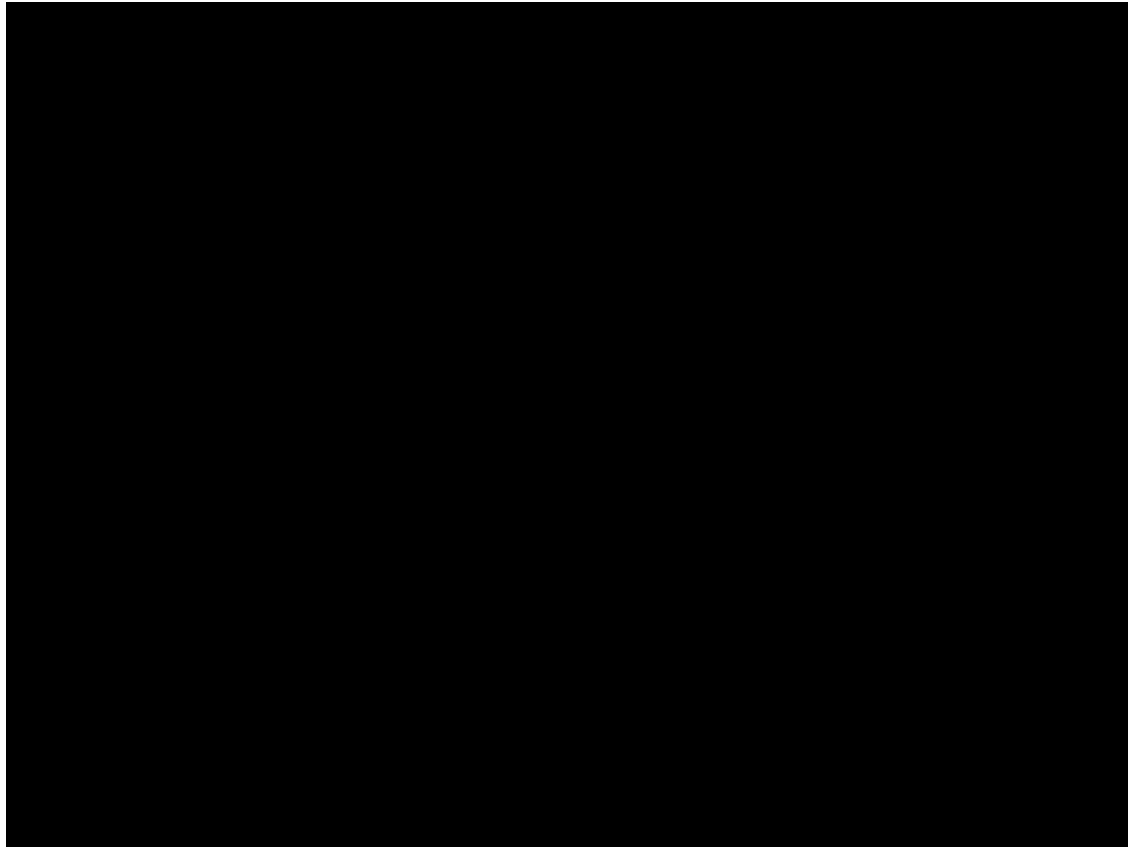


Digisim - PCB Design and industrial electronics Workshop

By Ayush Agarwal
ECE'24
IIT BHU Varanasi

Event Trailer (Quick summary , watch at end)



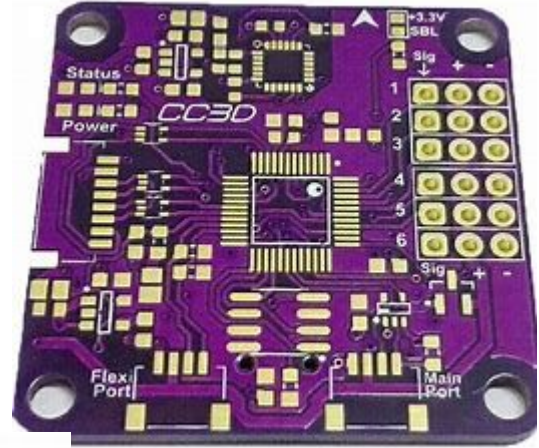
So ,
what
is
PCB ?

To the layman , it is green chips



(Not the potato chips tho xD)

Tho it can be of different colours too :)



PCB stands for

Printed
Circuit
Board

Here is the wikipedia definition



- Main page
- Contents
- Current events
- Random article
- About Wikipedia
- Contact us
- Donate
- Contribute
- Help
- Learn to edit
- Community portal
- Recent changes
- Upload file
- Tools
- What links here
- Related changes
- Special pages
- Permanent link
- Page information
- Cite this page
- Wikidata item
- Print/export
- Download as PDF
- Printable version

Not logged in | Talk | Contributions | Create account | Log in

Article | **Talk**

Read | **Edit** | View history

Search Wikipedia

Printed circuit board

From Wikipedia, the free encyclopedia

Not to be confused with [Printed electronics](#) or [Polychlorinated biphenyl](#).
"PC board" redirects here. For the mainboard of personal computers, see [Motherboard](#).

A **printed circuit board** (**PCB**; also **printed wiring board** or **PWB**) is a medium used in [electrical and electronic engineering](#) to connect [electronic components](#) to one another in a controlled manner. It takes the form of a [laminated](#) sandwich structure of conductive and insulating layers: each of the conductive layers is designed with an artwork pattern of traces, planes and other features (similar to wires on a flat surface) [etched](#) from one or more sheet layers of copper [laminated](#) onto and/or between sheet layers of a [non-conductive](#) substrate.^[1] Electrical components may be fixed to conductive pads on the outer layers in the shape designed to accept the component's terminals, generally by means of [soldering](#), to both electrically connect and mechanically fasten them to it. Another manufacturing process adds [vias](#): plated-through holes that allow interconnections between layers.

Printed circuit boards are used in nearly all electronic products. Alternatives to PCBs include [wire wrap](#) and [point-to-point construction](#), both once popular but now rarely used. PCBs require additional design effort to lay out the circuit, but manufacturing and assembly can be automated. [Electronic design automation](#) software is available to do much of the work of layout. Mass-producing circuits with PCBs is cheaper and faster than with other wiring methods, as components are mounted and wired in one operation. Large numbers of PCBs can be fabricated at the same time, and the layout has to be done only once. PCBs can also be made manually in small quantities, with reduced benefits.^[2]

PCBs can be single-sided (one copper layer), double-sided (two copper layers on both sides of one substrate layer), or multi-layer (outer and inner layers of copper, alternating with layers of substrate). Multi-layer PCBs allow for much higher component density, because circuit traces on the inner layers would otherwise take up surface space between components. The rise in popularity of multilayer PCBs with more than two, and especially with more than four, copper planes was concurrent with the adoption of [surface mount technology](#). However, multilayer PCBs make repair, analysis, and field modification of circuits much more difficult and usually impractical.

The world market for bare PCBs exceeded \$60.2 billion in 2014^[3] and is estimated to reach \$79 billion by 2024.^{[4][5]}

Contents [hide]

1

History

1.1

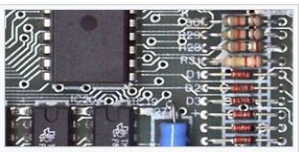
Predecessors

1.2

Evolution of PCB



PCB of a DVD player. PCBs may be made in other colors.



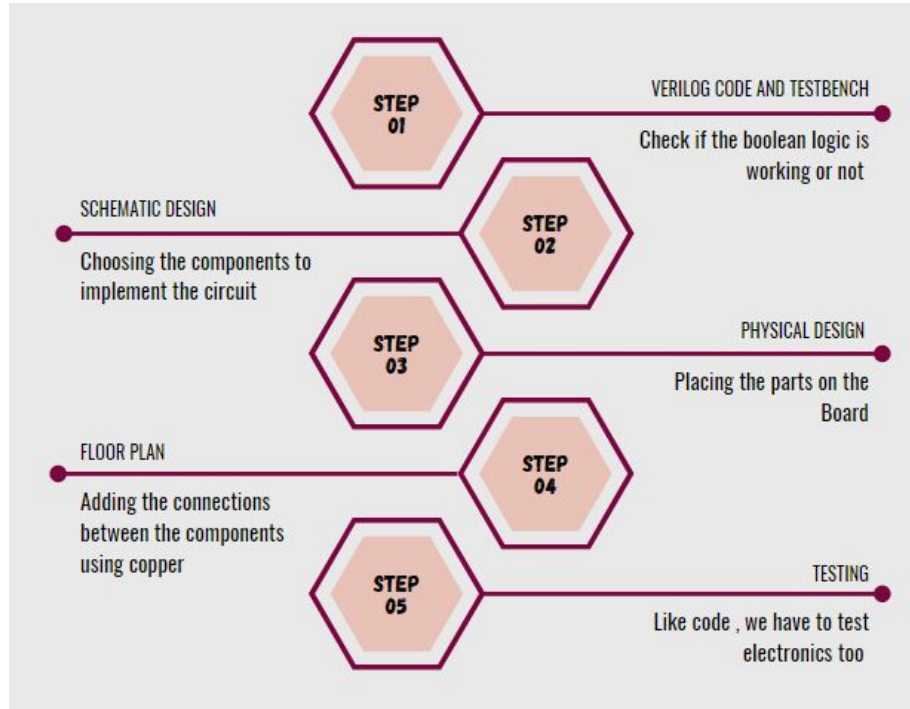
Part of a 1984 [Sinclair ZX Spectrum](#) computer board, a PCB, showing the conductive traces, [vias](#) (the through-hole paths to the other surface), and some electronic components mounted using through-hole mounting.

In short

PCB are the way real life commercial electronics are made

In the industry

How is a PCB made in industry ? [SIMPLIFIED]



(The actual process has many more parts which we will discuss when we have covered more topics)

(The verilog part wont be discussed in this event , that would be covered in ichip, dont worry , we can directly make schematic)

Why do PCB design ?

