PANTECH SOLUTIONS
Technology Beyond the Dreams

PCB Design Workshop

OrCAD™
CADENCE PCB SOLUTIONS

www.pantechsolutions.net

*Disclaimer -Pantech is not associated with Orcad all the Logo are owned by the respective owners.

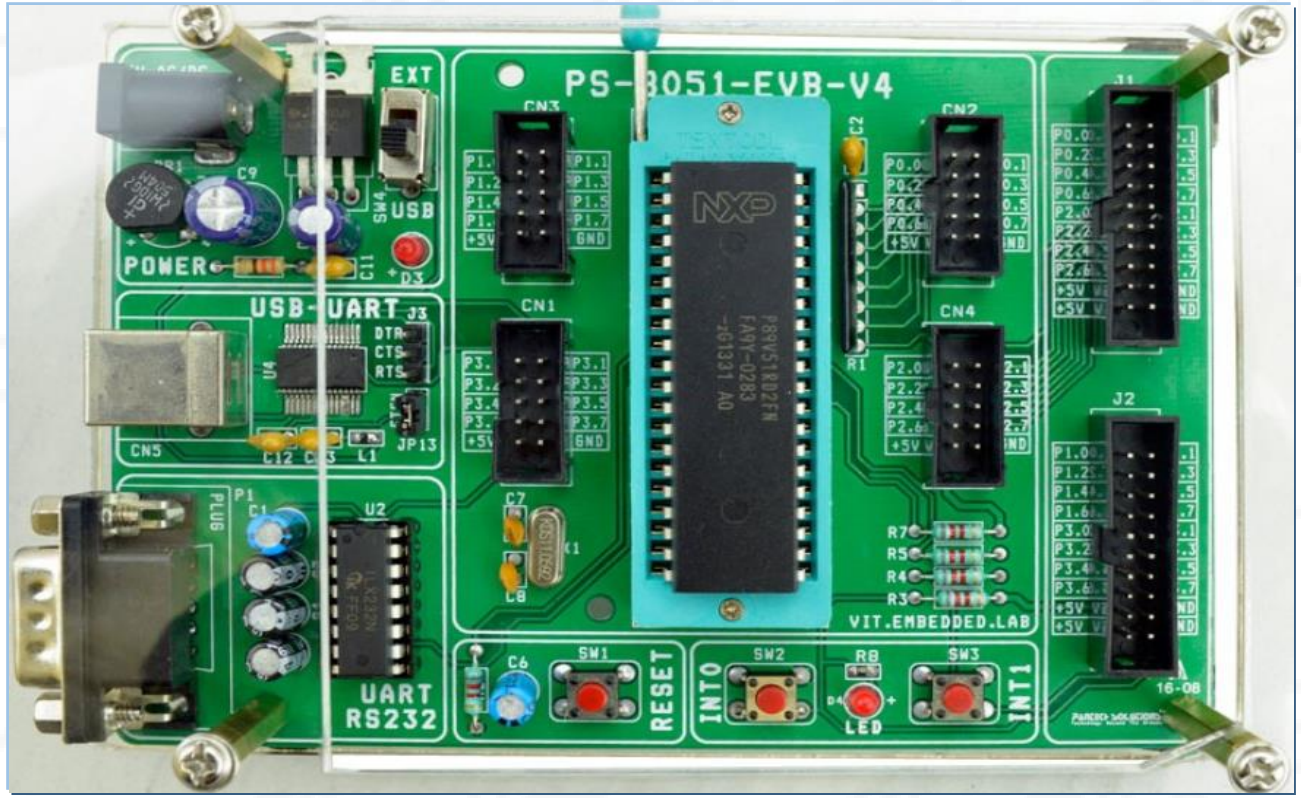
What you will Learn? -Week 1

- ✓ Day 1- Introduction to PCB Design and Terminologies and Installation of Orcad Trail version
- ✓ Day 2 -Introduction to Schematic Capture(Creating a simple project)
- ✓ Day 3- Introduction to Allegro and Foot print Creation (Creating a simple project)
- ✓ Day 4- Importing Schematics in allegro ,Placement and route (Creating a simple project)
- ✓ Day 5- Gerber Creation, BOM,PDF (Creating a simple project)
- ✓ Day 6- How to Design a 8051 Microcontroller Board

What you will Learn? -Week 2

- ✓ Day 7- Library Creation-8051 Board
- ✓ Day 8- Schematics Design-8051 Board
- ✓ Day 9- Footprint Creation -8051 Board
- ✓ Day 10- Design rules check-Import and Placement
- ✓ Day 11- Layout
- ✓ Day 12- Layout Design ,Gerber Creation, Recap, schematic design consideration,Layout Design Consideration

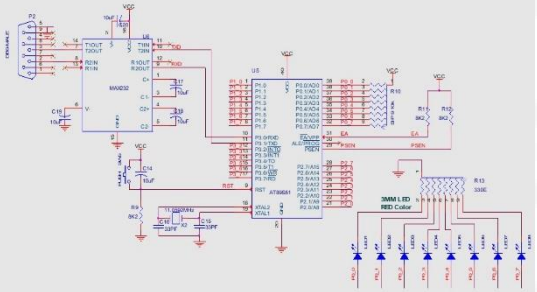
8051 Board Design



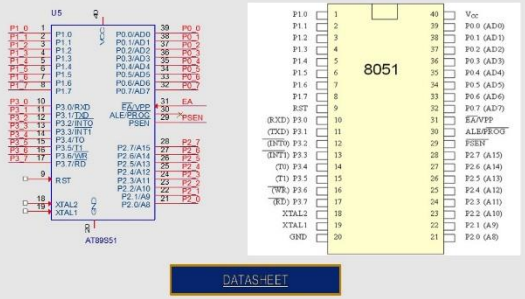
Key Features of 8051 Project Board

- Supports (Atmel | NXP | Dallas) MCU (SST89E516RD)
- 2 No's of Tactile switch for interrupts study
- ISP Programming for NXP and Dallas MCU using USB
- 4 No's of 10-Pin Expansion Connector
- 4 No's of 20-Pin Expansion Connector
- On-Board 5V Regulator @ 1A and 3.3v @800mA
- Reset Circuit
- Power-on LED Indication
- 40 pin ZIF socket for Microcontroller
- UART

