ELECTRONICS CLUB

Centre For Innovation

ELECTROMANIA

ELECTROVERSE OF MADNESS

PCB Design DAY - 1 : Introduction to PCB design

ELECTRONICS CLUB

Centre For Innovation

What will we be doing in the following sessions ?

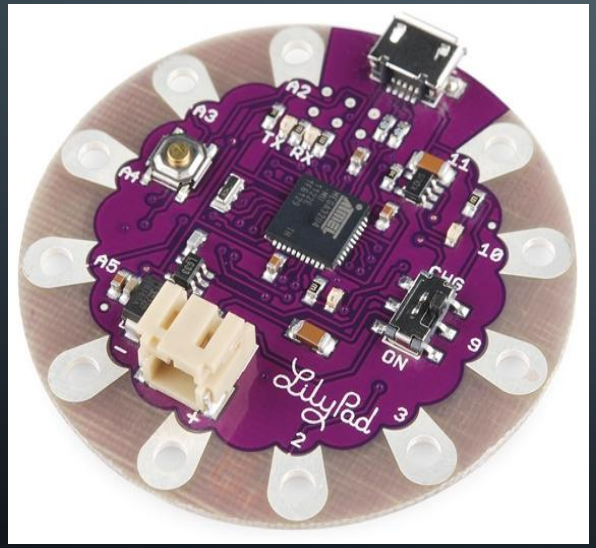
1. **Move away, Newbs**
   Intro to PCB Design- 555 Timer circuit
2. **Seizure Alert!!**
   PCB Design - Day 2 - POV (Persistence of Vision) Fidget Spinner (schematic)
3. **Hangover - Part 2**
   PCB Design - Day 3 - POV Fidget Spinner (board layout)and Gerber Files

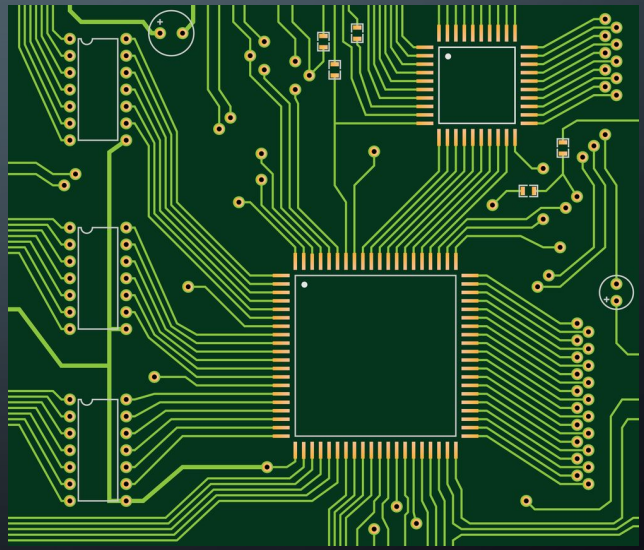
ELECTRONICS CLUB

Centre For Innovation

What is a PCB ?

PCB is an acronym for printed circuit board. It is a board that has lines and pads that connect various points together. In the pictures below, there are traces that electrically connect the various connectors and components to each other.





ELECTRONICS CLUB

Centre For Innovation

Why PCB ?

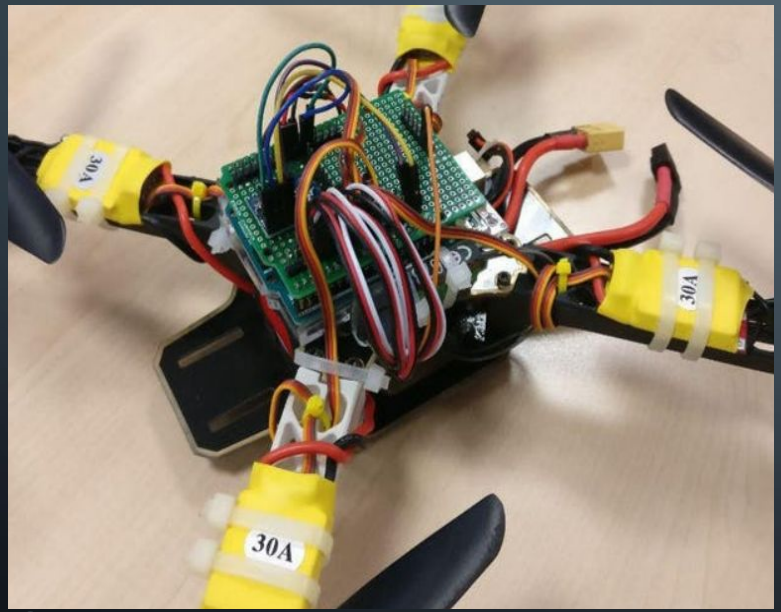
While it is possible to use Arduino boards of different sizes(small to large) they will never be the “perfect” size for our product. Using jumper wires makes it look even more messy. There are many projects/products where size isn’t an issue, other times a small form factor is crucial. Designing a custom PCB allows for very small projects, as it is purpose-built for that application.