- Very few btech graduates even from the best IITs know PCB design , puts you ahead of 90% of competition in core industry .
- Fun
- Takes very less time to learn
- Skill to add in resume



So , Starting PCB Design

Eagle cad intro , installation

Schematic board and gerber files

Single and multi layers

Surface mount vs normal components

Vias

Optimal design rules

Example design

PCB Designing - Software

- Eagle CAD is a very popular PCB design software , so we will use it for the event
- Installation link -

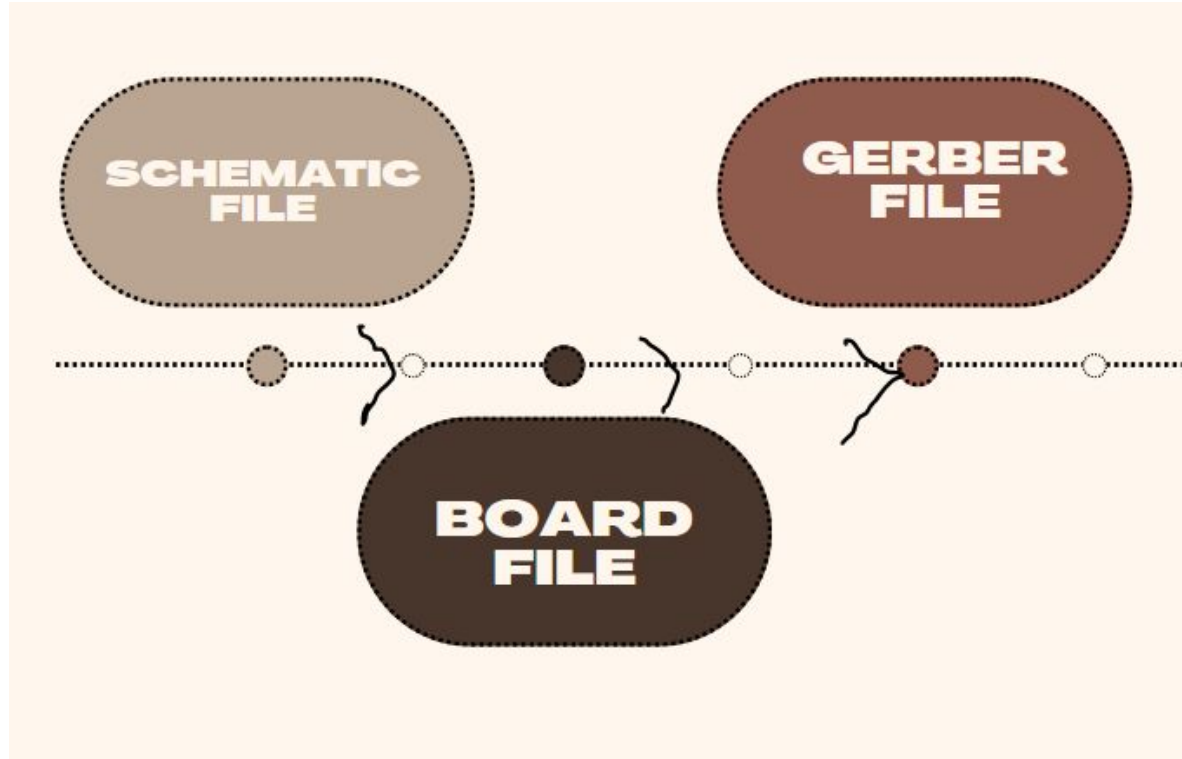
[Download EAGLE | Free Download | Autodesk](#)



Basic concepts in PCB

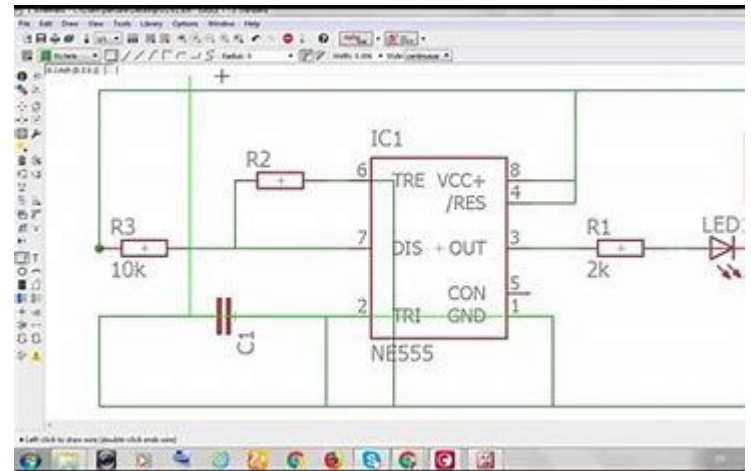
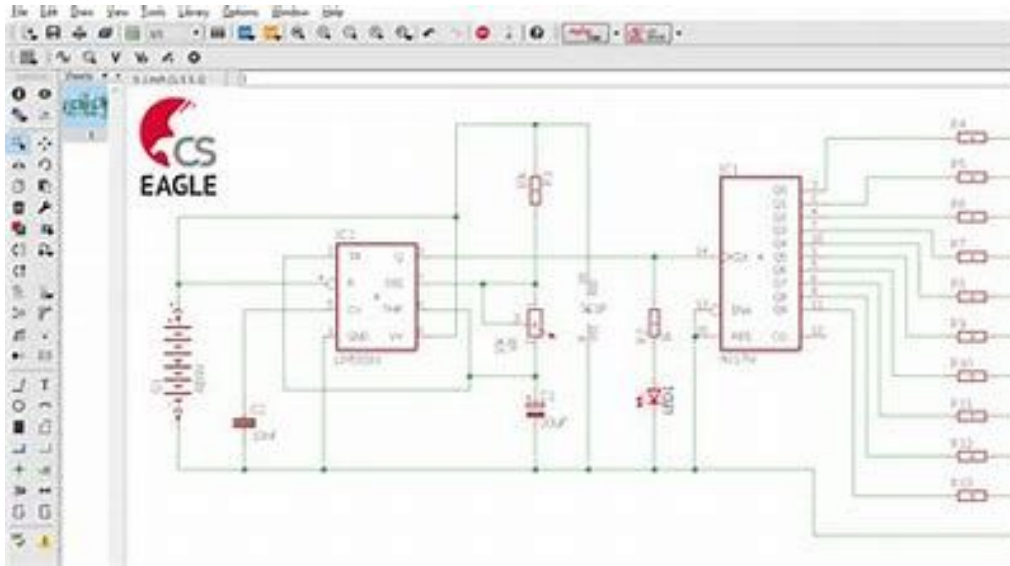
How is PCB made in Eagle CAD

3 steps -



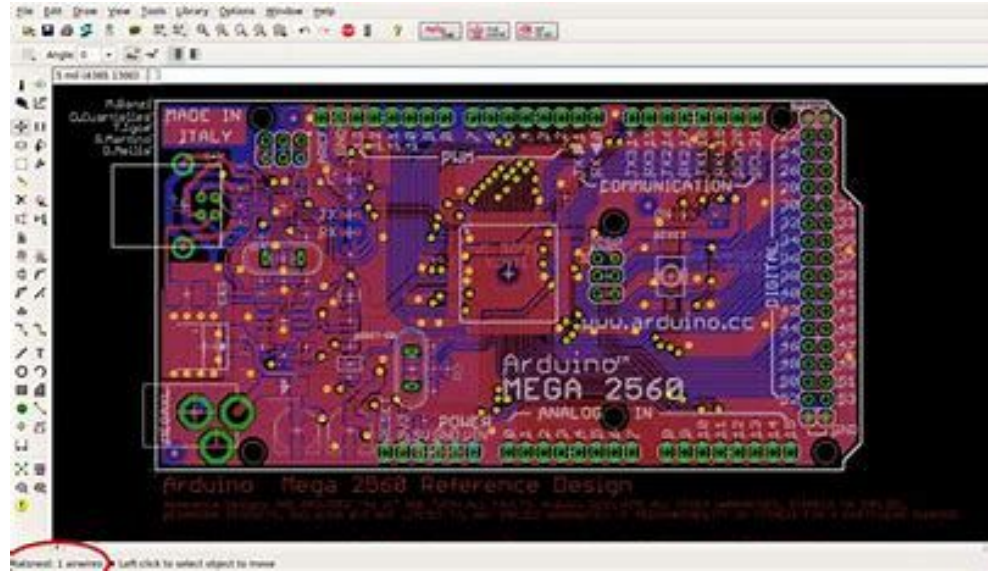
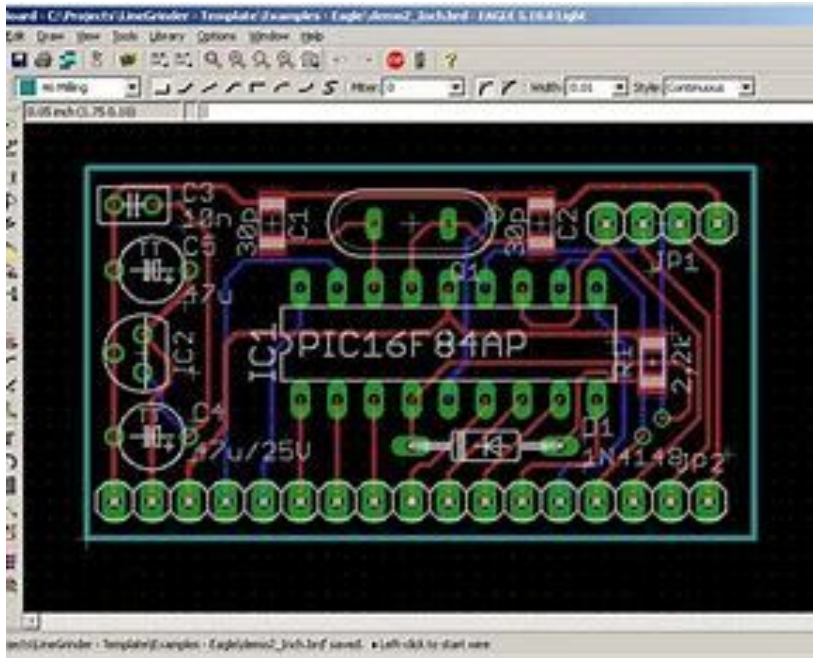
Schematic File

The file with the circuit diagram



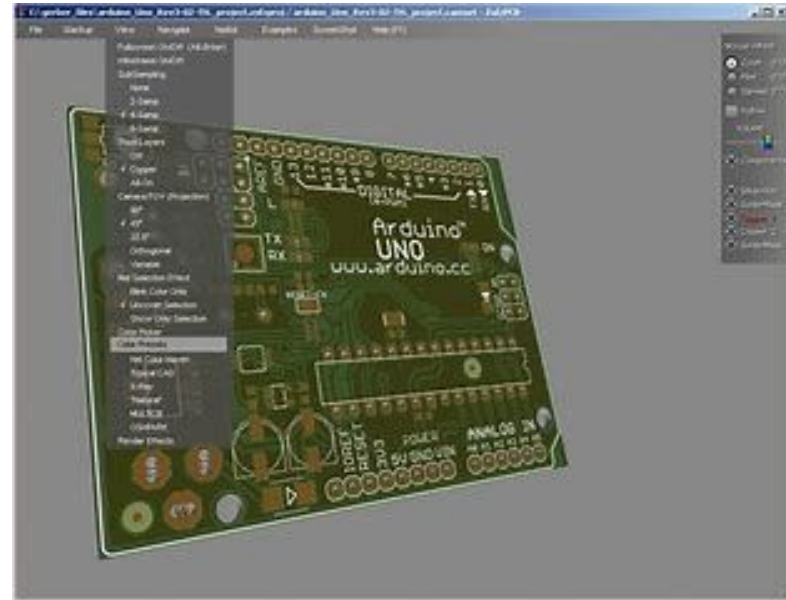
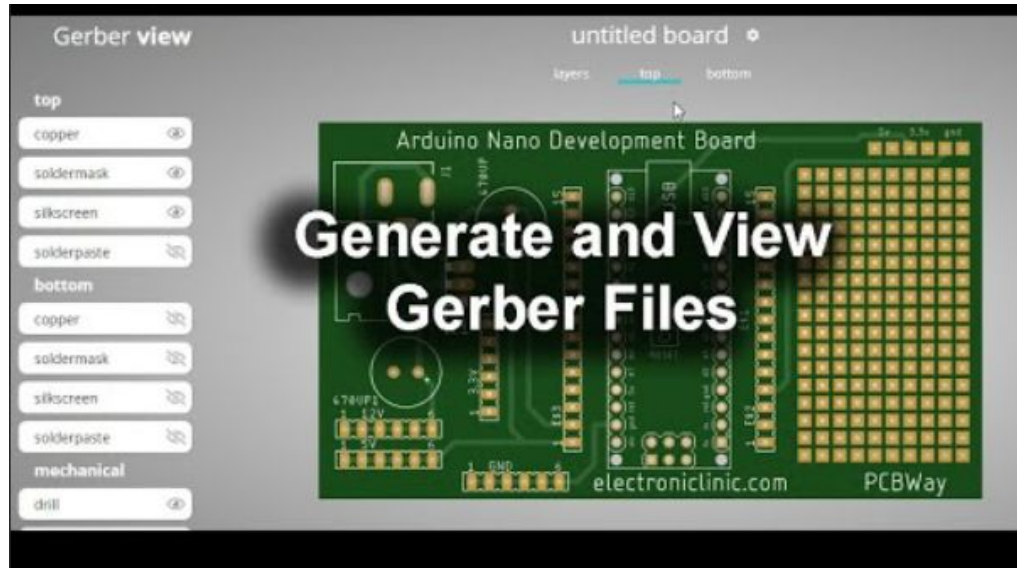
Board File