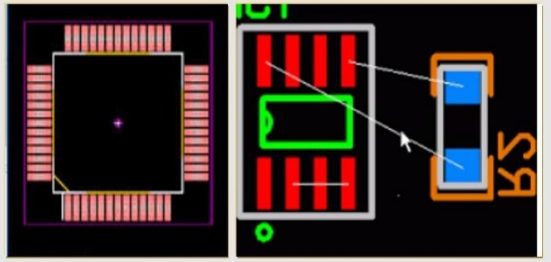
HOW TO MAKE SCHEMATICS



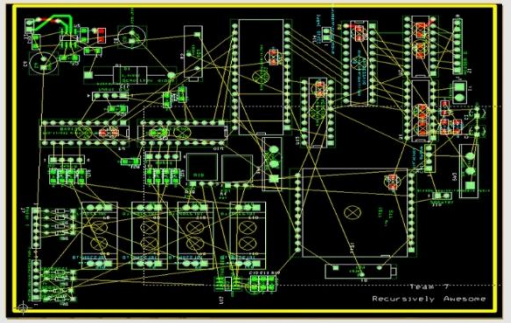
How to Create schematic Library



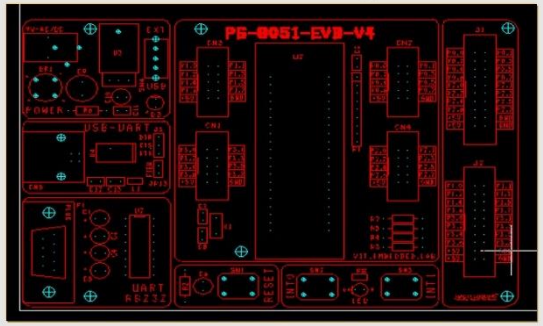
How to Make PCB Foot prints



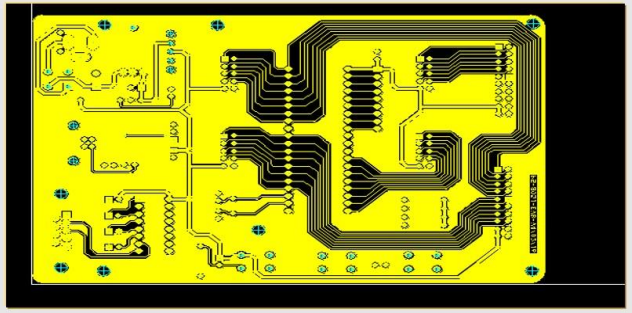
How to Place Components



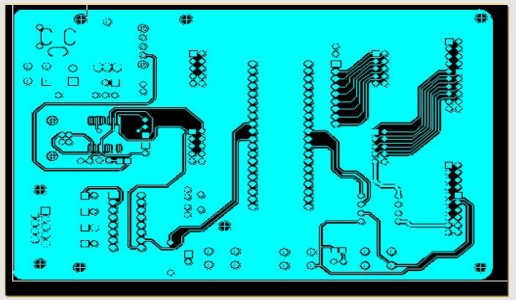
How to Make Legend



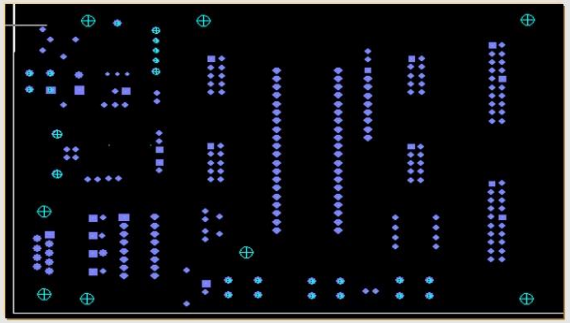
Bottom layer -Routing



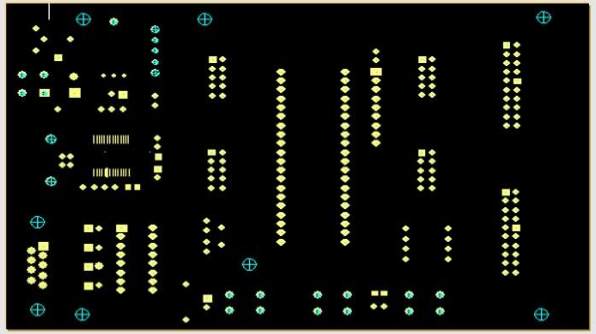
Top Layer-Routing



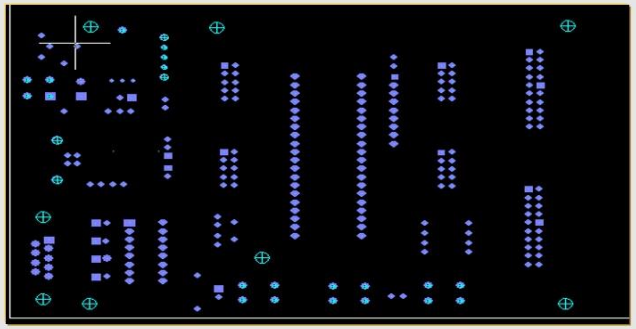
How to add Drill chart



Solder Mask -Top



Solder Mask -Bottom



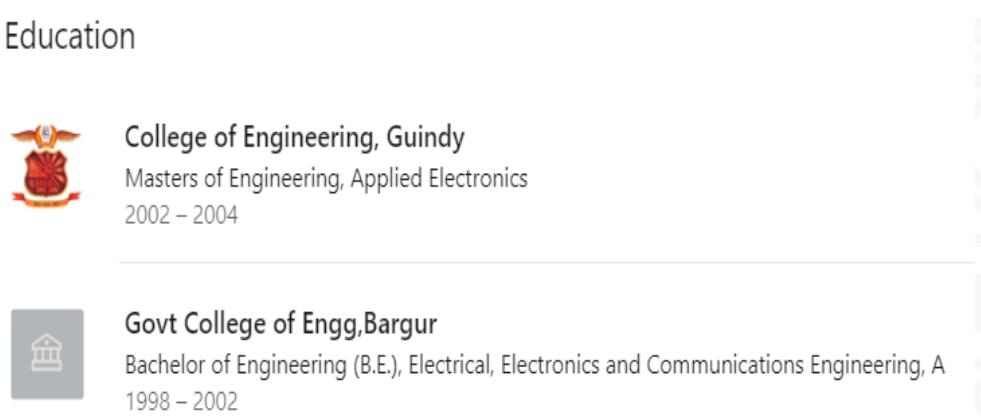
How to Design 8051 Board

About Pantech

- ✓Started in the Year 2004
- ✓Lab equipments and Sensor Interface
- ✓Manufacturer of Brainsense EEG Headset
- ✓Reconfigurable Algorithms on AI
- ✓Manufacture of AI development Boards
- ✓Power electronics, Fuel cell and Renewable Energy trainers

www.pantechsolutions.net

About me



https://www.linkedin.com/in/jeevarajan/

My Primary Expertise

Microcontroller Architecture: 8051,PIC,AVR,ARM,MSP430,PSOC3

DSP Architecture: Blackfin,C2000,C6000,21065L Sharc

FPGA: Spartan,Virtex,Cyclone

Image Processing Algorithms: Image/Scene Recognition, Machine Learning, Computer Vision, Deep Learning, Pattern Recognition, Object Classification ,Image Retrieval, Image enhancement and denoising.

Neural Networks : SVM,RBF,BPN

Cryptography :RSA,DES,3DES,Ellipti curve,Blowfish,Diffe Hellman

Compilers: Keil,Visual DSP++,CCS, Xilinx Platform studio,ISE, Matlab, Open CV

Announcement

- Attendance Link at 9 pm
- Minimum attendance required for an Free-E-Certificate is 12 Days +Project submission(Gerber or Schematics). Attendance link will be valid for 1 hrs. after the event.
- For Internship Candidates no attendance required ,it will be accessed from the LMS Portal. (learn.pantechsolutions.net)
- Recorded Video Streaming for LAB classes to improve Learning Experience

- https://t.me/joinchat/PkU4n8P3E05iNzhl

Certificate of Internship at Rs 300

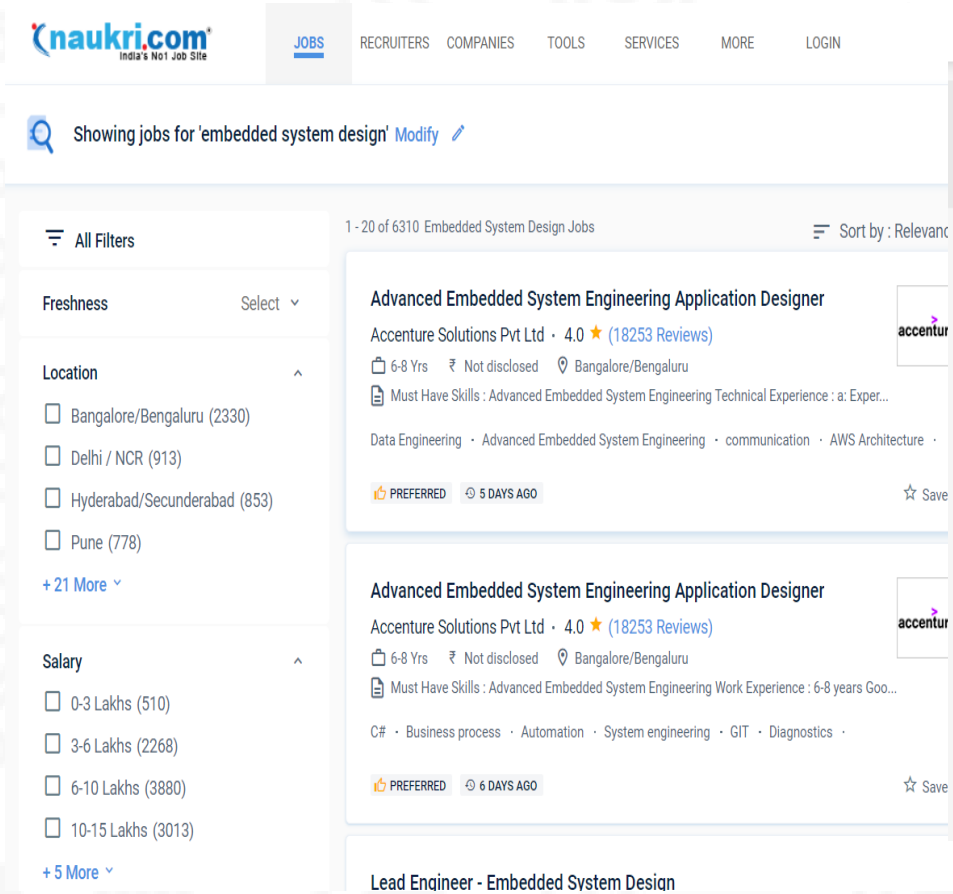
- What You get
  - Recorded Video access for 6 Months
  - Free Hackathon session on ZOOM for 2 Hrs(Create a simple 2 layer pcb)
    - Prerequisites(Have to installed Orcad before the event)

