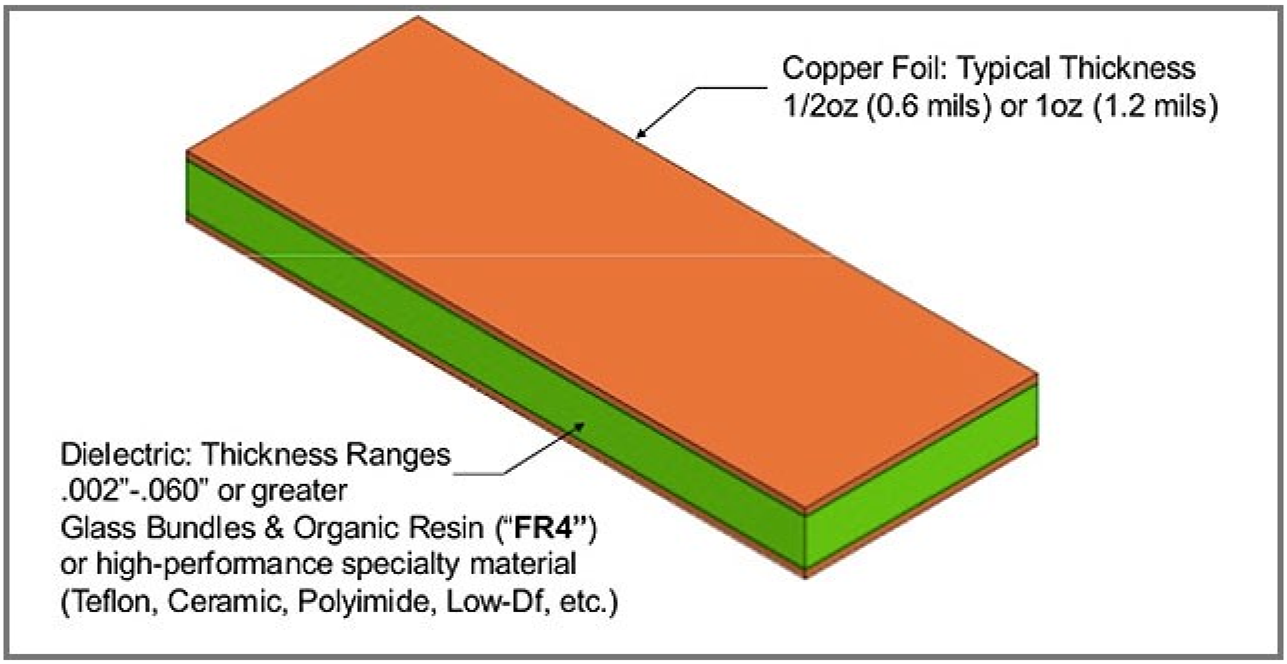
PCB Fabrication and the use of FUSION360

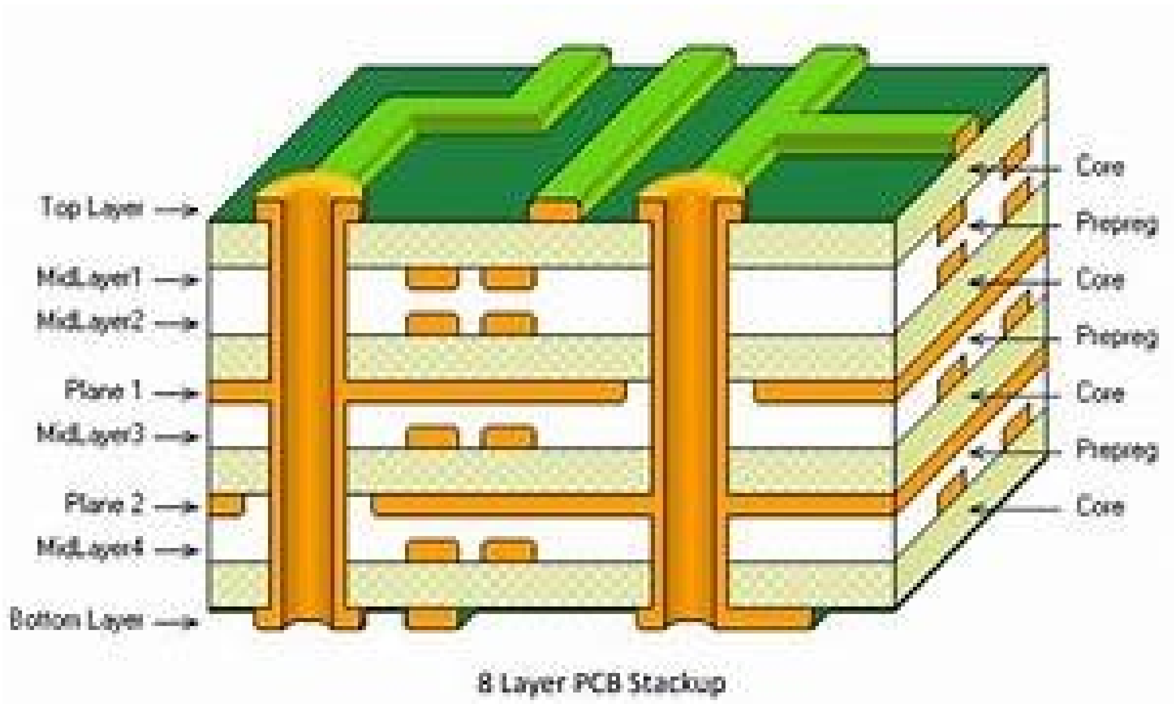
- **How to get started in the design process**
- **Terminology**
- **What are PCB's made of**? FR-4 (or FR4) is a NEMA grade designation for glass-reinforced epoxy laminate material. FR-4 is a composite material composed of **woven fiberglass cloth with an epoxy resin binder that is flame resistant** (self-extinguishing)
  FR Stands for Flame Retardant
  Grade designations for glass epoxy laminates are: G-10, G-11, FR-4, FR-5 and FR-6. Of these, ***FR-4 is the grade most widely in use today.***