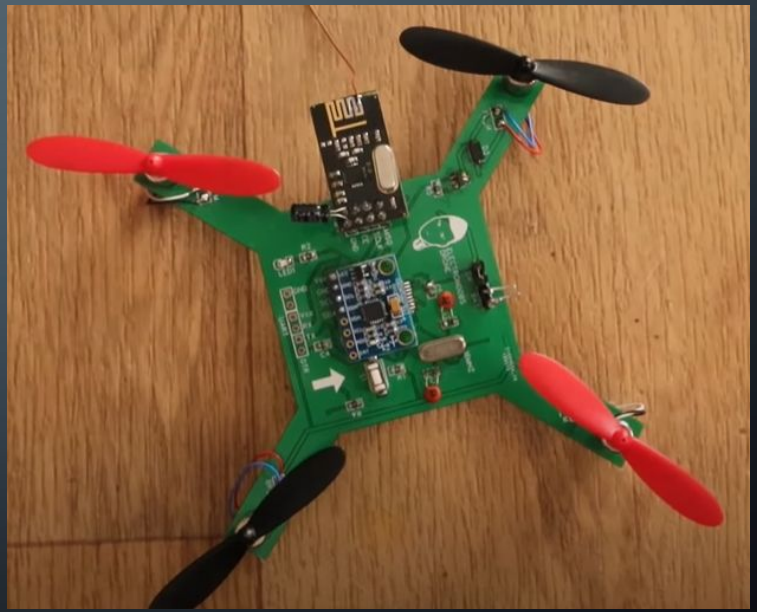
ELECTRONICS CLUB

Why are PCBs better than using already existing MCU

Centre For Innovation

On the right we have a drone shaped PCB with holes in it for the fitting of motors(The level of customisation…) And all the wiring in left image is done within the board itself(Yayy…No jumper wires). This makes the drone a lot lighter, and hence low power motors are enough to keep them Zooming!!!



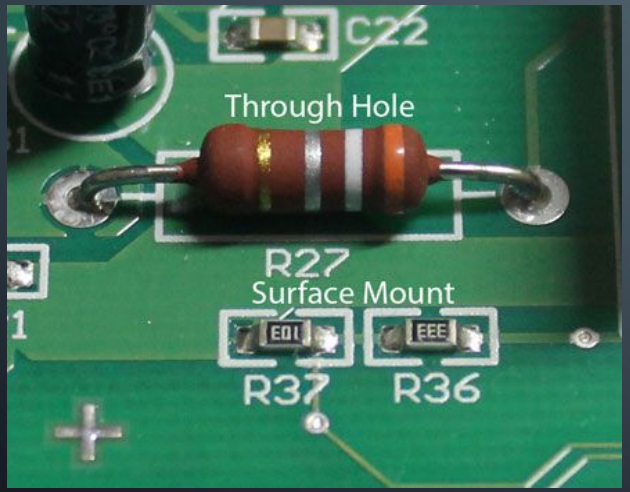


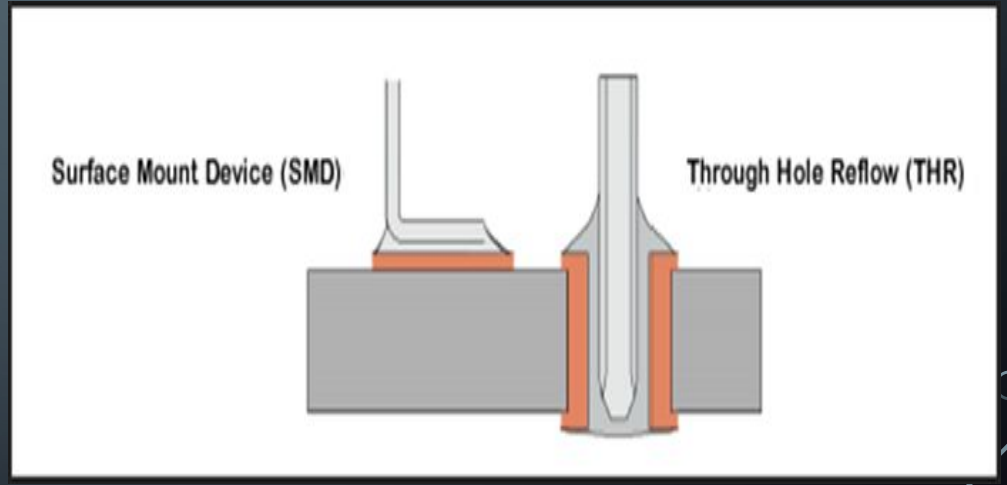
ELECTRONICS CLUB

Centre For Innovation

What are the types of COMPONENTS ?