The window where all the components are placed on the Board and copper connections are made between them



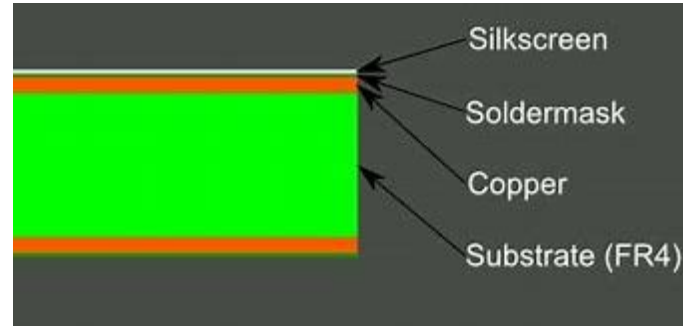
Gerber File

The output file which is sent to the machine for making the design physically (like CAD file for a 3D printer)



What are PCB's fundamentally made up of ?

- Silkscreen layer - white , mostly for writing on the pcb
-
- Solder Mask - Protection layer , gives colour , mostly green
-
- Copper layer - copper , conductive , used for connection , most work here
-
- Substrate layer - fiberglass , insulator , polymer



Source - [What are printed circuit boards made of? | ABL Circuits](#)

(here the word layer refers to the different materials layers and not the different conductive layers)

Common PCB layers

Each type of PCB contains a different number of layers that contribute to its functionality. However, no matter which type of PCB you choose, each board contains the same essential foundation. This means that all PCBs contain the following four layers:

Substrate layer

This is usually made from fibreglass, which gives the board its rigidity. Substrate layers can also be made with epoxies, but these lack the durability that fibreglass provides.

Copper layer

As you'd expect from the name, the copper layer of a PCB is made from a thin layer of copper foil that's laminated to the board using heat.

When we talk about different 'layers' of PCB, we're talking about how many copper layers they consist of. For example, a single sided PCB will only have one layer of conducting material on one side of the board. In this scenario, the other side of the board is used for incorporating different electronic components. Meanwhile, a double sided PCB will mount the conductive copper and components on both sides of the board.

The thickness of the copper layer will be determined by the amount of power the PCB needs to withstand. PCBs that need to handle a higher level of power will have a thicker level of copper.

Solder mask layer

The solder mask layer is placed on top of the copper and provides most PCBs with their green colour. This layer insulates the copper and ensures that it doesn't come into contact with any other elements.

Silkscreen layer

The silkscreen layer is primarily added for the benefit of humans. It involves adding letters, numbers and symbols to the board so it's easier for users to understand the functionality of different pins and LEDs.

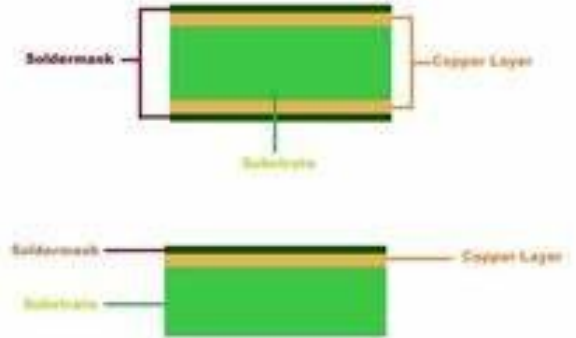
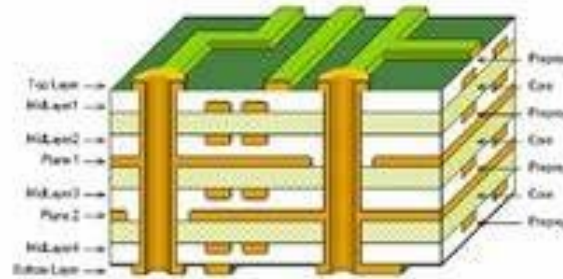
Single layer vs Multi layer PCB (here layer is copper layer)

Single Layer PCB -
Components and
wiring only on one side

Double Layer PCB -
top and bottom have
the circuit

Multi layer PCB -
wiring and stuff done
even in the inside of
the Chip's thickness

Printed Circuit Boards Single Vs Double Vs Multilayer



[Insert live demonstration using paper]

So how do I connect different layers in PCB ?

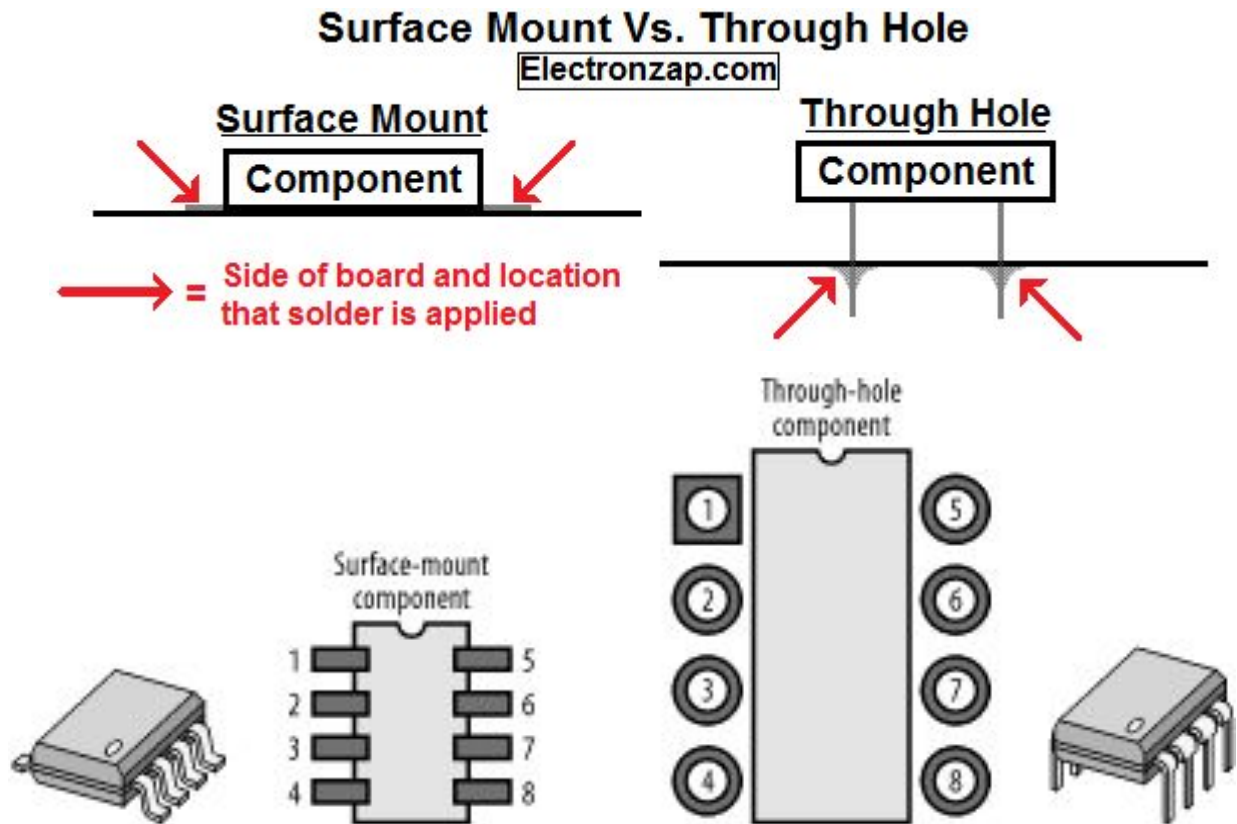
Using either

holes (in hole mount technology)

Or

Vias (in Surface mount technology)

Hole Mount vs Surface mount technology



Hole Mount vs Surface mount technology

Through-Hole	Surface Mount
Components require holes to be drilled	Components are mounted to surface pads; no holes are required.
Components are mounted typically on one side	Components can be mounted on both sides
Assembly is manual	Assembly is automated
Soldering can be manual or automated	Soldering is typically automated.
Stencils are not used	Uses stencils
Vias in pads not possible.	Vias in pads possible.
Suited where component density is low	Works well for high component density
Rework is easier	Rework is complex

Surface mount changed the industry heavily when it first came

Through hole is still not extinct , it is used by hobbyists and places where connection should be tighter



A surface mount resistor is a **tiny rectangular ceramic body with silver conductive edges on either end**. Also referred to as surface mount technology, an SMD resistor offers advantages in saving space on printed circuit boards (PCBs). It features the resistance value code printed onto it, where there is space.



Aluminum Smd
Electrolytic Capacitor ...



Ceramic SMD Capacitors, For Electronics
Equipments, Rs 0.5/unit | I...



Electrolytic Surface
Mount Capacitor Star...



API Delevan - Power
Inductors - Surface Mount



Surface Mount Inductor Packages



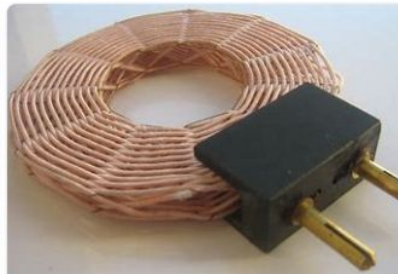
1nH - 100nH Ceramic Chip
Inductors , Surfac...



40 Amps Surface Mount
Inductor



Surface Mount Inductors : CWS
Coil Winding Specialist, manu...



What is an Inductor?