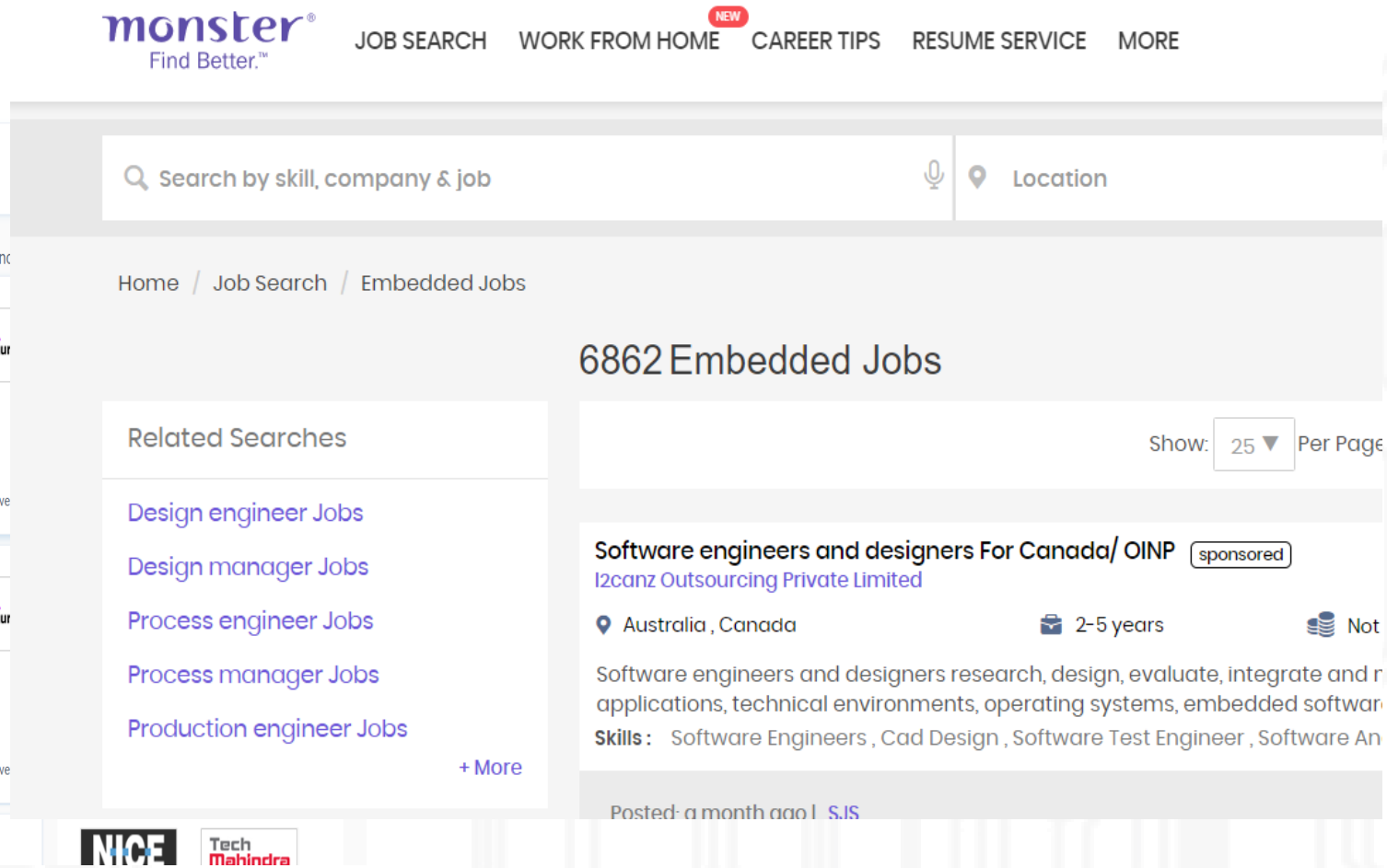
Combo Offer

Combo 1 - PCB Design + Embedded System Design&Iot + FPGA =999 Rs(6 Months Validity)

Combo 2 - PCB Design + Embedded System Design&Iot + Arduino =999 Rs(6 Months Validity)

Job Opportunities in Embedded System design





PCB Design Challenge-12 Days

SKILL SET + MINDSET

Mindset Lesson

- ✓ Have a Definite Goal
- ✓ Pursue Your goal with a Positive Attitude

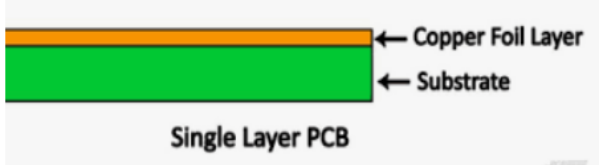
“The Strangest Secret in the world-Earl Nightingale”

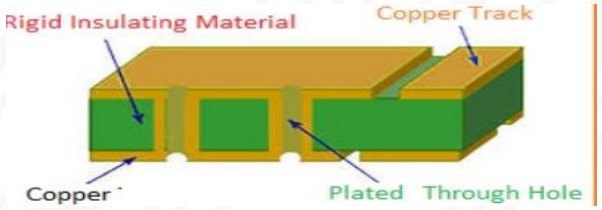
What is PCB

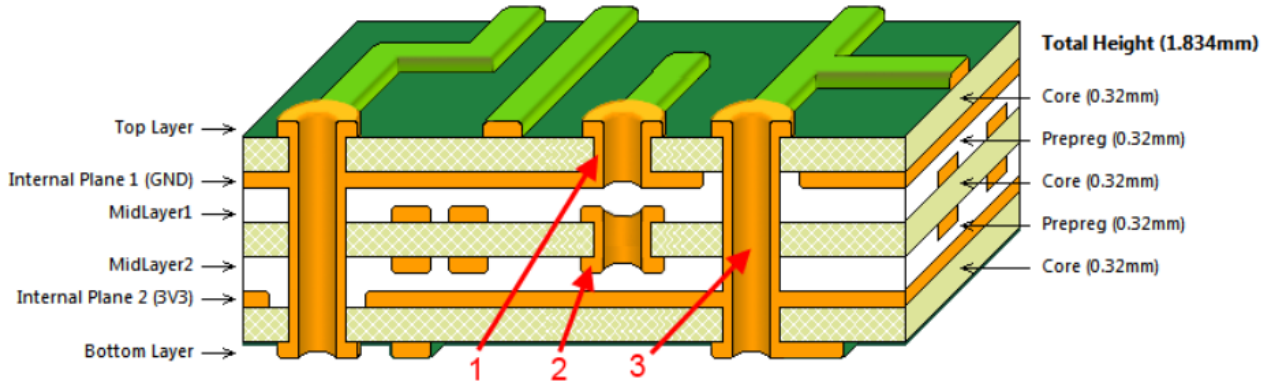
- A **printed circuit board** (**PCB**) mechanically supports and electrically connects electronic components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate.
- Components are generally soldered onto the PCB to both electrically connect and mechanically fasten them to it

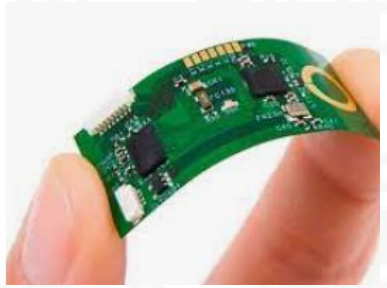
Types of PCB'S

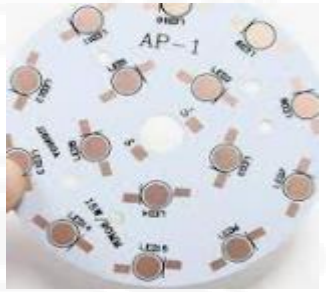
- ✓Single Layer PCB
- ✓Double Layer PCB
- ✓Multi Layer PCB
- ✓RIGID PCB
- ✓Flexible PCB
- ✓High frequency PCB
- ✓Aluminum Packed PCB

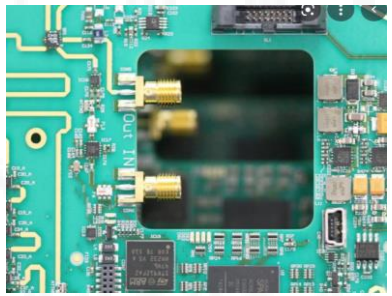












PCB LAYERS

- **Substrate Layer**

  The substrate layer of any PCB is usually made from fibreglass, which gives the board its rigid form. Generally speaking, the majority of boards will have this material for their substrate, with the exception of flexible PCBs, which are built on flexible plastic such as Kapton. Substrate PCB layers can also be made with other materials such as epoxies, however they lack the durability that FR4 (found in fibreglass) provides for a high quality board.

- **Copper Layer**

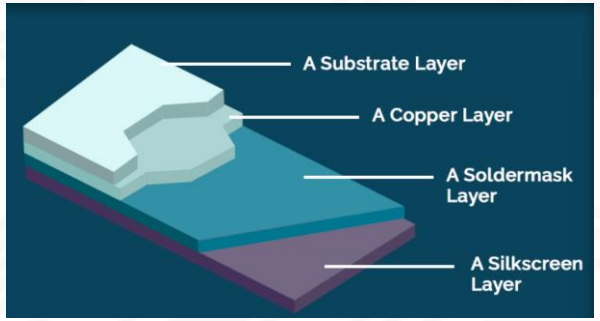
  The thickness of the copper layer on your PCB will depend on the power the PCB needs to withstand.

- **Solder mask layer**

Once the copper layer has been applied the solder mask layer is placed on top. This gives the PCB its green color and is used to insulate the copper layer to avoid any contact with any other metal or elements of the board that could disrupt the copper traces.