PCB's are made up of layers-   Shown is the Copper Top Layer, The Dielectric, and the Copper Bottom Layer. Todays PCB's have changed from the basic 2-layer boards to boards that have four-six and as many as 12-16 layers of dielectrics and conductors.
In addition to these layers are silkscreen top, silkscreen bottom, solder mask top, solder mask bottom, and drill holes



Ex 2 layer board

Example of a PCB stack up using multilayers (8)



Ex -8 Layer board

Stackup

Stackup of a 2-layer Board using FR-4 as shown in the first image

Thickness	*Layer*	*Tolerance*
60 mil (1.5240mm)	*core*	*+/-6mil (0.1524mm)*
1.4 mil (0.0356mm)	*1 oz copper*	
0.6 mil (0.0152mm)	*solder resist*	*+/-0.2mil (0.00508mm)*
0.6 mil (0.0152mm)	*silkscreen*	*+/-0.2mil (0.00508mm)*

Material Specs

Spec Value
Substrate 175Tg FR4 King board KB6167F Datasheet
Board Thickness 63mil (1.6mm) nominal
Dielectric 4.5 at 10Mhz
Soldermask Color Purple/Green/yellow/Black/White/Red Mask Datasheet
Minimum soldermask web 4 mil (0.1016mm)
Maximum soldermask alignment 3mil (0.0762mm) Covers retraction, expansion, and shift
Silkscreen minimum line width 5 mil (0.127mm) (recommended minimum) 3 mil (0.0762mm) (short lines, text, graphics) Silkscreen Datasheet
Maximum board size 16in (406.4mm) by 22in (558.8mm)
Minimum board size 0.25in (6.35mm) by 0.25in (6.35mm)

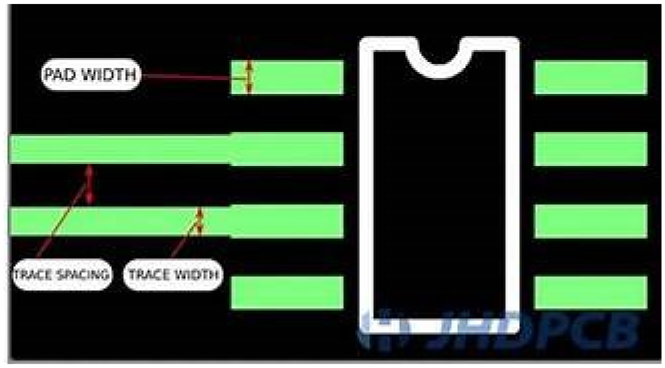
Copper Specifications

Spec Value

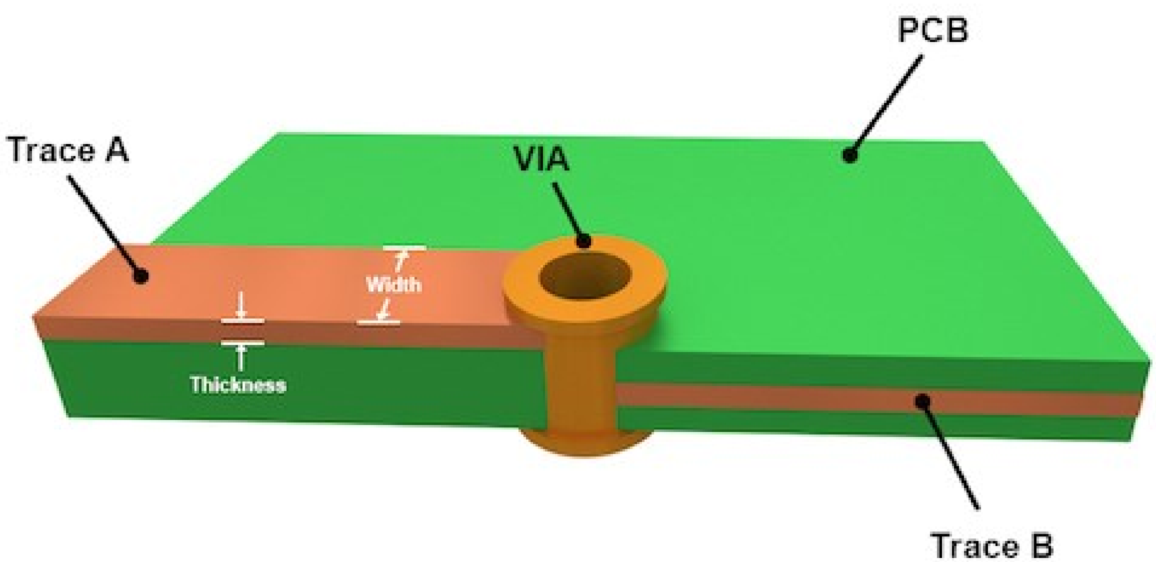
Copper Layers 2

Copper Weight 1oz

Trace Spacing 6mil (0.1524mm)



Trace Width 6mil (0.1524mm)



Annular Ring 5mil (0.127mm)