Inductor 100uH 3A



Types of Inductors - YouTube

Resistor

Passive electrical component



Capacitor - Wikipedia



Using the HC-05 Bluetooth Module at

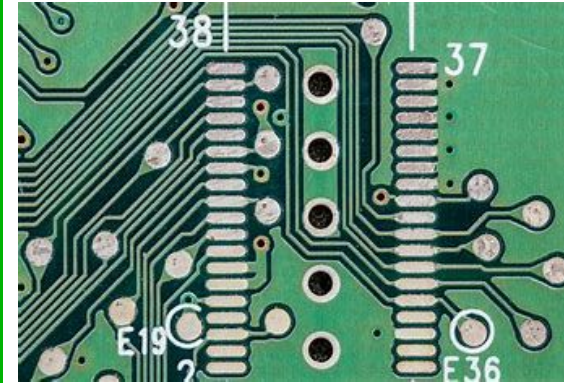
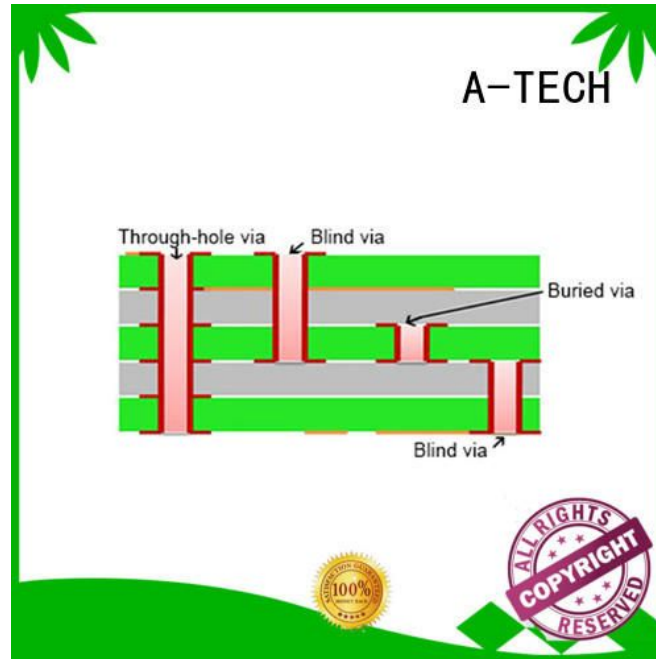
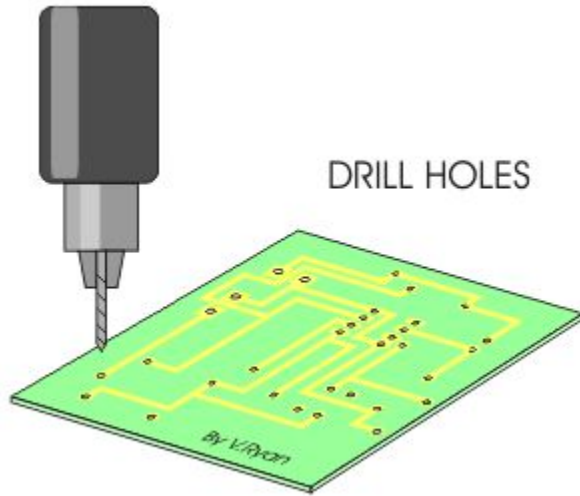


5pcs 100uF 250V 16x26mm
Capacitor

So how to connect components when PCB is multilayered and SMT?

Using **Vias**

Vias are metal cylinders (Look like holes) which help in connecting different layers of the PCB

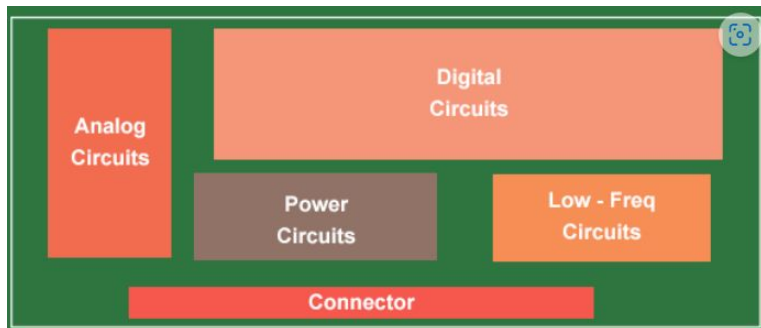


Floor Plan -

“Component placement affects how you **manufacture a PCB** and whether you can even manufacture a PCB. Improper component placement can dramatically increase board costs. In fact, design decisions as simple as the orientation of a component can affect its solderability.”

Tips For Effective Component Placement Layout In PCB Design

- **Group Components By Function** - Ensure that the components are segregated according to their functional block in the circuit. The rule of thumb is to keep the noisiest signals away from the highly-sensitive ones.



- **Keep Components Away From Heat Dissipating Area** - In power-demanding applications, the voltage regulator heats up significantly. In power-demanding applications, the voltage regulator heats up significantly. The same applies when you're using power op-amp or other heat emitting device.

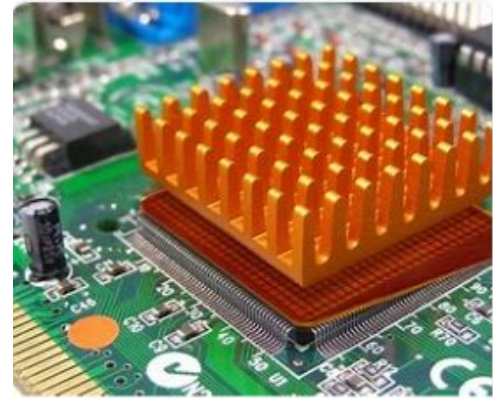
Managing the heat - Hot running components, such as processors, should be located towards the center of the design to better dissipate their heat throughout the board. Individual power supplies should not be located next to each other. It is also important to be mindful of the airflow across the board so as to not block it from providing cooling. This may require relocating a taller component that is blocking the airflow. Also, allow room in your placement for **heat sinks** and other thermal dissipation devices and tactics.



Aluminium heat sink for PCB board



Heat Sink Attach Archives - Advanced Thermal Solutions



Heat Sink | Tactilus | Tactile Surface Pressure Mapping System...

- **Place Board-To-Wire Connectors Near The Edge** - Not only it does look organized, but keeping connectors near the edge prevent unwanted contact with other components on the PCB.
- **Make Space For Copper Traces** - else you'll find that it's impossible to complete routing all the trace. When placing components, ensure that there are sufficient gaps for copper traces to pass through, especially near components with hundreds of pins .
- **Component orientation** - Align similar components in the same direction. This helps in effective routing in PCB design and ensures an error-free soldering process during assembly.

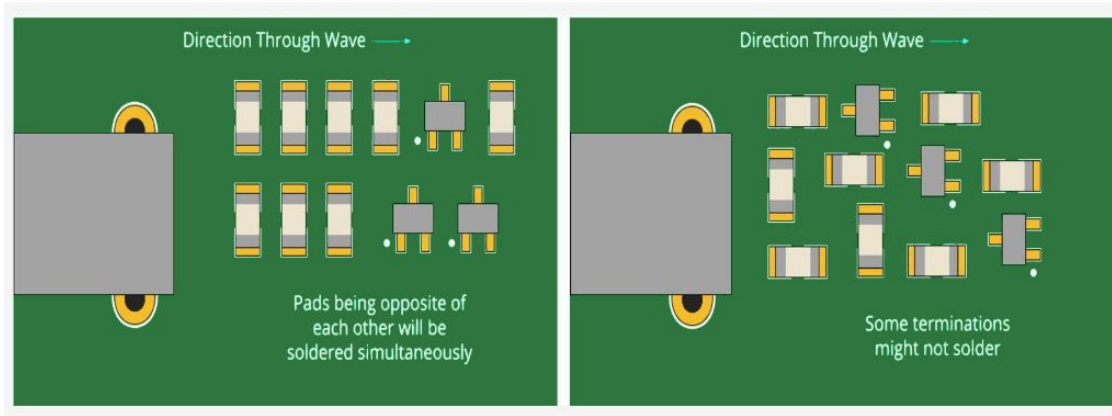


Figure 9: Good Components Orientation (left) and Poor Components Orientation (right)

- In 2 layer PCB , one layer is done horizontally and other is done vertically

Mils or mm ??

In PCB designing , we have different unit for measuring things called mils

Convert units

Length		▼
1	=	0.0254
Mil		Millimeter
▼		▼

FORMULA for an approximate result, divide the length value by 39.3700787

(1 mil is 1/1000 of an inch)

Convert units

Length		▼
39.3700787	=	1
Mil		Millimeter
▼		▼

FORMULA for an approximate result, divide the length value by 39.3700787

ERC and DRC

ERC - Electrical Rule Check

Checks connections and stuff

DRC - Design rule check

Checks if PCB design can be done on a machine or not

Both check the PCB if it is correct or not , and are already there in eagle cad , and are run at the end to detect any errors and correct them

Quick 42 seconds break to learn this all

Schematic , board and gerber files

Silkscreen , Solder mask , Copper layer and Substrate layers

Single and multi layer pcbs

Holes and vias

Surface mount and through hole technology

Floor plan optimisation , thermal optimisation

Mils vs mm

ERC and DRC

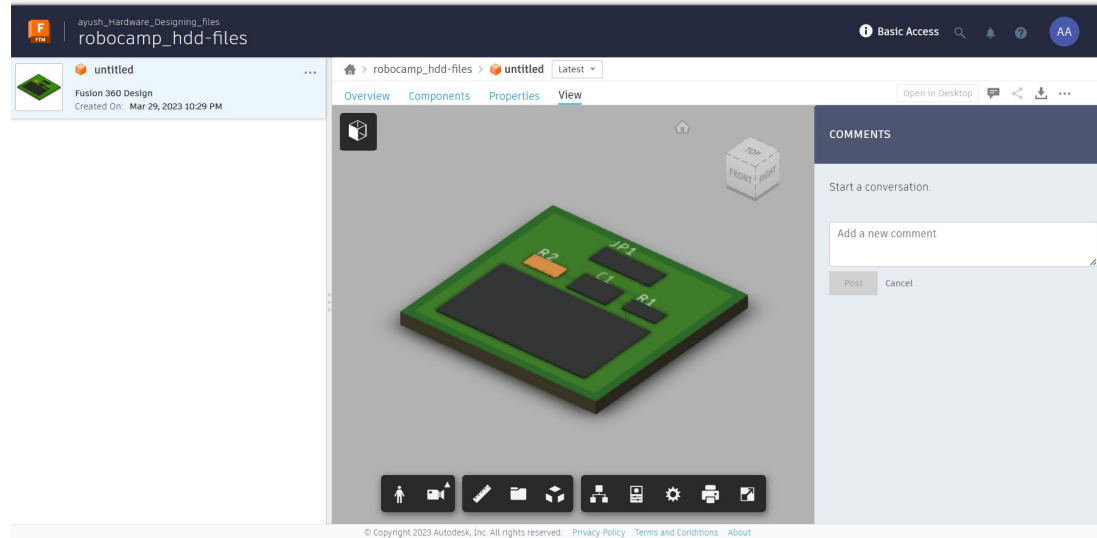
To view what U have made and what will be the output of the PCB machine :

[Online Gerber Viewer - PCB Prototype the Easy Way - PCBWay](#)

Make sure to see the real life view of the PCB u have just made :)

To use this PCB in a cad model

There is an option on the right side for fusion 360 , click on it . U don't need fusion 360 downloaded already , just an account with autodesk company , and u will be able to see ur chip from any angle just like a CAD design . here is a basic cad file example . (For those of u who dont know what cad means , its a file format for 3d models , which robotics engineers make



Wait ... dont screenshot ur project

Click on the right side manufacturing button , then preview , then export jpg .