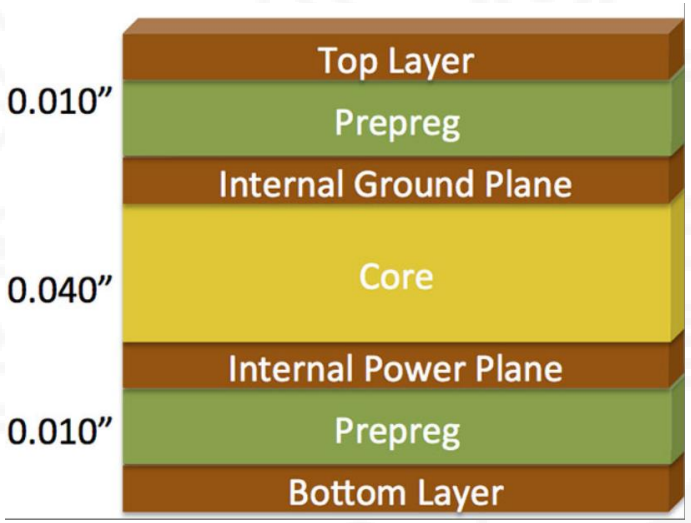
- **Silkscreen layer**

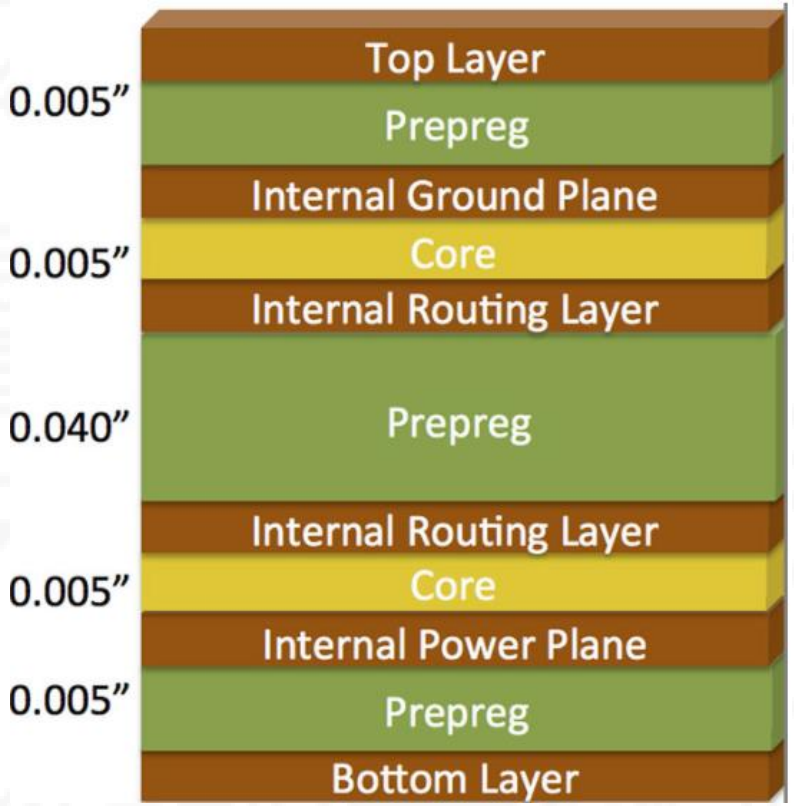
The silkscreen is mainly for the purpose of us humans to help us better understand the board and the functionality of different pins or LEDs, by adding letters, numbers and symbols to the board.

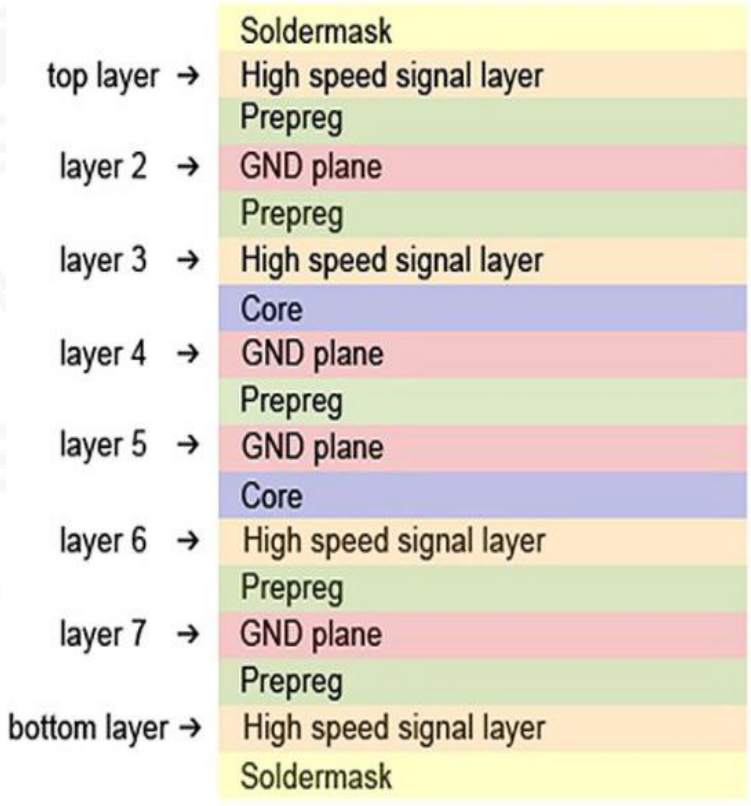


https://www.pcbpower.com/blog-detail/Printed-Circuit-Board-Layers

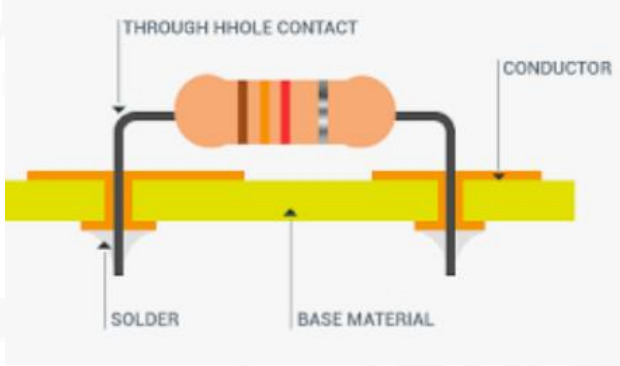
Multilayer PCB's

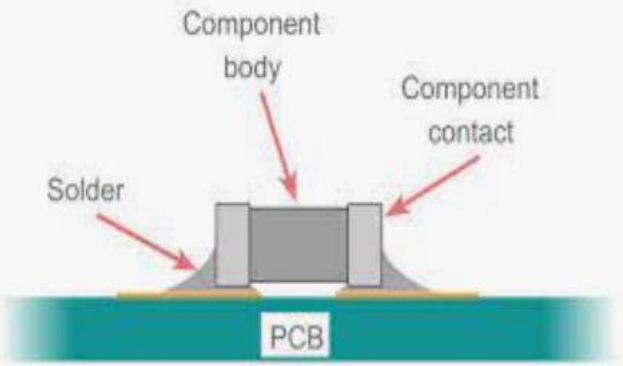




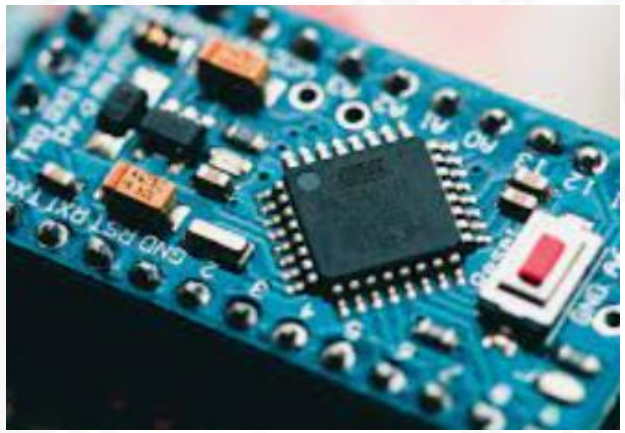


Characteristics of PCB





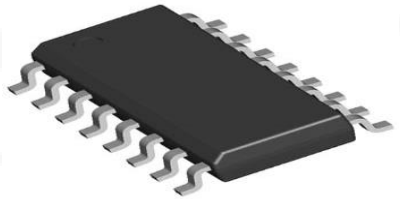
- **Through-hole technology**
- **Surface Mount Technology**
- **Circuit properties of the PCB**
  - Power and ground traces may need to be wider than signal traces.
  - In a multi-layer board one entire layer may be mostly solid copper to act as a ground plane for shielding and power return
- **Materials**
  - **FR-4**
  - **Flexible**
  - **Aluminium**
  - **RoHS compliant PCB**
- **Copper thickness**



Selection of packages

DIP

QFP

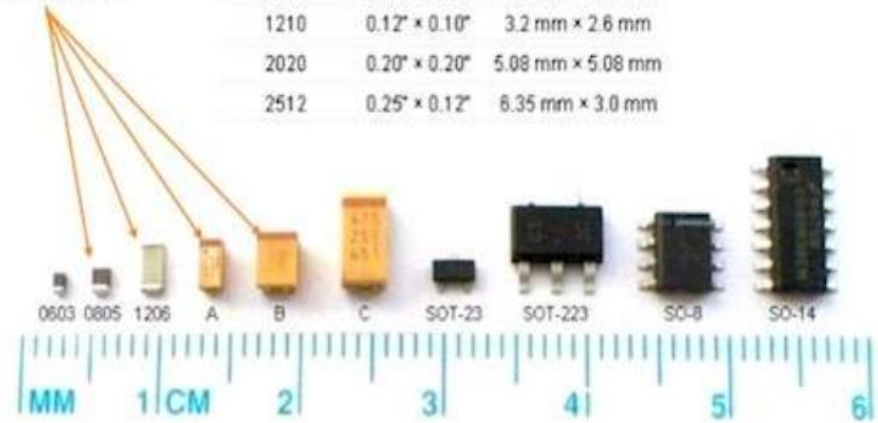


SOIC

PLCC

BGA

Package type	Size in inches	Size in mm
0201	0.024" × 0.012"	0.6 mm × 0.3 mm
0402	0.04" × 0.02"	1.0 mm × 0.5 mm
0603	0.063" × 0.031"	1.6 mm × 0.8 mm
0805	0.08" × 0.05"	2.0 mm × 1.25 mm
1206	0.126" × 0.063"	3.2 mm × 1.6 mm
1210	0.12" × 0.10"	3.2 mm × 2.6 mm
2020	0.20" × 0.20"	5.08 mm × 5.08 mm
2512	0.25" × 0.12"	6.35 mm × 3.0 mm

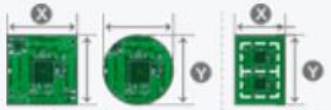


PCB Specification Selection

Board type : Single pieces | Panel by Customer | Panel by PCBWay

Different Design in Panel : 1 | 2 | 3 | 4 | 5 | 6 | e.g.