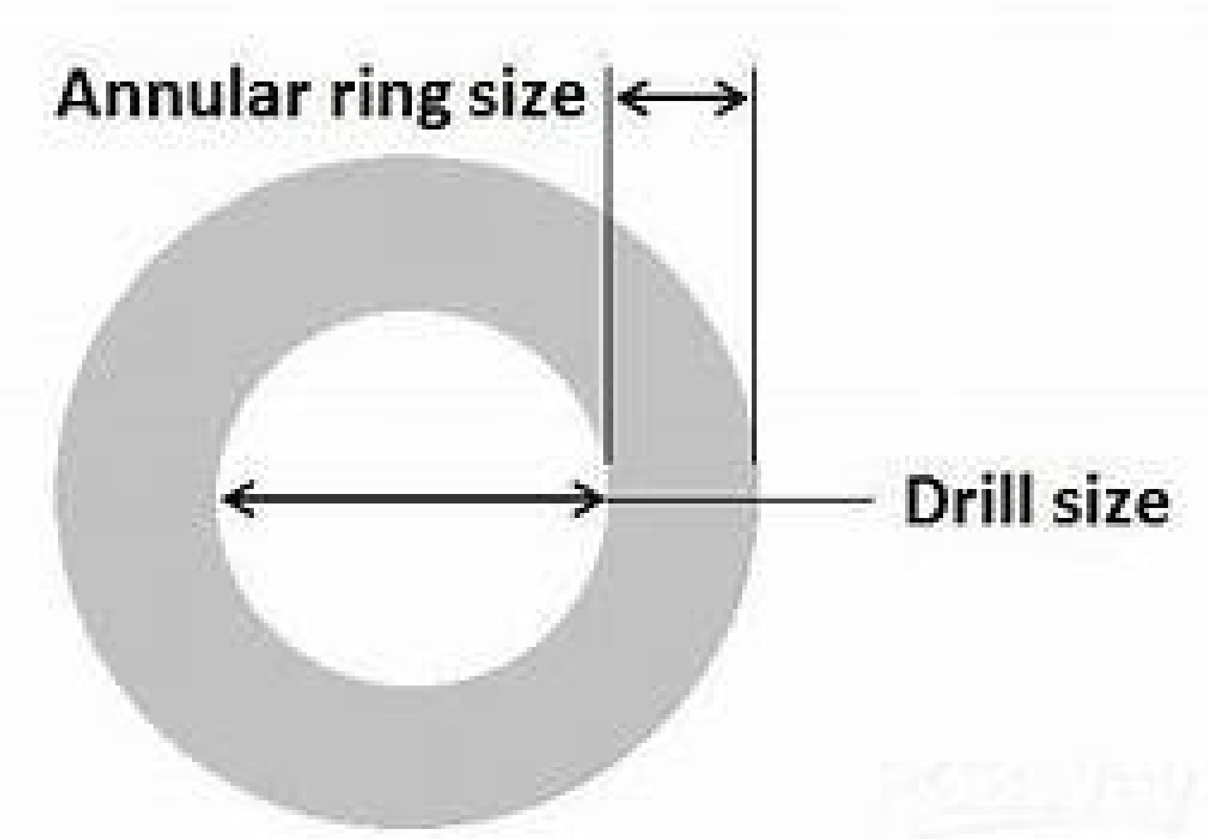
Board Edge Keep out 15mil (0.381) from nominal board edge

Via Plating Thickness 1mil (0.0254mm)

Drill Specifications

Spec **Value**

Minimum Annular Ring 5mil (0.127mm)



Maximum Drill Size **None** **Drill sizes above 250mil (6.35mm) will be fabbed, but with larger milling tolerances.**

Minimum Drill Size **10mil (0.254mm)**

Minimum Slot Size **20mil (0.508mm) (drill slot only)** **Additional information on slots**

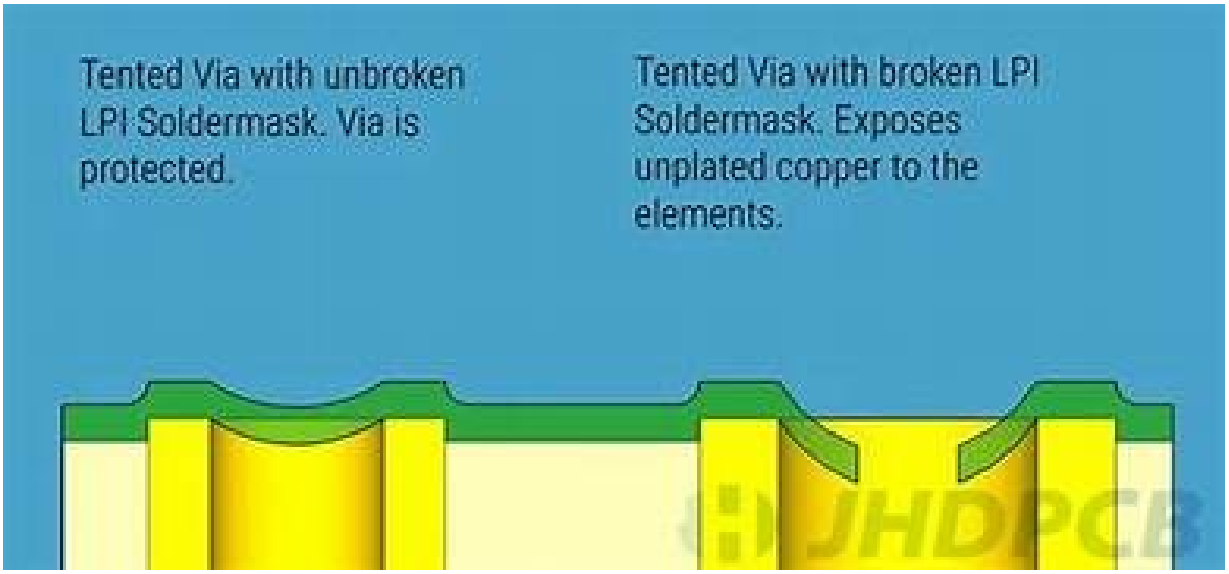
Drill Size tolerance **Max: +/- 2.5mil (0.0635mm)**