No need to screenshot ur projects to add to github

Flip the board

Note that in real life u can solder components on both side of the PCB . U can do that in eagle too , click on flip the board symbol (top left middle) , and done . To place a component on the bottom side of the board , click on the mirror tool and then click on the component , the red colour of the base will change to blue (for surface mount devices)

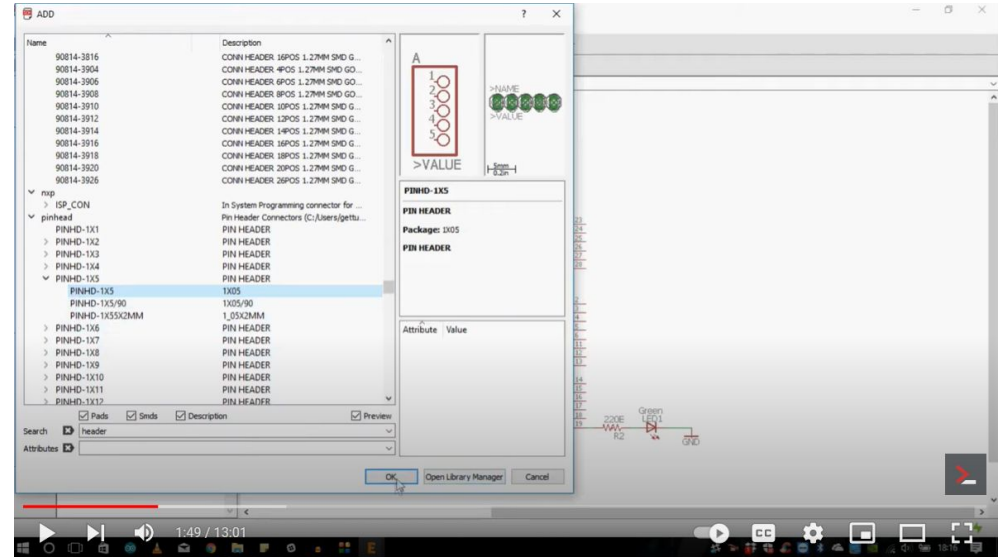
DRC - 45 degree turns vs 90 deg turns

U might notice that the autorouter prefers 45 degree turns of wire rather than 90 degree turns . This is because 45 degrees ensures lesser error while printing the PCB , due to the properties of the PCB machine

Some more information

How to make Arduino like ends for connection of jumper wires to peripherals

Ans - u need to search for pinhead



Autodesk Eagle #4 | More Detailing on Arduino Schematic



Terminal Two

19.7K subscribers

Subscribe

178



Share



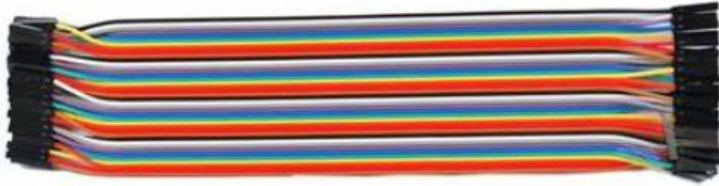
Download



[Autodesk Eagle #4 | More Detailing on Arduino Schematic - YouTube](#)

Male and Female connectors for jumper wires

Female to Female




Male to Female



Male to Male



 hogaki.com

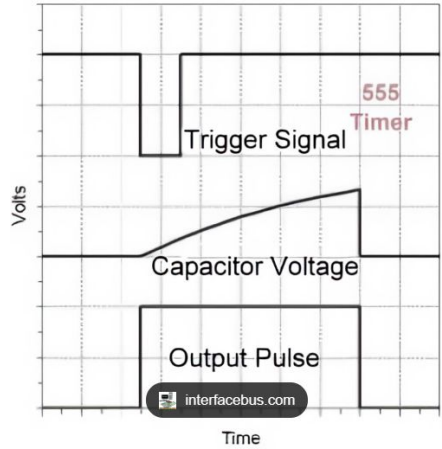


10Pcs 2mm 2x7 Pin 14 Pin
Straight Male Shro...

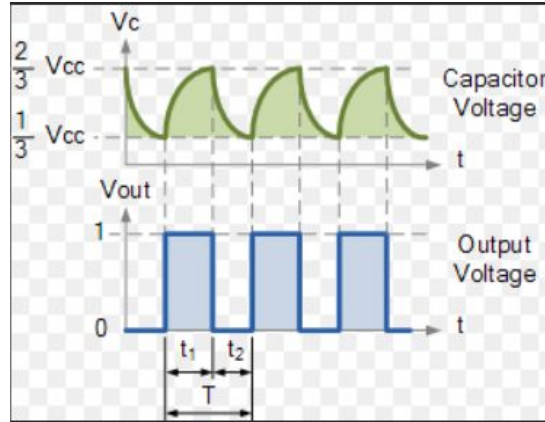


SMT 2.54mm Double Row
Female Breakawa...

555 timer - from signal jammers , to time bomb , to alarm , and tripwire circuits , one IC for all

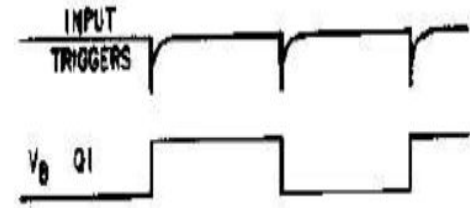


Monostable Multivibrator



Astable Multivibrator

Ask me if u need more info on these :)



Bistable Multivibrator

Very good sources :

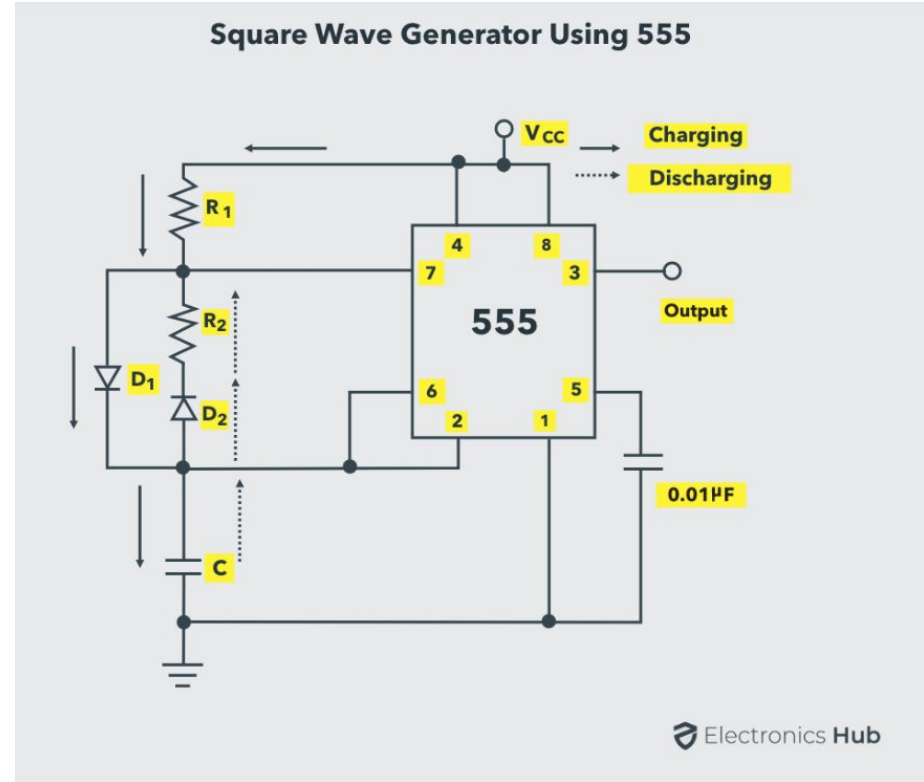
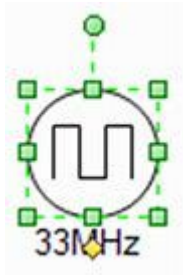
[Introduction to 555 Timer: The Internal Block Diagram and the Pin Diagram Explained - YouTube](#)

[Monostable Multivibrator using 555 Timer Explained \(with Working, Applications and Derivation\) - YouTube](#)

[555 Timer as Astable Multivibrator \(Working, Design and Derivations\) - YouTube](#)

[555 Timer as Bistable Multivibrator Explained - YouTube](#)

Astable Multivibrator - Our clock Signal



In this circuit, while charging, the capacitor charges through R_1 and D_1 by passing R_2 . While discharging, it discharges through D_2 and R_2 .

Therefore, the charging time constant is $T_{ON} = T_C$ and is given by:

$$T_{ON} = 0.693 * R_1 * C$$

And the discharging time constant $T_{OFF} = T_D$ is given by:

$$T_{OFF} = 0.693 * R_2 * C$$

Therefore, the duty cycle D is given by:

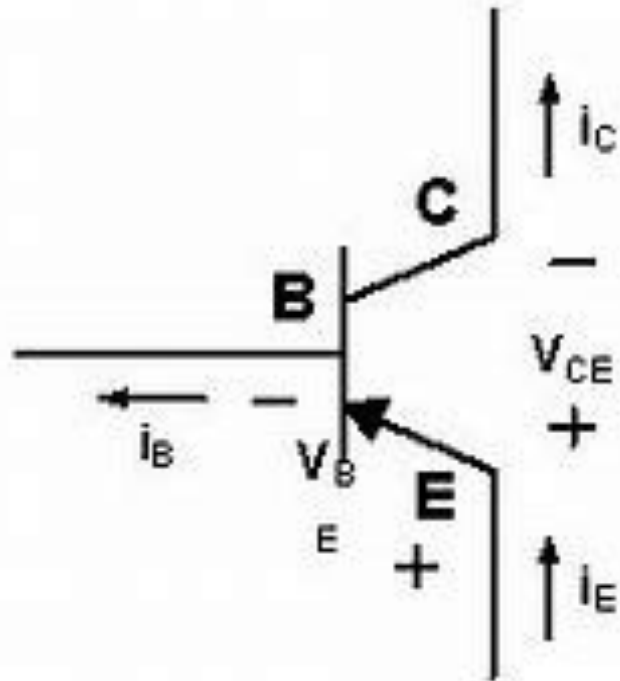
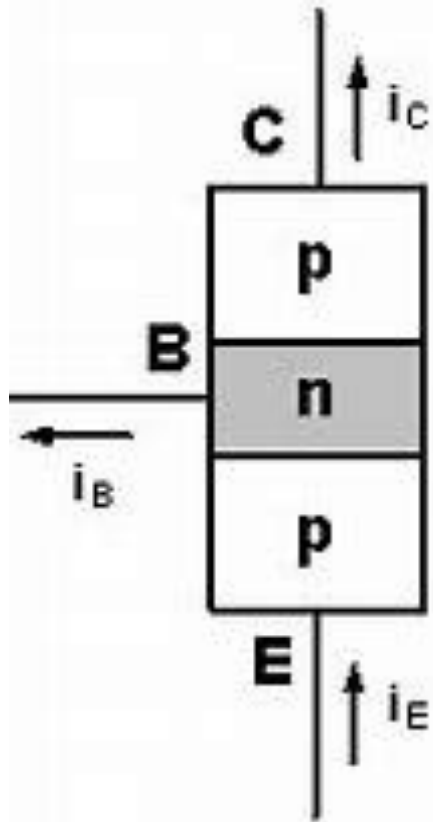
$$D = R_1 / (R_1 + R_2)$$

In order to get a square wave, the duty cycle can be made 50% by making the values of R_1 and R_2 equal. The waveforms of the square wave generator are shown below.