* Size (single): Length X Width mm inch'↔mm



* Quantity (single): pcs

Layers: 1 Layer | 2 Layers | 4 Layers | 6 Layers | 8 Layers | 10 Layers | 12 Layers | 14 Layers

Material: FR-4 | Aluminum | Rogers | HDI(Buried/blind vias) ≥4 Layers | Copper Base

*Material model can be remarked below. HDI is available for 4-layer or more.

FR4-TG: TG 130-140 | TG 150-160 | TG 170-180

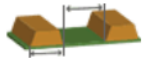
Thickness: 0.2 | 0.4 | 0.6 | 0.8 | 1.0 | 1.2 | 1.6 | 2.0 | 2.4 | 2.6 | 2.8 | 3.0 | 3.2

≥1.7-6.0 * Unit: mm

https://www.pcbway.com/

PCB Specification Selection

Min Track/Spacing: 3/3mil | 4/4mil | 5/5mil | **6/6mil** | 8/8mil ↑



Min Hole Size: 0.15mm | 0.2mm | 0.25mm | **0.3mm ↑** | 0.8mm ↑ | 1.0mm ↑ | No Drill

Solder Mask: **Green** | Red | Yellow | Blue | White | Black | Purple | Matte black | Matte green | None

Silkscreen: **White** | Black | None

Edge connector: Yes | **No**

Surface Finish: **HASL with lead** | HASL lead free | Immersion gold(ENIG) | OSP | Hard gold | Immersion silver(Ag) | ENEPIG | None(Plain copper)

Via Process : **Tenting vias** | Plugged vias | Vias not covered

*For Gerber files, this choice is useless.It will be made according to files as default.

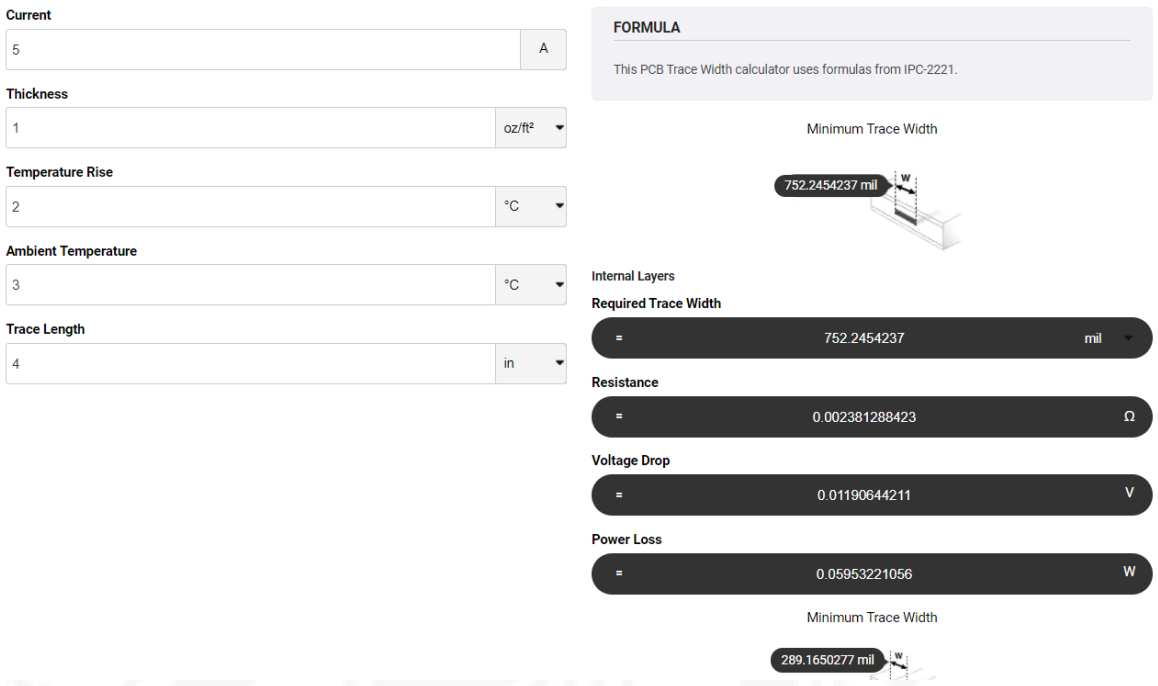
Finished Copper: Bare board(0 oz Cu) | **1 oz Cu** | 2 oz Cu | 3 oz Cu | 4 oz Cu | 5 oz Cu | 6 oz Cu | 7 oz Cu | 8 oz Cu | 9 oz Cu | 10 oz Cu | 11 oz Cu | 12 oz Cu | 13 oz Cu

*Min Track/Spacing ≥ 8/8mil, 3 - 13oz Cu opitions will be available.

https://www.pcbway.com/

PCB Trace width calculator

- https://www.digikey.in/en/resources/conversion-calculators/conversion-calculator-pcb-trace-width



- https://en.wikipedia.org/wiki/IPC_(electronics)

Types of Manufacturing Process

- ✓ Subtractive
- ✓ Additive
- ✓ Semi-additive processes

Simple single layer PCB Manufacturing at home

Copper Clad Plates

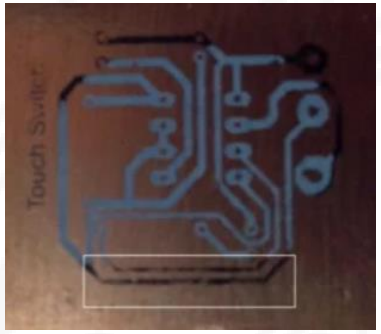
Rubbing away the top oxide layer

Place the printed side of the paper on the plate

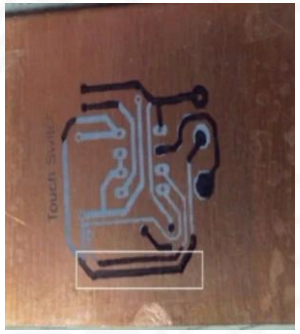
Using the permanent marker to sketch the PCB

Iron the paper onto the plate

Peeling the paper



Light trace



Darkening the trace

Etching the plate

Etched copper plate

Multilayer Manufacturing process

- Step 1 – The Design
- Step 2 – Printing the Design
- Step 3 – Creating the Substrate
- Step 4 – Printing the Inner Layers
- Step 5 – Ultraviolet Light
- Step 6 – Removing Unwanted Copper
- Step 7 – Inspection
- Step 8 – Laminating the Layers
- Step 9 – Pressing the Layers
- Step 10 – Drilling
- Step 11 – Plating
- Step 12 – Outer Layer Imaging
- Step 13 – Plating
- Step 14 – Etching
- Step 15 – Solder Mask Application
- Step 16 – Silkscreening
- Step 17 – Surface Finish
- Step 18 – Testing