Typical: +/- 1.0mil (0.0254)

Drill Positional Tolerance **Max: 2mil (0.0508mm)**

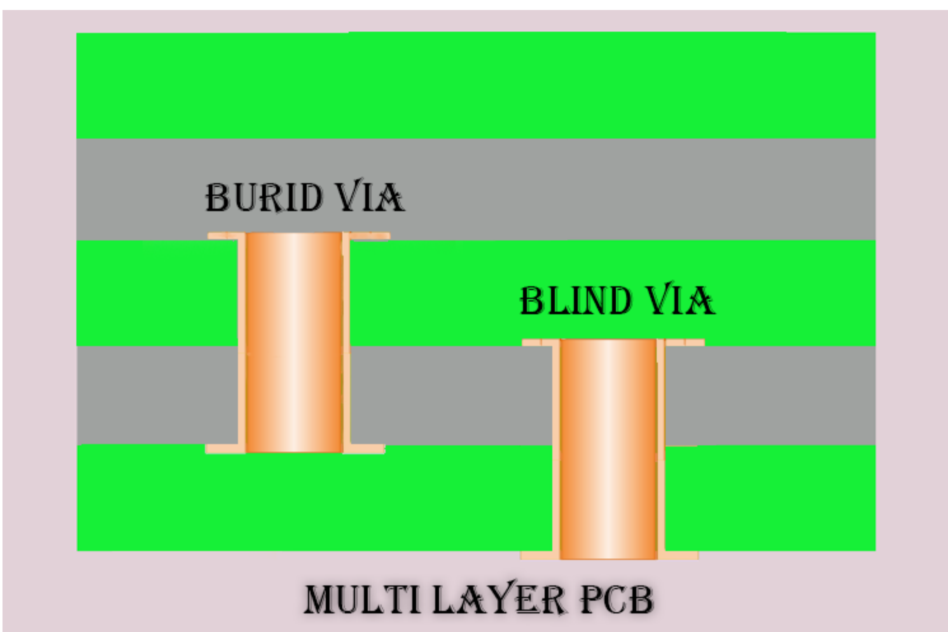
Typical: <1mil (0.0254mm)