Adding a project to github tutorial

Not so important content for Digisim starts here (tho
very important for electronics) ->

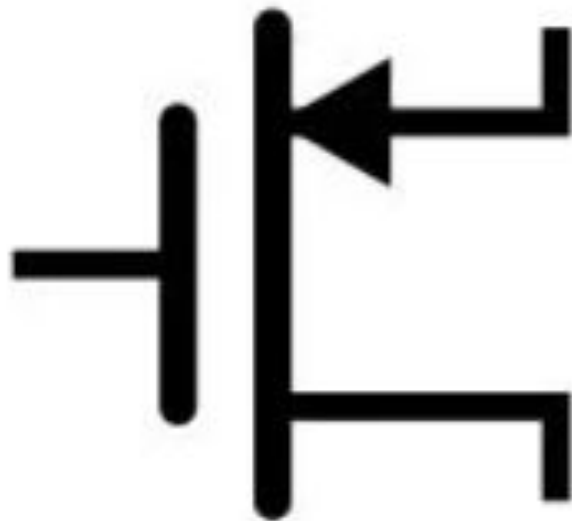
What is this ?



If you said BJT (Bipolar Junction Transistor) ...



Now what is this ?



Enter the better transistors , the MOSFETs

- less power dissipation,
- low operating voltage,
- less noise,
- simple manufacturing process,
- low chip area
- easily scalable

What are MOSFETs you may ask

MOSFETs stand for

Metal

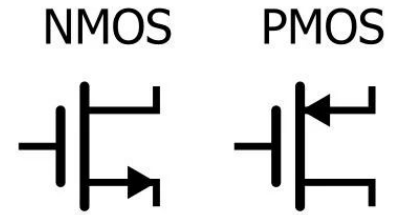
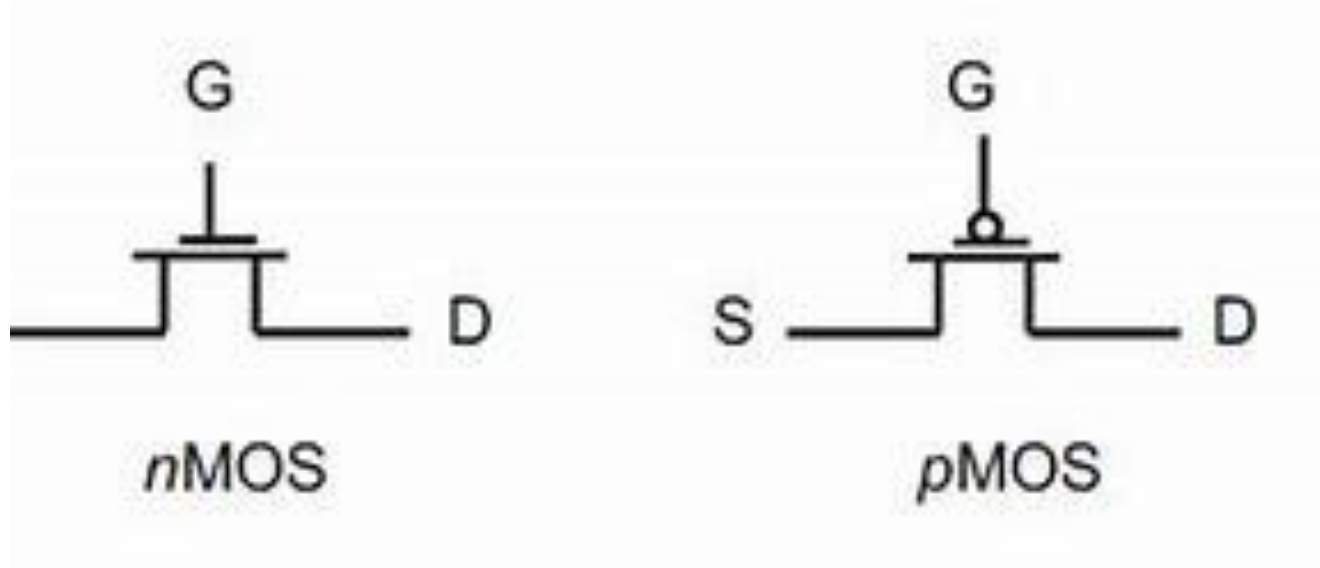
Oxide

Semiconductor

Field-Effect Transistors

Don't worry ! Its way easier than studying BJTs

The NMOS and the PMOS

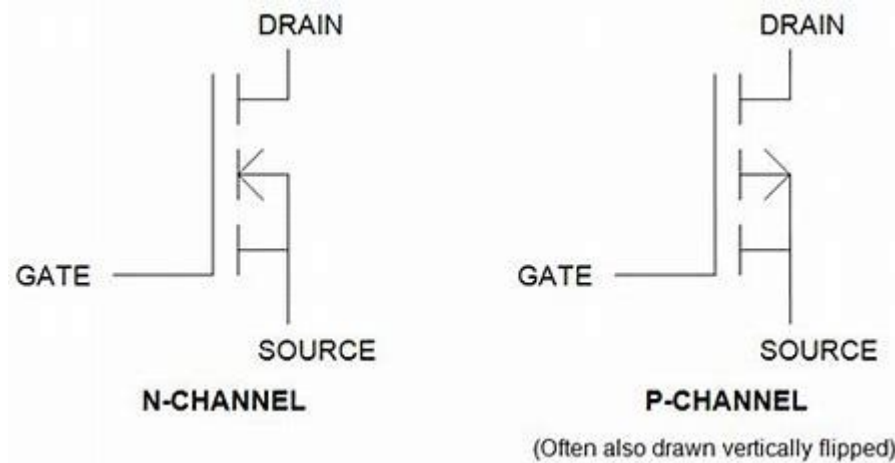


(Like pnp and npn in bjt , they are different in their doping)

(nmos is also called n channel mosfet , due to its internal structure)

MOSFETs for Digital Logic Circuits

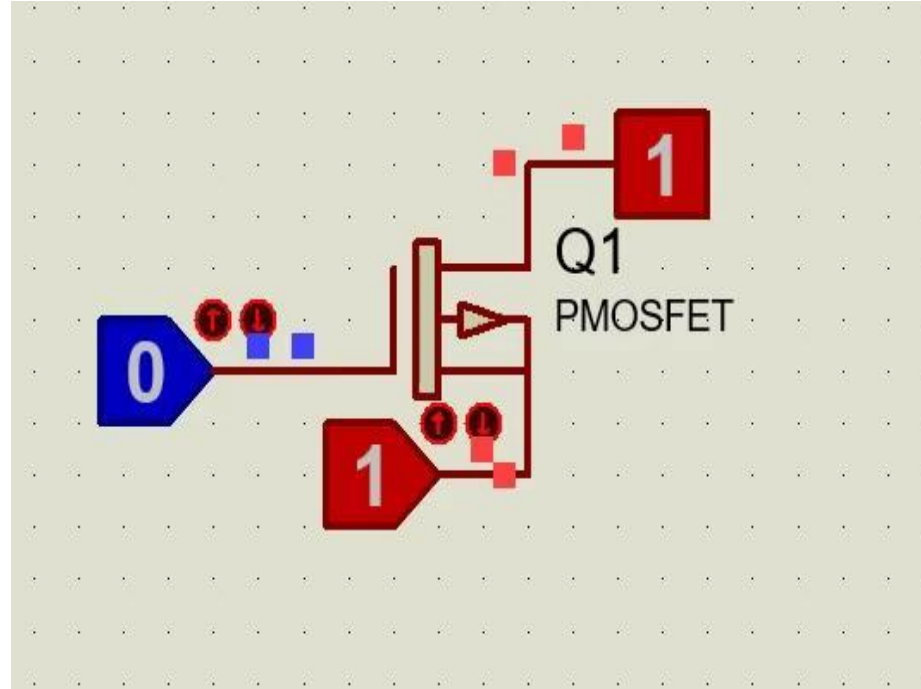
MOSFETs make very very good switches , way better than BJTs



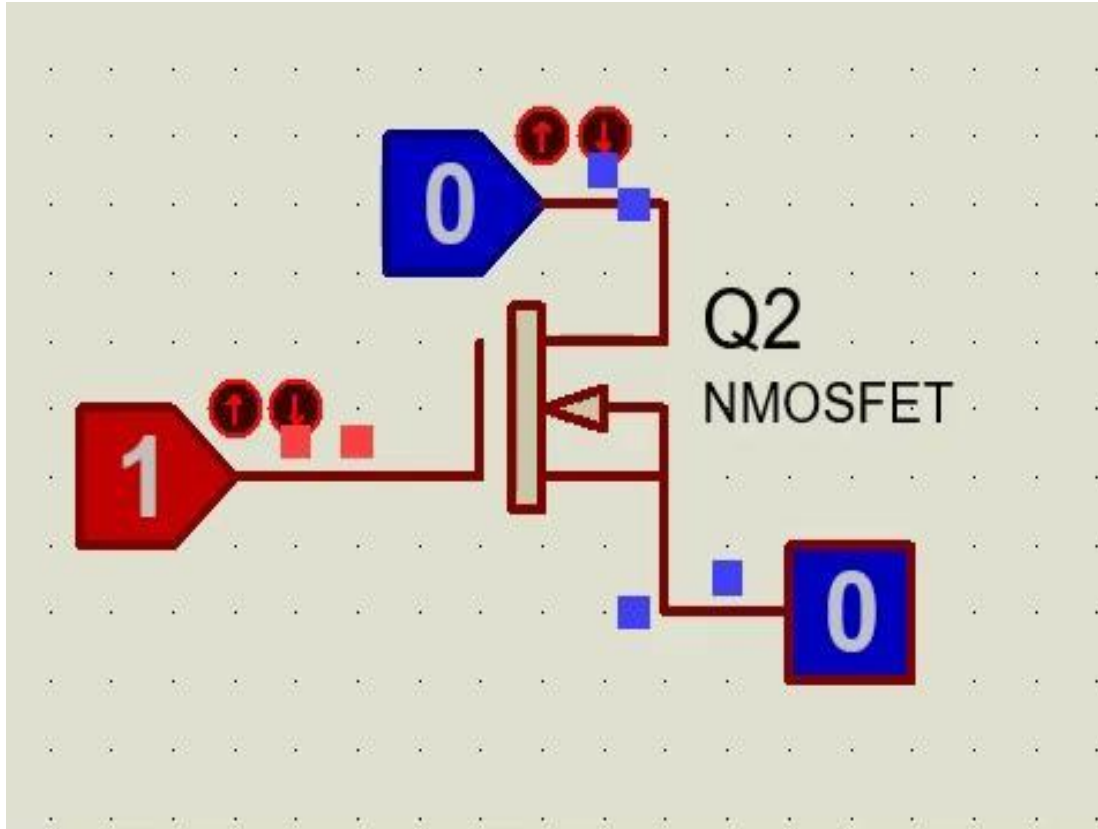
3 terminals , the gate , the source and the drain , source and drain are same

Remember this simple rule

PMOS acts as a CLOSED SWITCH when input at gate is 0 (bubble in its symbol will help in remembering) and OPEN SWITCH when input at gate is 1