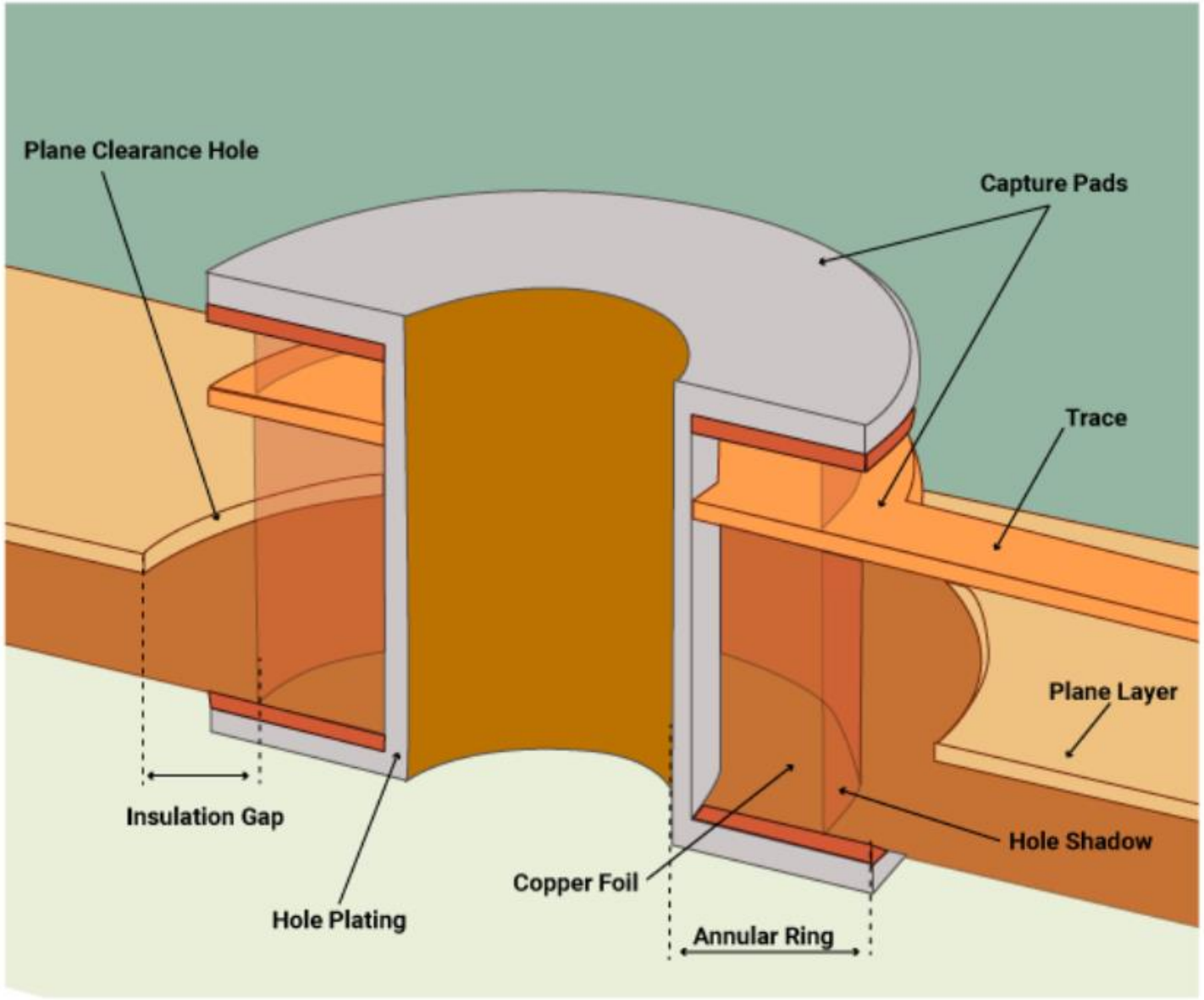
Key Steps in PCB Design Process

- Schematic capture
- Preparing the schematic for layout
  - Attaching package symbols (footprints)
  - Creating a netlist
- Setting up the PCB design environment
- Layout
- Prepare for manufacture
- Generating artwork (Gerber files) & drill files
- Generating documentation
- Submitting PCB files for fabrication check

Tools for PCB Design

- ✓Orcad Capture-CIS , Allegro
- ✓Altium
- ✓Pads (Power PCB)
- ✓SOLIDWORKS PCB
- ✓KiCad EDA
- ✓Ultiboard by National Instruments
- ✓DipTrace
- Eagle（Easily Applicable Graphical Layout Editor） ...
- EasyEda. ...

PCB Terminologies-What are PADS



A pad is the exposed region of metal on a circuit board that the component lead is soldered to

PCB Terminologies-What are PADS

E = Pitch

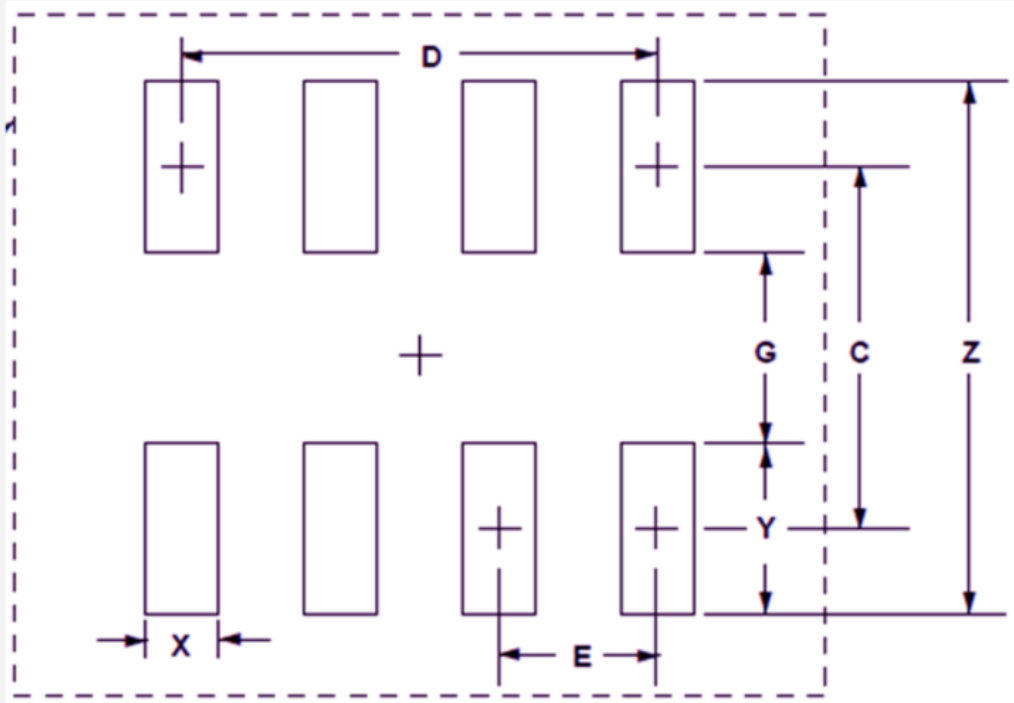
C = Row distance

X = Pad width

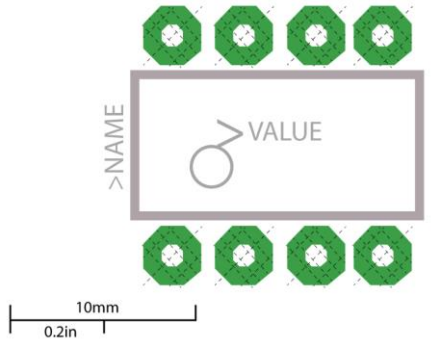
Y = Pad length

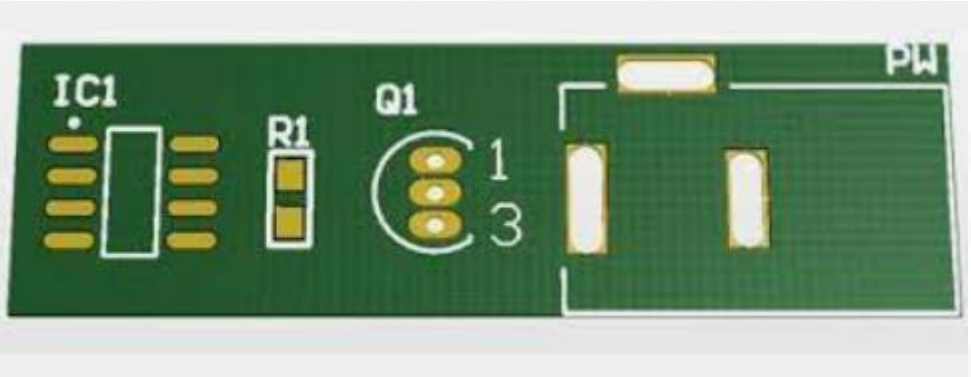
C = G+Y

Z = 2Y+ G



Pad design for surface mount pads





Surface mount pads

Pads used to mount surface mount components are called surface mount pads. These pads have the following features:

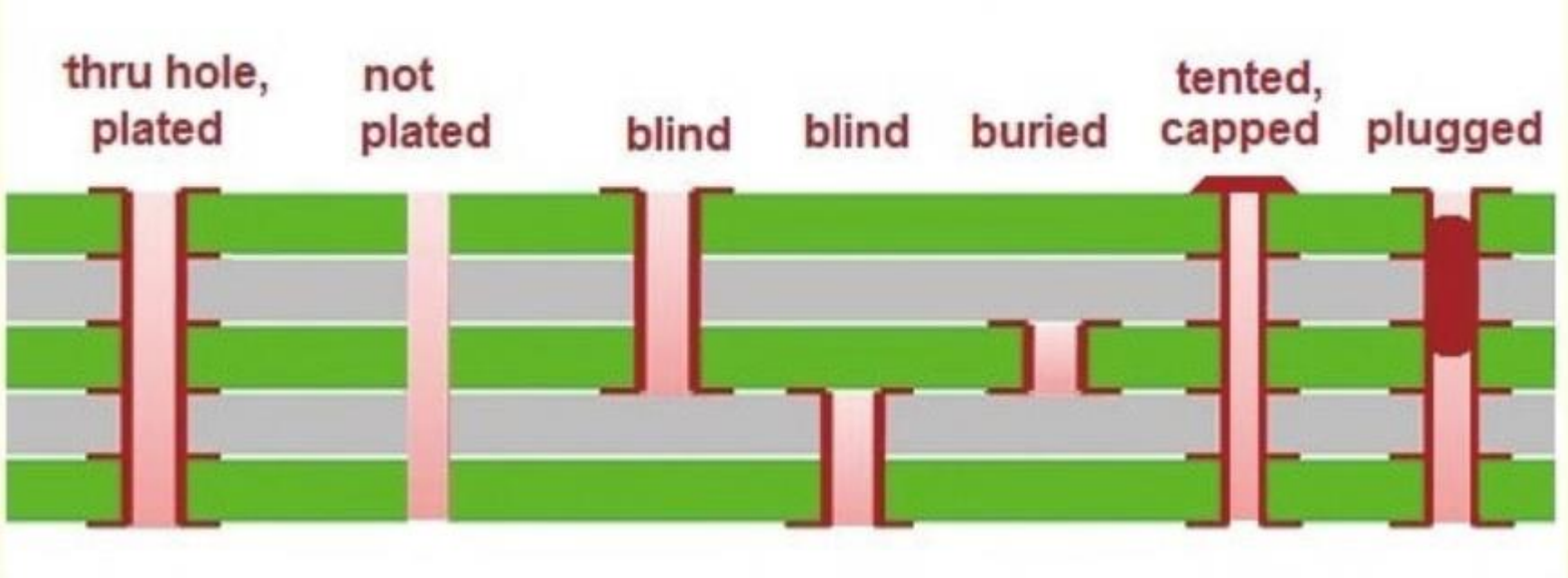
1.Pad which shows the copper area. This can be rectangle, round, square, or oblong.

2.Solder mask layer

3.Solder paste

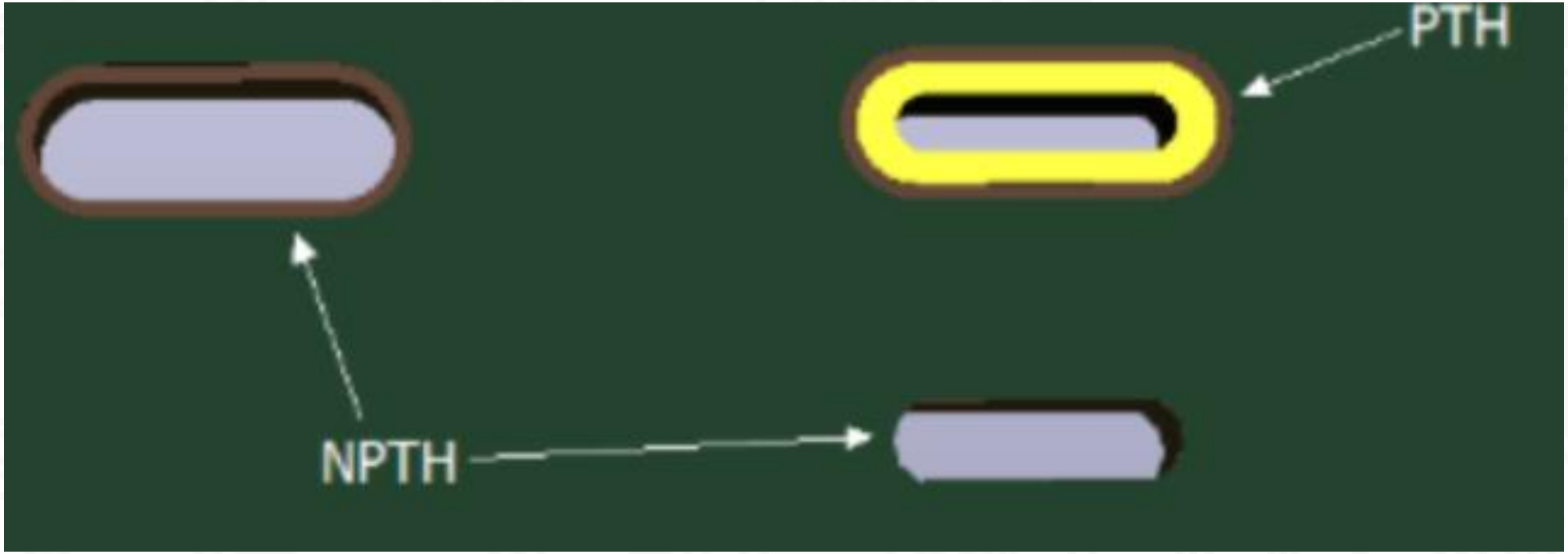
4.Pad number (number of pads present for the component)

What is Via Interconnect access ?

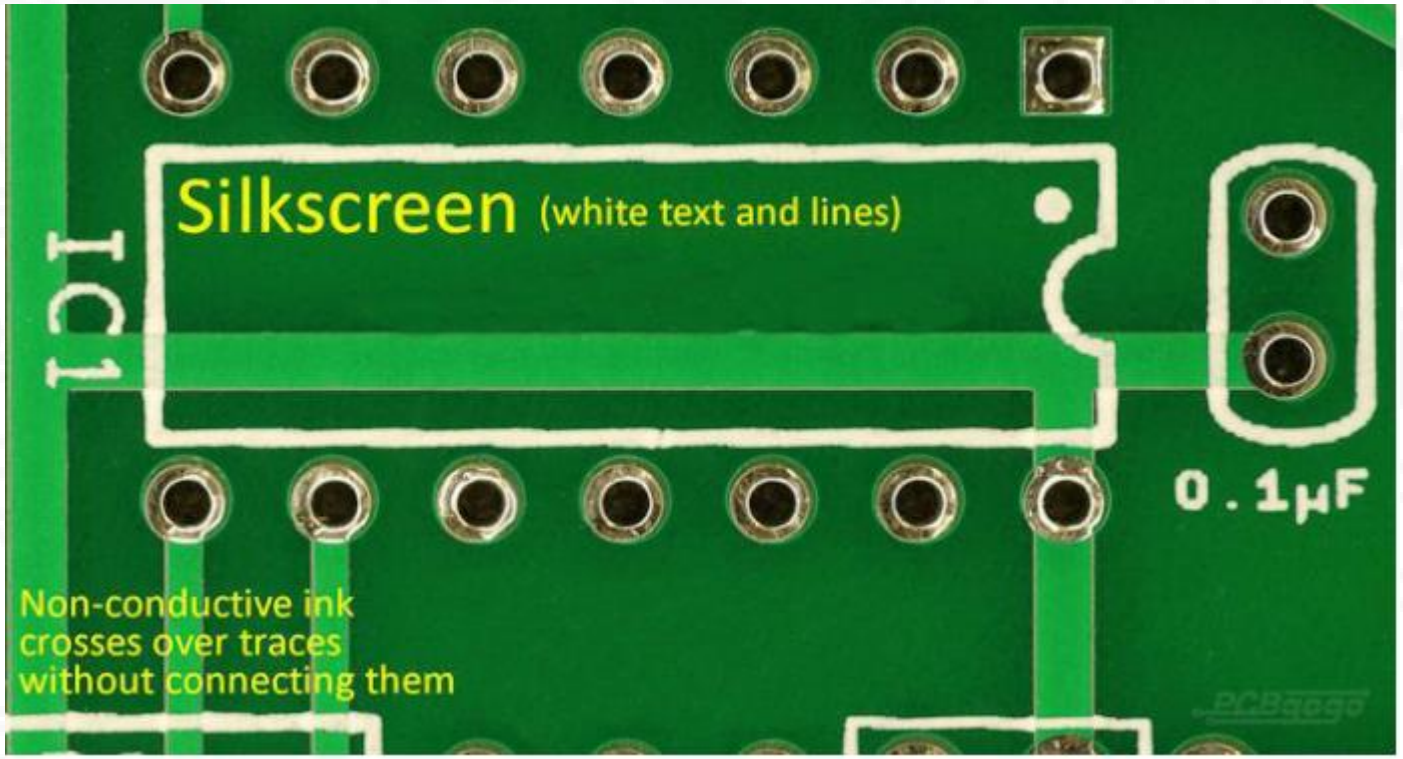


•a through via goes through all the layers. Depending on the cladding in the area where the via is drilled, it may connect to all or just some of the internal layers;
•a blind via goes from the top or bottom layer through one or more layers but is not open at the far end. Plating this via is difficult since a bubble can form at the bottom and prevent proper flow of the solder;
•a buried via connects only internal layers and has no direct access from the top or bottom of the finished bo

What is Plated Vs Non Plated holes?



What is Silkscreen?