Via Tenting **Yes (filled hole and flat surface not guaranteed) (PTH)**

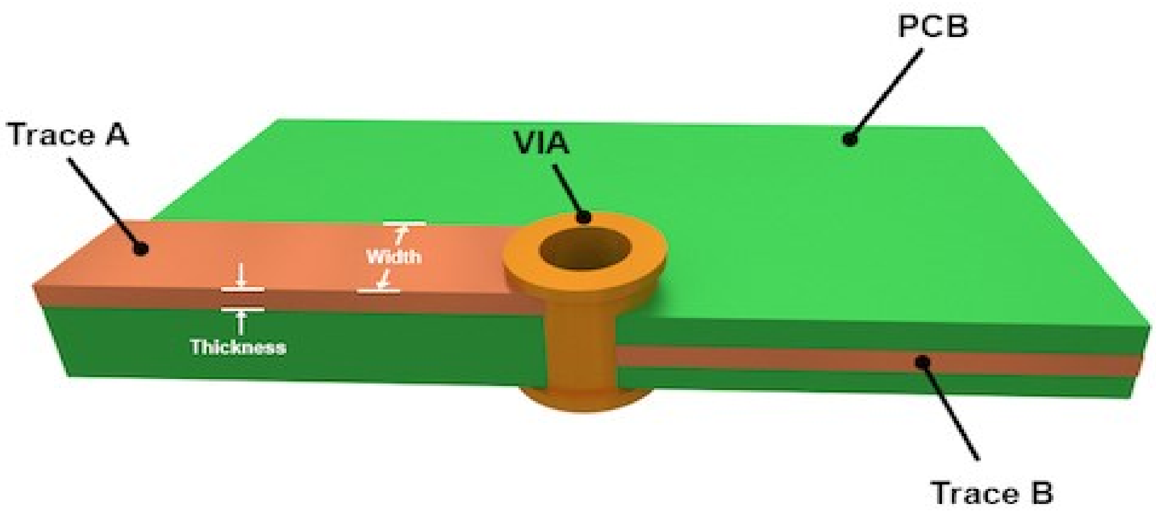


Buried Via **No** **Not used in 2-layer boards**

Blind Via **No** **Not used in 2-layer boards**



TRACE WIDTH

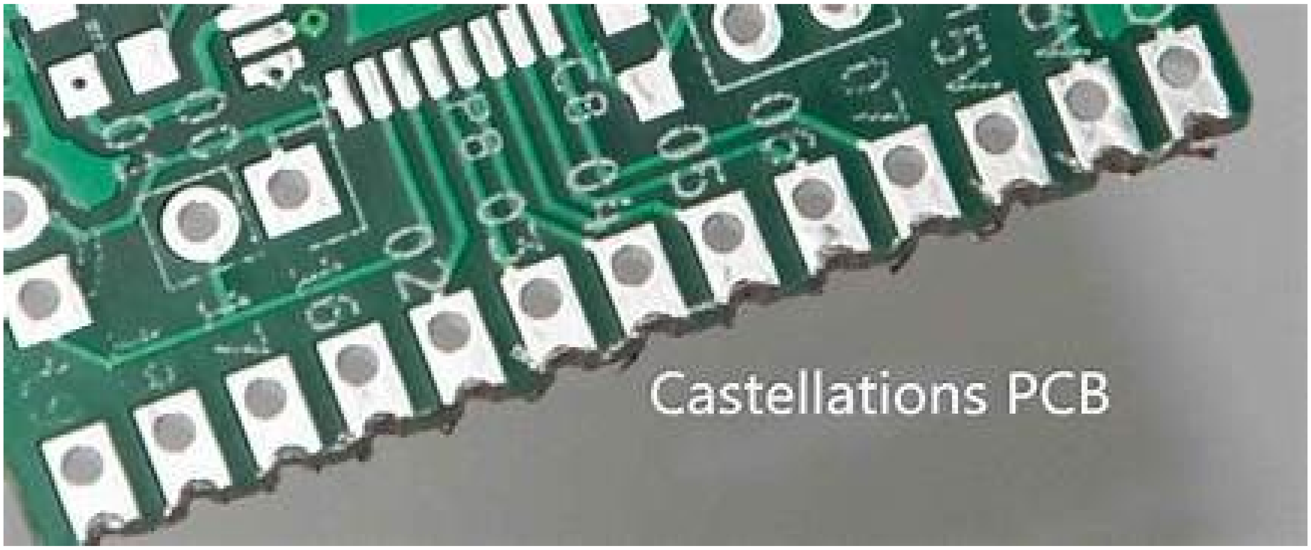


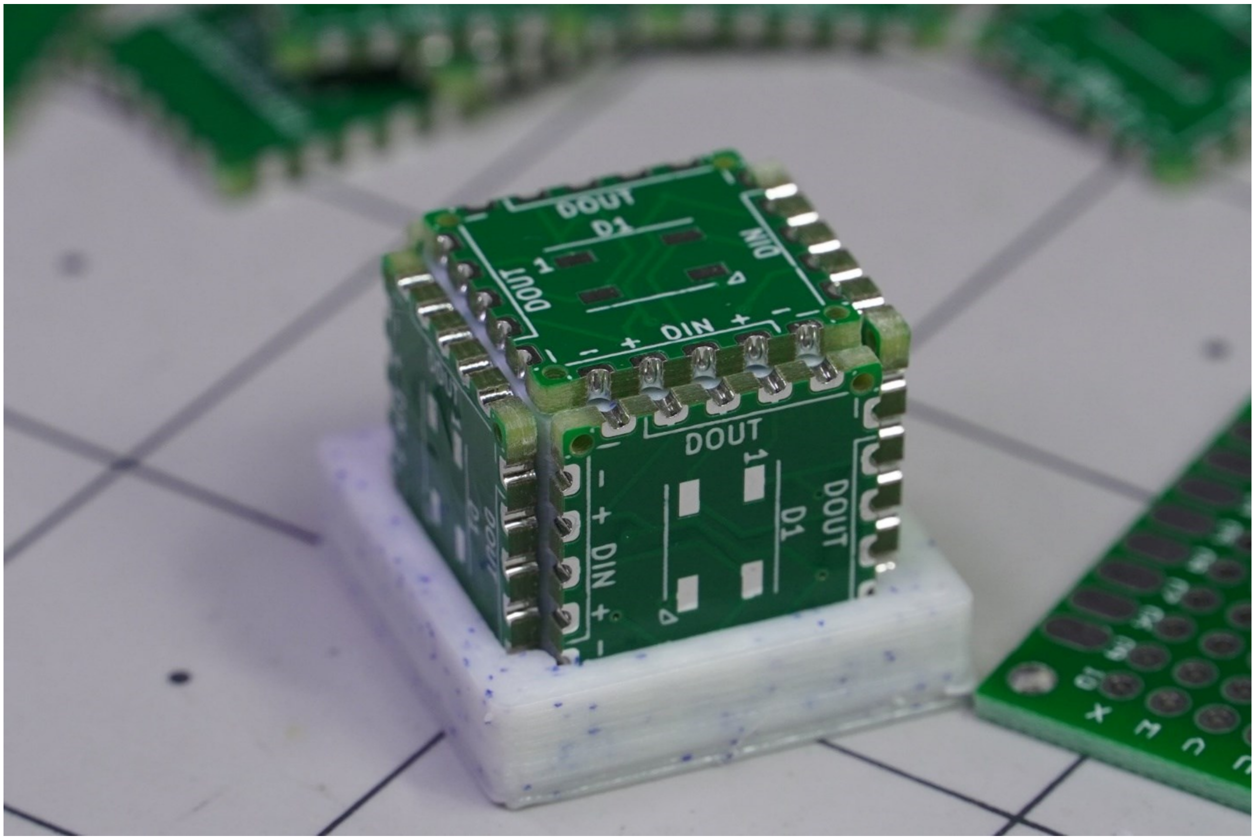
Filled Plated Vias (via-in-pad) **No**

Overlapping drills **Allowed, but not guaranteed. May result in missing or slotted holes.**

5 mil (0.127mm) clearance is recommended between holes.

Castellations **Allowed, but not guaranteed** **Details and recommendations**





Maximum Drill Size None Drill sizes above 250mil (6.35mm) will be fabbed, but with larger milling tolerances.

Terminology

JLCPCB is a good resource for pcb fabricators. To help answer any questions on terminology this is a good place to start.