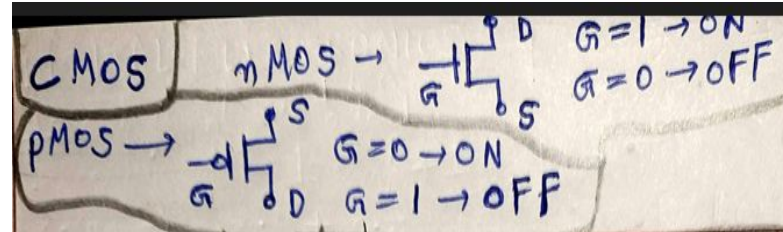
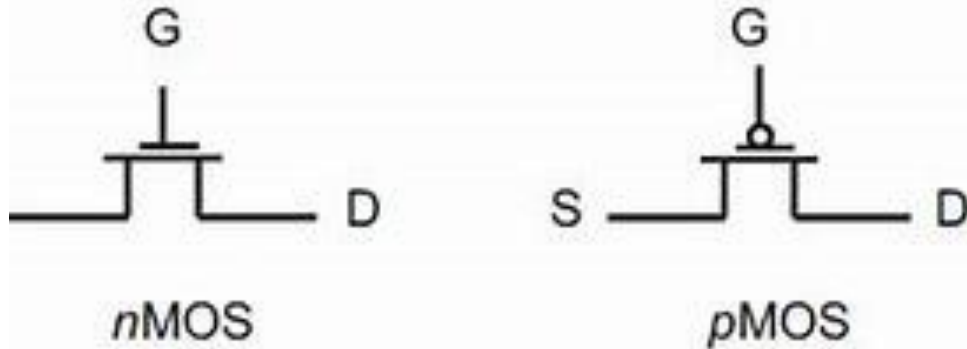
NMOS acts as a CLOSED SWITCH when input at gate is 1 and OPEN SWITCH when input at gate is 0



40 seconds break so that you can remember this

PMOS acts as a CLOSED SWITCH when input at gate is 0 (bubble in its symbol will help in remembering) and OPEN SWITCH when input at gate is 1

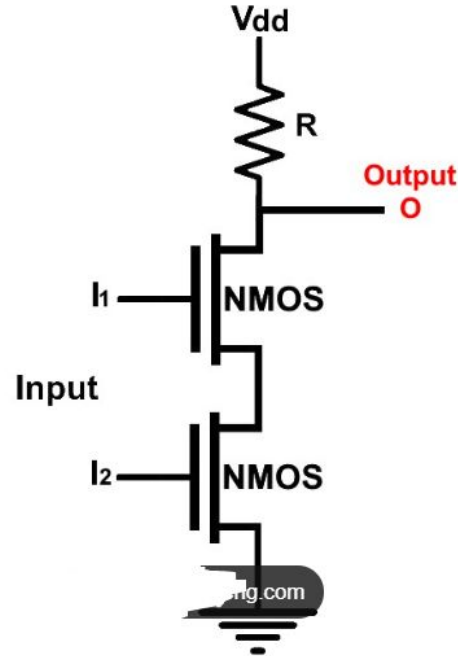
NMOS acts as a CLOSED SWITCH when input at gate is 1 and OPEN SWITCH when input at gate is 0



OK , hope you learnt that info , now lets see how you apply this knowledge

What is this circuit doing ?

Grab your pen and paper , and tell me your answer



Did you get this kind of answer ??

A	B	Z
0	0	1
0	1	1
1	0	1
1	1	0

So , what is this circuit ??

It is a NAND gate !!!!!

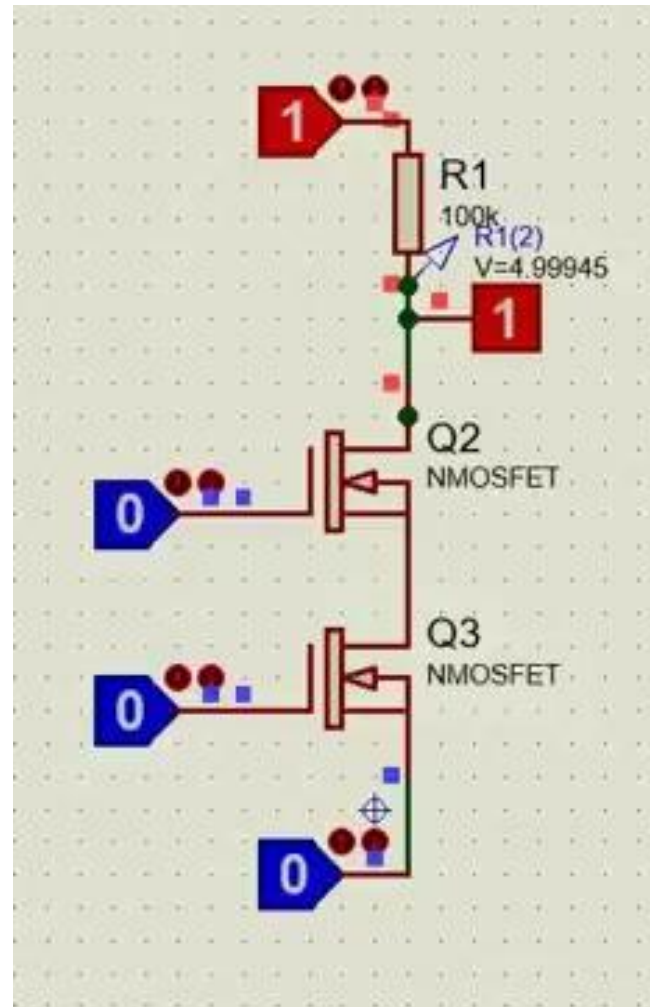
When both A and B are 1 , only then the ground is shorted to the output (the voltage from Vcc dissipates across the resistor) and the output can become 0 , else the output remains 1

If you managed to solve it

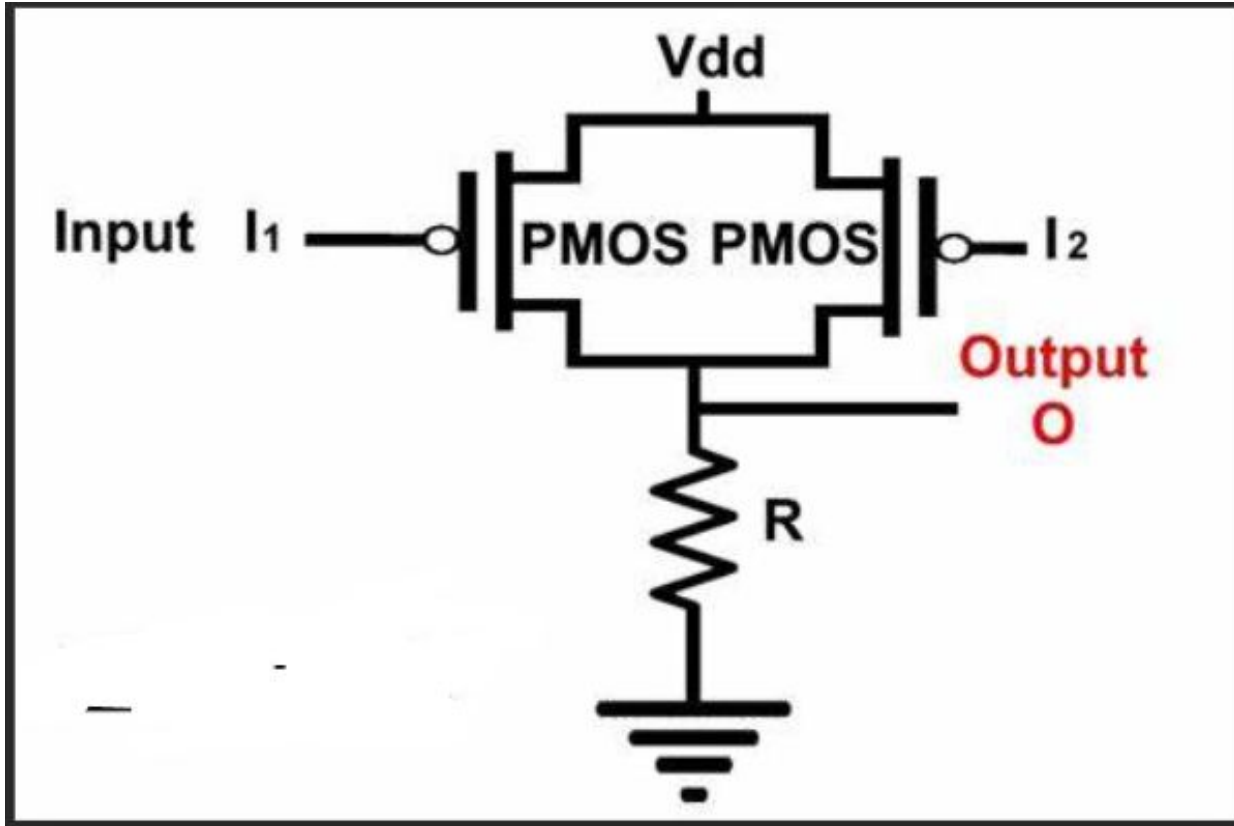


For those who didnt get it

Lets see the proteus simulations :

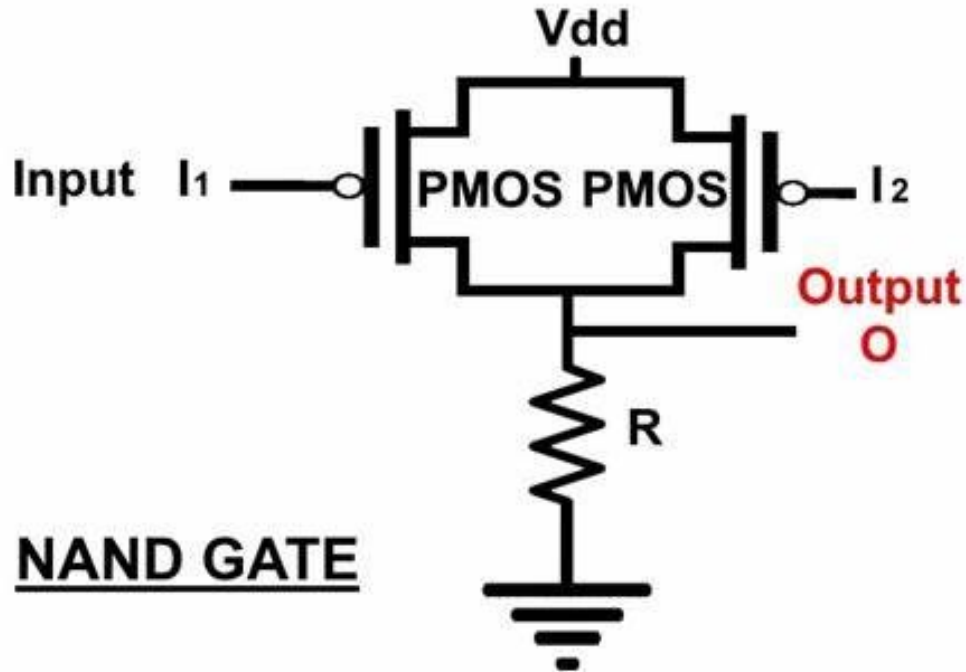


Ok , one more try , what does this circuit do ?



Its a NAND Gate , again !!