A silkscreen can provide the following information:

- ✓ Polarity of parts
- ✓ Location of parts through reference designators
- ✓ Identify test points
- ✓ Identification numbers unique to each board

What is Solder Mask

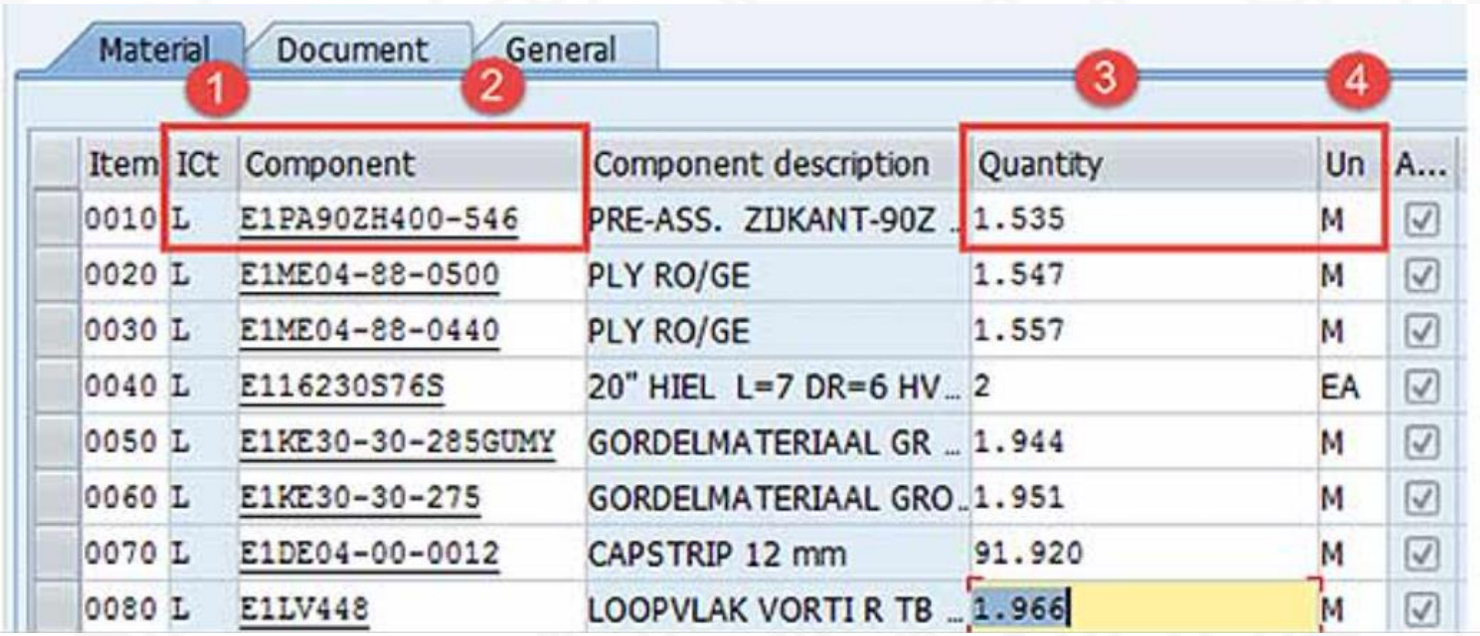
- Solder mask is a thin layer of polymer that is **put on a circuit board to protect the copper from oxidation and shorts during operation**. It also protects the PCB from environmental influences such as dust and several other contaminants that may lead to shorts in the long run.

- It is **due to the solder mask**, which protects the copper circuits printed on the fiber glass core to prevent short circuits, soldering errors, etc. The color of the solder mask gives the board its appearance.

What is Bill of materials (BOM)?

A bill of materials (BOM) is a comprehensive inventory of the raw materials, assemblies, subassemblies, parts and components, as well as the quantities of each, needed to manufacture a product.

- ✓ **Item No**
- ✓ **Component Name**
- ✓ **Description**
- ✓ **Qty**
- ✓ **Manufacturing Part no**
- ✓ **Order code**
- ✓ **Qty**
- ✓ **Price**
- ✓ **Type**
- ✓ **No of soldering Pads**



Item	ICt	Component	Component description	Quantity	Un	A...
0010	L	E1PA90ZH400-546	PRE-ASS. ZIJKANT-90Z ..	1.535	M	☑
0020	L	E1ME04-88-0500	PLY RO/GE	1.547	M	☑
0030	L	E1ME04-88-0440	PLY RO/GE	1.557	M	☑
0040	L	E116230S76S	20" HIEL L=7 DR=6 HV...	2	EA	☑
0050	L	E1KE30-30-285GUMY	GORDELMATERIAAL GR ...	1.944	M	☑
0060	L	E1KE30-30-275	GORDELMATERIAAL GRO..	1.951	M	☑
0070	L	E1DE04-00-0012	CAPSTRIP 12 mm	91.920	M	☑
0080	L	E1LV448	LOOPVLAK VORTI R TB ...	1.966	M	☑

What is a Reference Designator?

- A reference designator unambiguously identifies a component within an electrical schematic or on a printed circuit board. The reference designator usually consists of one or two letters followed by a number, e.g. R13, C1002

MK	Microphone
MP	Mechanical part (including screws and fasteners)
P	Plug (most-movable connector of a connector pair) | Plug connector (connector may have "male" pin contacts and/or "female" socket contacts)
PS	Power supply
Q	Transistor (all types)
R	Resistor
RN	Resistor network
RT	Thermistor
RV	Varistor
S	Switch (all types, including push-buttons)
T	Transformer
TC	Thermocouple

https://dexpcb.com/manual/standard-reference-designators.htm

What is Netlist ?

- The netlist **contains the electrical connections between the components on the circuit board**, and is usually held in textual format (see EDIF). In printed circuit board production a netlist (generated from the production data) is used to carry out an electrical test (E-test) to find incorrect or missing connections

Layers in Gerber

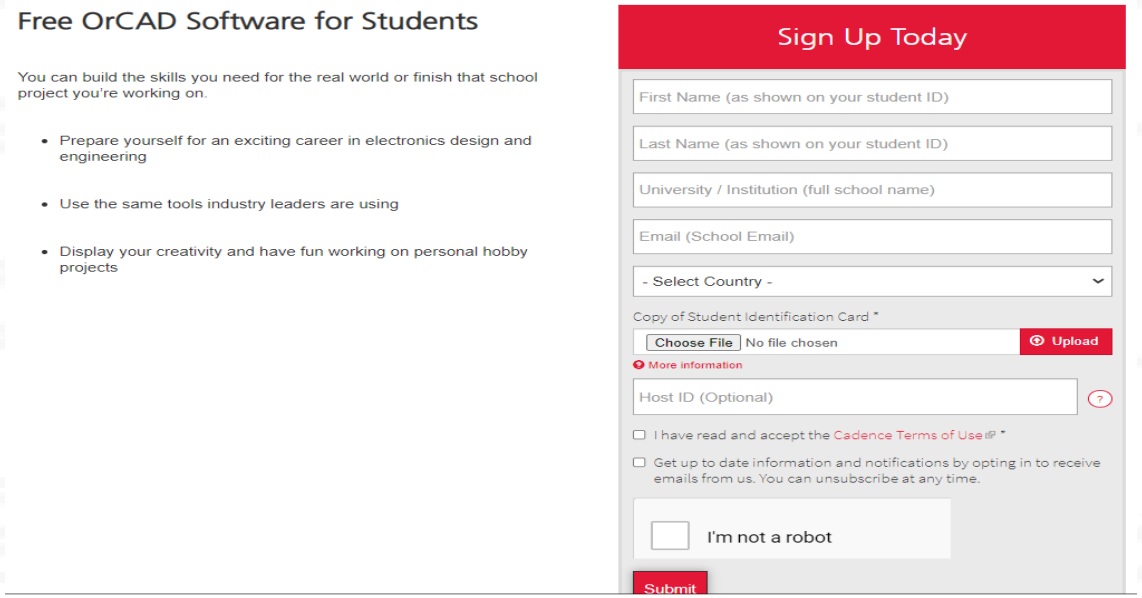
- Global Layer
- Top Layer
- Bottom Layer
- Silk screen Top
- Silk Screen Bottom
- Assembly TOP
- Assembly Bottom
- Inner Layer
- Drill layer

What is Artwork

Artwork is basically a manufacturing tool used in fabricating printed wiring because it uniquely defines the pattern to be placed on the board. Artwork displays only those items that have to be generated as copper patterns in the manufacture of the PCB. Therefore, the artwork will necessarily include solder pads, lands and conductors true to scale in respect of their dimensions, but shown at the scaled level.

Downloading the TRIAL VERSION and Installation Video

- https://www.orcad.com/orcad-academic-program



Raju.D.S |Sr.Technical Sales Engineer |
FTD Automation Private Limited, Bangalore-560095|
Tel: **+91 (080) 42600900** |Mobile:**8880907722/9538505552** |
Mail ID:**rajuds@ftdautomation.com**
www.ftdautomation.com

Today's Assignment

- https://www.pcbpower.com/blog-detail/Printed-Circuit-Board-Layers
- https://www.pcbway.com/
- https://www.digikey.in/en/resources/conversion-calculators/conversion-calculator-pcb-trace-width

Certificate of Internship at Rs 300

- What You get
  - Recorded Video access for 6 Months
  - Free Hackathon session on ZOOM for 2 Hrs(Create a simple 2 layer pcb)
    - Prerequisites(Have to installed Orcad before the event)

Combo Offer

Combo 1 - PCB Design + Embedded System Design&Iot + FPGA =999 Rs(6 Months Validity)

Combo 2 - PCB Design + Embedded System Design&Iot + Arduino =999 Rs(6 Months Validity)

Thank You