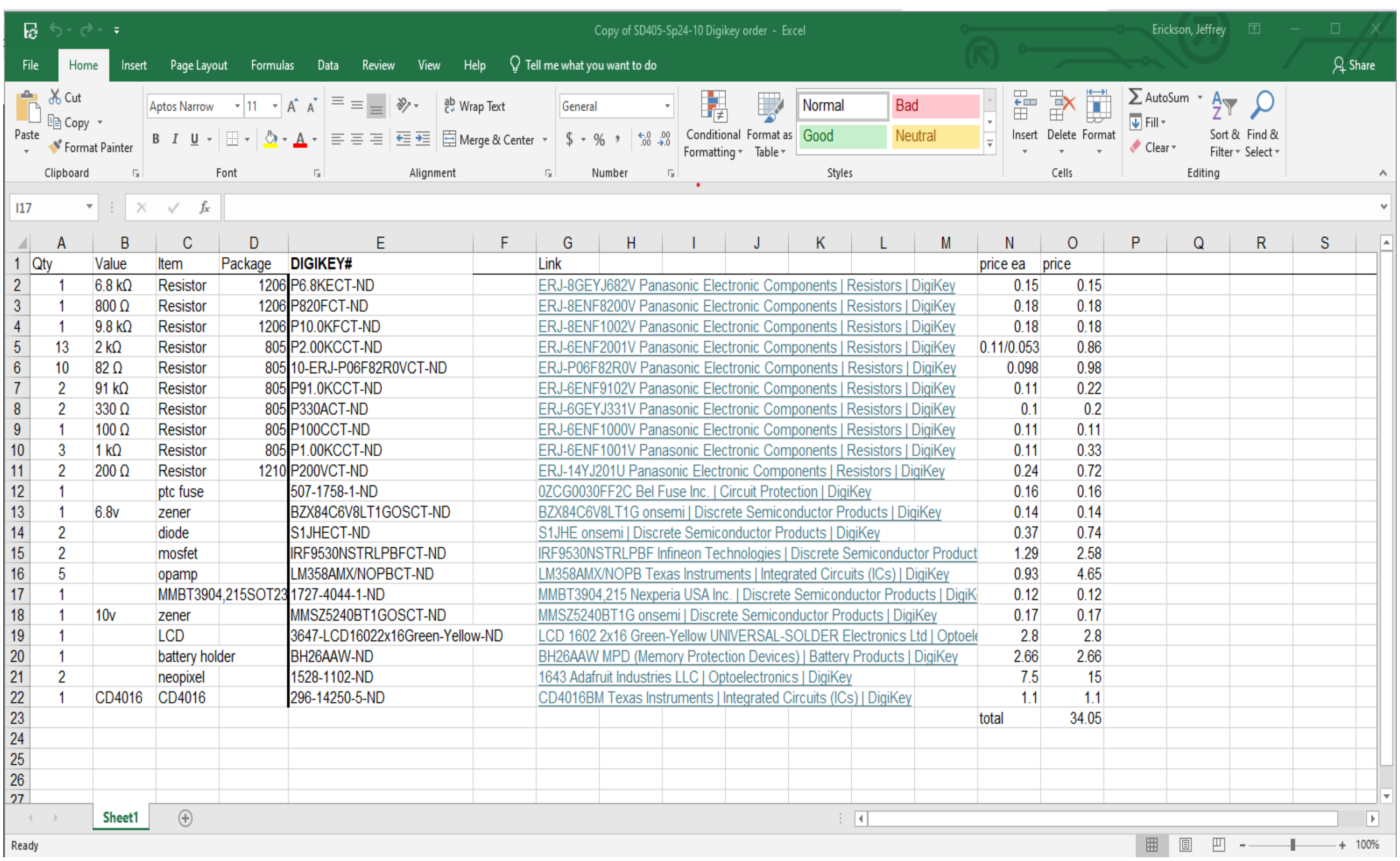
https://jlcpcb.com/capabilities/pcb-capabilities

https://www.bisonacademy.com/ECE405/Lectures/02%20Fusion360%20Slides%20plus%20notes.pdf