And your homework to figure out why if u couldn't



Enter CMOS = PMOS+NMOS

CMOS stands for

COMPLEMENTARY MOS

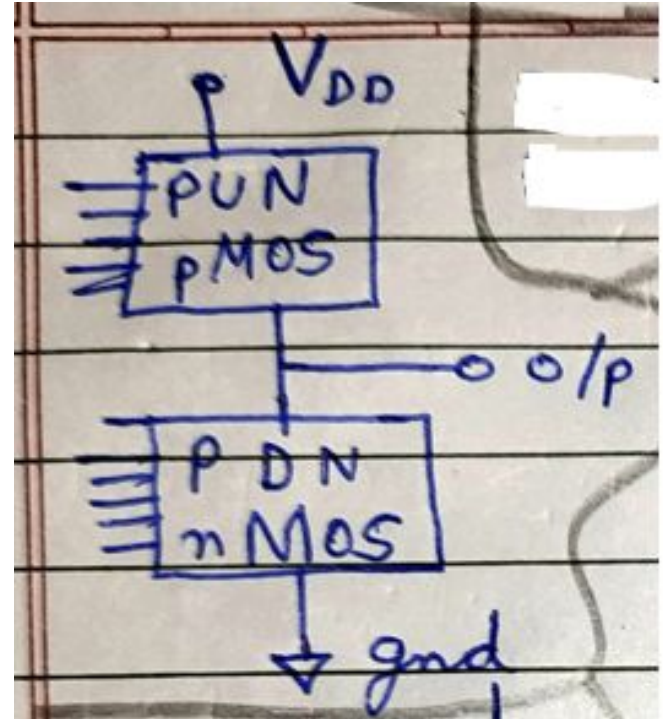
CMOS is a way of making Logic gates using MOSFETS

CMOS has 2 parts , the pull up network PUN and the pull down network PDN

VCC on top , output in middle , ground at bottom

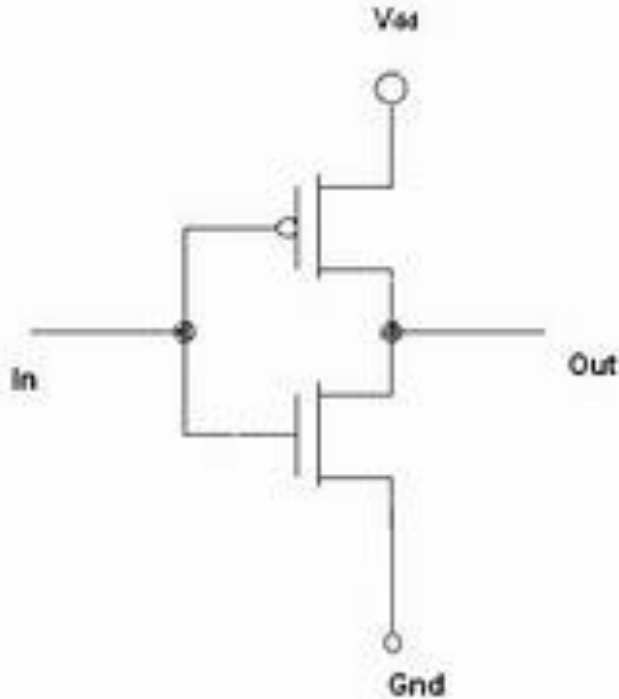
VCC to middle is PUN , middle to ground is PDN

After research , scientists found out that PUN is best made out of PMOS and PDN is best made out of NMOS



The CMOS INVERTER

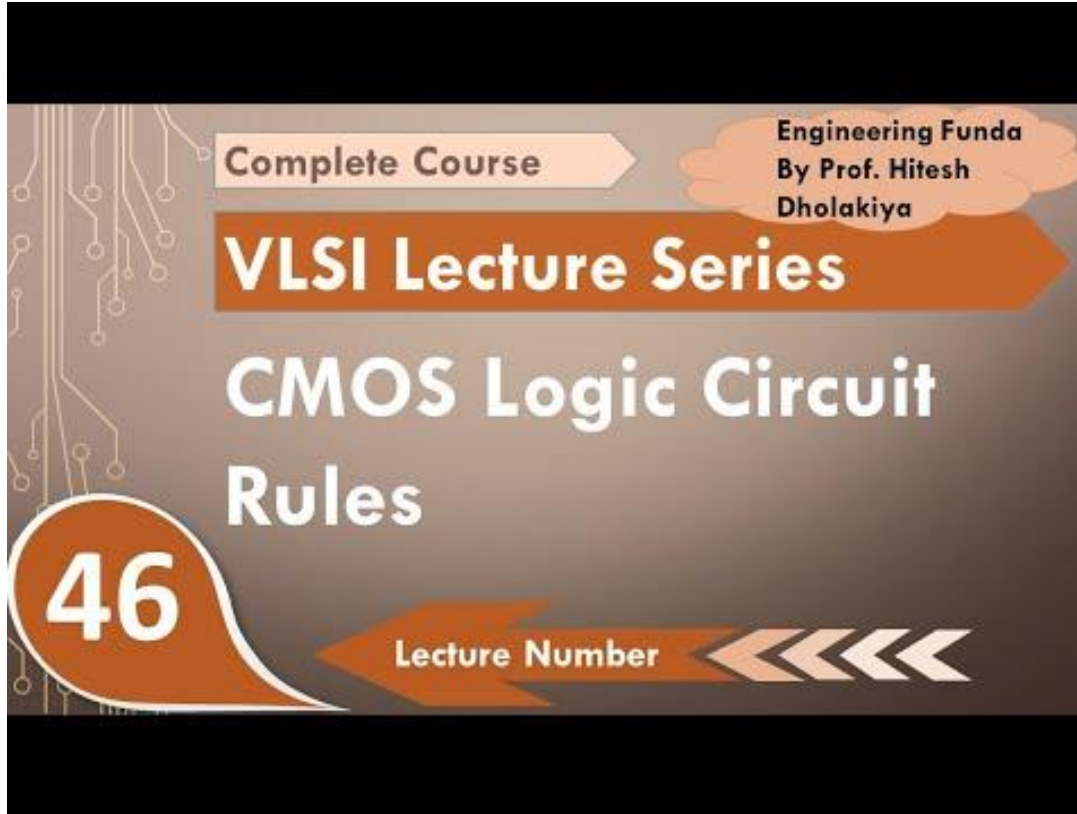
Basic CMOS inverter



Input	Output
A	Y
0	1
1	0

Take some time and understand why this works using your previous knowledge

How to make any boolean logic expression circuit using CMOS? [IMP topic for core intern peeps] [HOMEWORK]



This guy explains best so here is the link :

[CMOS Circuit - YouTube](https://www.youtube.com/playlist?list=PLgwJf8NK-2e4f1EbNBDcft4XxepPfJGvu)

[<https://www.youtube.com/playlist?list=PLgwJf8NK-2e4f1EbNBDcft4XxepPfJGvu>]

Watch first 5 videos of the playlist , 30 min work , easy topic , ek q aayega ispe in intern paper :)

Short summary :

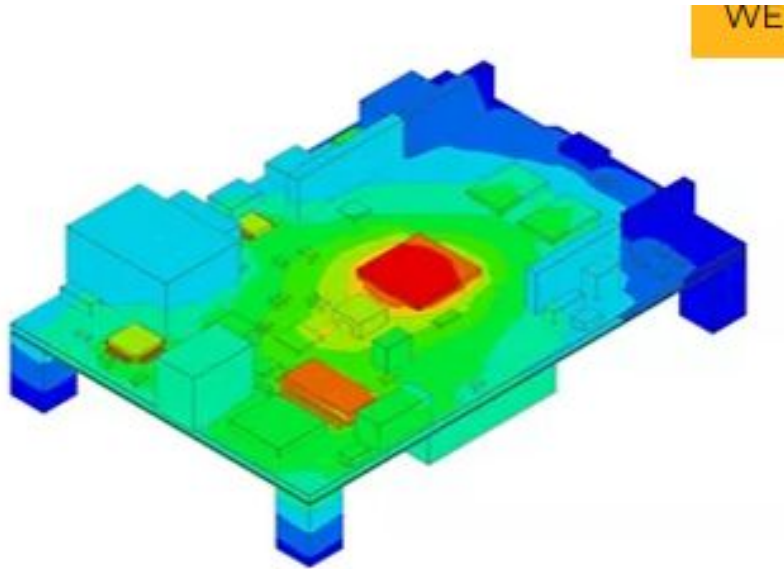
- For Logic circuit $(\cdot) \rightarrow \text{AND}$, $(+) \rightarrow \text{OR}$.
- (\cdot) Operation
 - pmos \rightarrow Parallel
 - nmos \rightarrow Series.
- $(+)$ Operation
 - pmos \rightarrow Series
 - nmos \rightarrow Parallel

Static timing analysis - Setup and hold time violations

SETUP TIME is defined as the minimum amount of time before the clock's active edge that the data must be stable for it to be latched correctly

HOLD TIME is defined as the minimum amount of time after the clock's active edge during which data must be stable.

Thermal analysis of PCBs - Altium and Ansys Icepack [STRICTLY NOT FOR DIGISIM]



[Ansys Icepak | Electronics Cooling Simulation Software](#)

Thermal analysis of PCB designs is done in order to figure out temperature variations across the PCB and to make sure that the PCB design doesn't melt during working, and that the heat is safely disposed off. This work is less significant to ECE people and more handled by other engineers, who are more proficient with aerodynamics and stress strain analysis of materials in CAD.

PCB Printing machine , and facilities in our college

Available at Precision
engineering Hub (PEH)
in IIT BHU Varanasi , and
in ECE dept too



ASIC , SOC and FPGA - terms an industrial electronics engineer should know

ASIC = Application Specific Integrated Circuit

Circuit made for a specific application

SOC = System on Chip

A common term for designs involving microprocessors

FPGA = Field Programmable Gate Array

Board on which logic gates can be programmed using verilog

Debugging

To remove all the routing on the board - “ Ripup ; “ on the command line

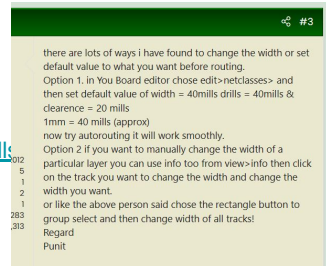
To remove an existing autorouter job [Can't stop the autorouter - EAGLE User Support \(English\) - Autodesk EAGLE - element14 Community](#)

Changing width in autorouter

Copper pour - using polygon and ratnest

changing gap between line and copper pours - [HOW-TO: Polygons and ground fills Prototypes](#)

To write name - layer - tvalues then text



To work on the bottom layer of the PCB

Flip the board

Then [Solved: how to move component to bottom layer? - Autodesk Community - EAGLE](#)

Some resources -

Amit rana udemy course (watch the half free part) - [Learn the Art and Science of PCB Design with Eagle | Udemy](#)

Another eagle cad tutorial -

<https://www.youtube.com/playlist?list=PL868B73617C6F6FAD>

Physical design course (contact me to access its content , only possible around 14-20 march) [Learn Physical Design Flow for Very Large Scale Integration \(VLSI\) | Udemy](#)

Author

Ayush Agarwal (ECE'24 IIT BHU)

<https://www.linkedin.com/in/ayush-agarwal-261041215/>

<https://github.com/ayush-agarwal-0502>