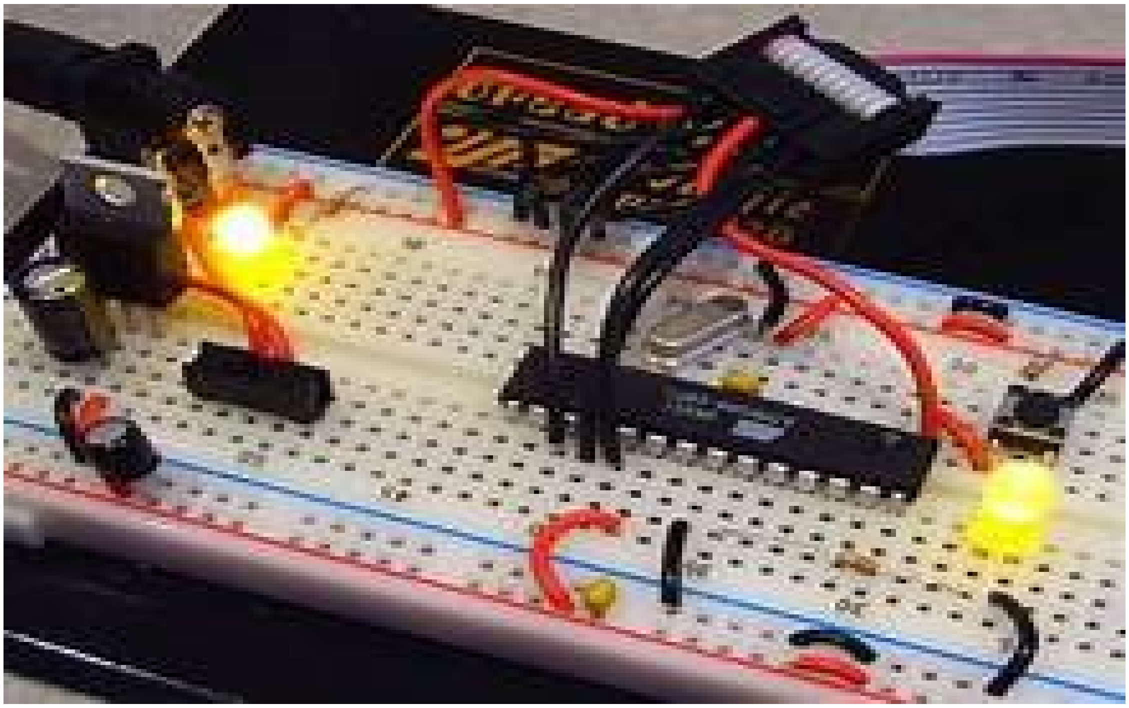
4 stages to PCB development

1. **Create a parts list from Breadboard Design**
2. **Create a schematic**
3. **PCB Design**
4. **Create the Gerber files**

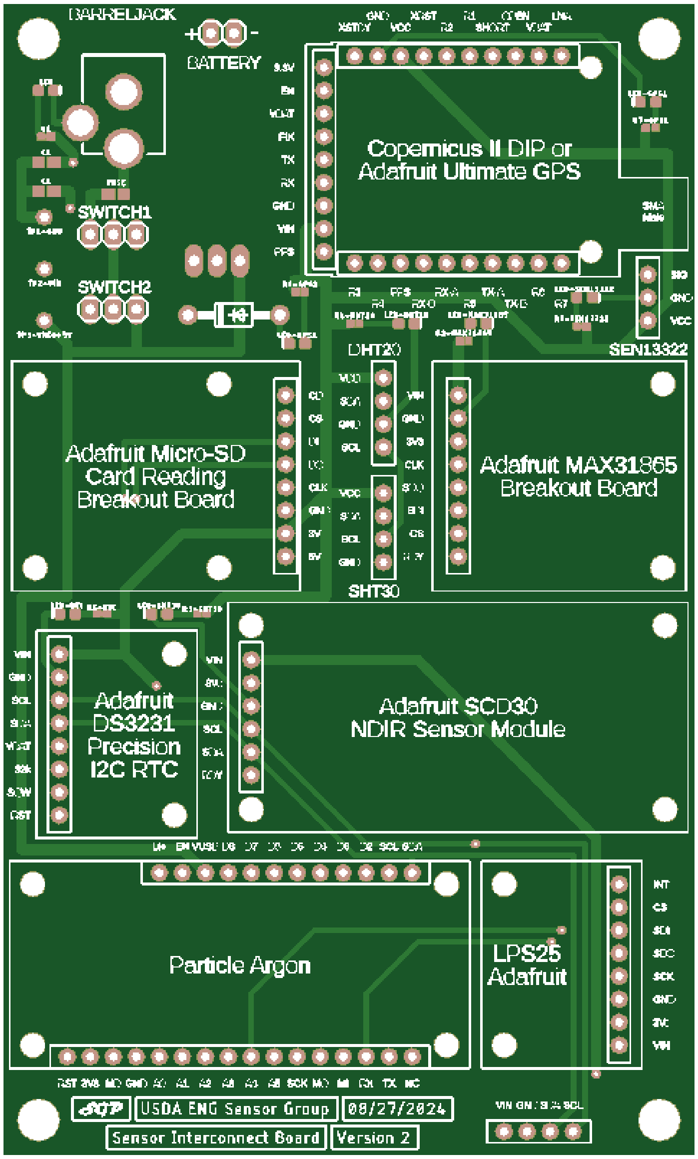
1) Start a parts list PARTS LIST from FUSION360



Qty	Value	Item	Package	DIGIKEY#	Link	price ea	price
1	6.8 kΩ	Resistor	1206	P6.8KECT-ND	ERJ-8GEYJ682V Panasonic Electronic Components | Resistors | DigiKey	0.15	0.15
1	800 Ω	Resistor	1206	P820FCT-ND	ERJ-8ENF8200V Panasonic Electronic Components | Resistors | DigiKey	0.18	0.18
1	9.8 kΩ	Resistor	1206	P10.0KFCT-ND	ERJ-8ENF1002V Panasonic Electronic Components | Resistors | DigiKey	0.18	0.18
13	2 kΩ	Resistor	805	P2.00KCCT-ND	ERJ-6ENF2001V Panasonic Electronic Components | Resistors | DigiKey	0.11/0.053	0.86
10	82 Ω	Resistor	805	10-ERJ-P06F82R0VCT-ND	ERJ-P06F82R0V Panasonic Electronic Components | Resistors | DigiKey	0.098	0.98
2	91 kΩ	Resistor	805	P91.0KCCT-ND	ERJ-6ENF9102V Panasonic Electronic Components | Resistors | DigiKey	0.11	0.22
2	330 Ω	Resistor	805	P330ACT-ND	ERJ-6GEYJ331V Panasonic Electronic Components | Resistors | DigiKey	0.1	0.2
1	100 Ω	Resistor	805	P100CCT-ND	ERJ-6ENF1000V Panasonic Electronic Components | Resistors | DigiKey	0.11	0.11
3	1 kΩ	Resistor	805	P1.00KCCT-ND	ERJ-6ENF1001V Panasonic Electronic Components | Resistors | DigiKey	0.11	0.33
2	200 Ω	Resistor	1210	P200VCT-ND	ERJ-14YJ201U Panasonic Electronic Components | Resistors | DigiKey	0.24	0.72
1		ptc fuse		507-1758-1-ND	0ZCG0030FF2C Bel Fuse Inc. | Circuit Protection | DigiKey	0.16	0.16
1	6.8v	zener		BZX84C6V8LT1GOSCT-ND	BZX84C6V8LT1G onsemi | Discrete Semiconductor Products | DigiKey	0.14	0.14
2		diode		S1JHECT-ND	S1JHE onsemi | Discrete Semiconductor Products | DigiKey	0.37	0.74
2		mosfet		IRF9530NSTRLPBFCT-ND	IRF9530NSTRLPBF Infineon Technologies | Discrete Semiconductor Product	1.29	2.58
5		opamp		LM358AMX/NOPBCT-ND	LM358AMX/NOPB Texas Instruments | Integrated Circuits (ICs) | DigiKey	0.93	4.65
1		MMBT3904,215SOT23		1727-4044-1-ND	MMBT3904,215 Nexperia USA Inc. | Discrete Semiconductor Products | DigiK	0.12	0.12
1	10v	zener		MMSZ5240BT1GOSCT-ND	MMSZ5240BT1G onsemi | Discrete Semiconductor Products | DigiKey	0.17	0.17
1		LCD		3647-LCD16022x16Green-Yellow-ND	LCD 1602 2x16 Green-Yellow UNIVERSAL-SOLDER Electronics Ltd | Optoel	2.8	2.8
1		battery holder		BH26AAW-ND	BH26AAW MPD (Memory Protection Devices) | Battery Products | DigiKey	2.66	2.66
2		neopixel		1528-1102-ND	1643 Adafruit Industries LLC | Optoelectronics | DigiKey	7.5	15
1	CD4016	CD4016		296-14250-5-ND	CD4016BM Texas Instruments | Integrated Circuits (ICs) | DigiKey	1.1	1.1
						total	34.05



2) Using your working Breadboarded circuit create a schematic

3) 

4) Create the GERBER FILES

Look through and Use this getting started document

https://www.bisonacademy.com/ECE405/Lectures/02%20Fusion360%20Slides%20plus%20notes.pdf

PCB Final Design Requirements

- ✓ Create a detailed schematic using Fusion360- Print the Schematic Design >use the Document View> Library>Print> Print the pdf
  Update OneNote-
  Section: HWK4
  Page: Fusion360 Schematic

- ✓ Create a PCB Design using Fusion360- Print the pdf design of the PCB *using the Document tab>Outputs>Print pdf*
  Update OneNote-
  Section: HWK4,
  Page: Fusion 360 PCB

- ✓ Print the Bill of Materials (Parts) list using *the Document tab>Library >Output> (next to printer Icon) Bill of Materials tabs in Fusion 360 Schematic Editor and install in OneNote*
  Update One Note-
  Section: HWK4
  Page: Bill of Materials

- ✓ Save the Gerber file from the PCB Design view using the Manufacturing Tab> under Manufacturing Drop down list>Export Gerber's or highlight the Export Gerber Icon> Rename the file to include your Group Designator
  Update One Note-
  Section: HWK4
  Page: Gerber's

When Creating PCB's these items are to be included for grading requirements.

❖ Your PCB Requirements

- 2.000" x 3.000" (ECE401 only) actual size is whatever you require in ECE405.
- Mounting holes 175 mils in each corner (Diameter)
- Power & Ground Traces: 40mils (required ECE401 only)
- All Other Traces: 20mils (Required ECE401 only)
- Ground plane on the bottom side of PCB
- Silk-Screen designators in correct order
- Board must show the project name /Board name and Vs.#/ Date
- (Font15 recommended for font size)
- If your proud of your work- include your name or Initial it.
- Test points should be available for measurement; example {9V, 5V, ground, Input, Output, and Collector(s)}

- ✓ Save the Gerber's and send via email for order processing to jeffrey.erickson@ndsu.edu.
- ✓ Recommend first sending your files to a Gerber viewer. They are free on the net. If you send your files to JLCPCB.com, there is a Gerber viewer. By layer and by 3d imaging.

Helpful Hints:

Grid settings: Lower Middle **OBJECT Properties Bar**>Grid settings> Size: 100mil, display on/off, style- Dots/Lines-------Suggest leaving in 100mils or 2.54mm spacing, (many parts do snap better in 50mils)

Board Size: How to change board size: In PCB Document View Highlight outer edge of black drawing surface (using the mouse highlight the outer edge of the Design box, change dimensions as needed

Vias: placing vias in PCB Design View >Route>VIA> select and place via or **while Manual routing stop where required and press the spacebar- 2nd press will change top to bottom trace**

Holes: creating mounting holes: in **PCB Design View>PLACE> Hole (NPTH)** Non-plated through hole- or it will short top layer to bottom layer

Top and Bottom traces: *switching from top and bottom traces-**change selection from Layers tab***

Track width: Changing width/size---- **Highlight the track**>Right click>Properties>Change Width size

Silkscreen: Adding Names/ board names >In PCB Design View >Document>Draw>TEXT> Under TEXT Properties page, type in the text> change the Font size> Must *be in correct Layer (21 Silkscreen Top or 22 Silkscreen Bottom)*

Changing Parts> This should be done from the schematic editor

- ✓ Auto Routing >Quick Route>Auto Router>Continue>Start> **Choose Variant that you like**>End Job

Ground Plane>PCB Design Editor> **Change Layer to Bottom** Layer in layers tab> then--Polygon> Polygon Pour > enter GND in the POLYGON POUR Editor page- then Done. Use Mouse to highlight the outer box of your PCB Design Note to edit it Highlight the images and delete Polygon Pour