Surface mount (SMD) and Through Hole (THR)



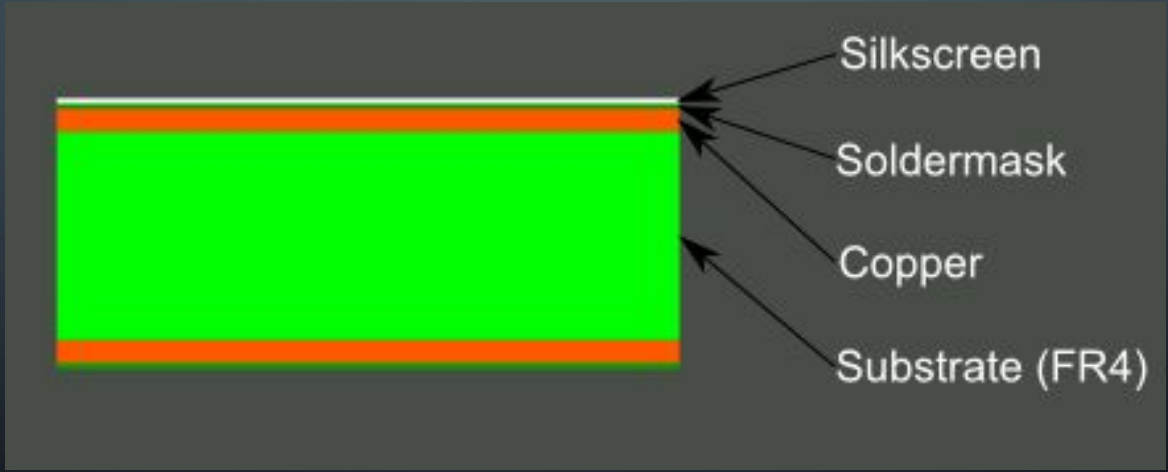


ELECTRONICS CLUB

Centre For Innovation

What is inside a PCB 🧐?

PCB is made of several layers each having its own purpose. Usually the layer order is mirrored on either side of the substrate. The layers are laminated using heat and adhesives.
We will see in brief about each layer.

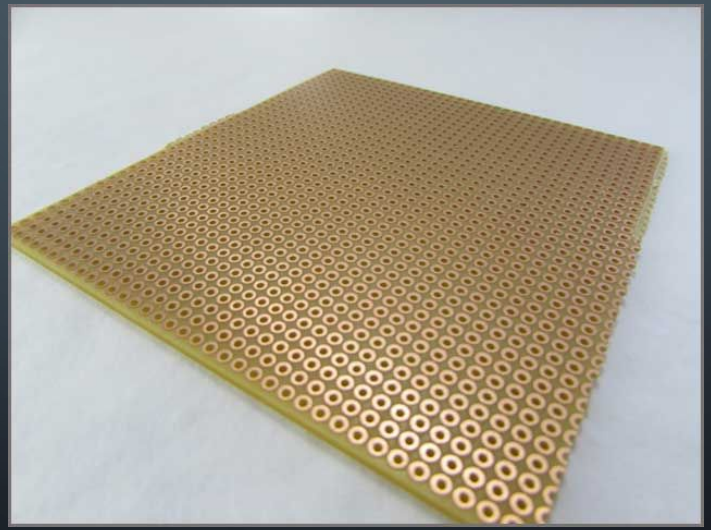


ELECTRONICS CLUB

Centre For Innovation

(i) Substrate

The base material, usually fiberglass is called the substrate. This solid core gives the PCB its rigidity and thickness. Usually the material named FR4 is used. The usual thickness of a PCB is 1.57mm or 0.063”, which mostly is the substrate.

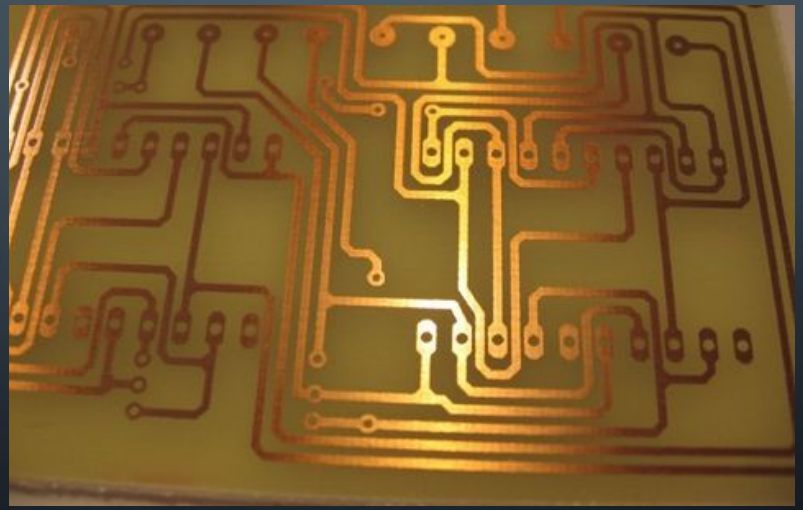


ELECTRONICS CLUB

Centre For Innovation

(ii) Copper

The next layer is a thin copper foil, which is laminated to the board with heat and adhesive. On common, double sided PCBs, copper is applied to both sides of the substrate.When we refer to a double sided or 2-layer board we are referring to the number of copper layers (2) in our PCB.

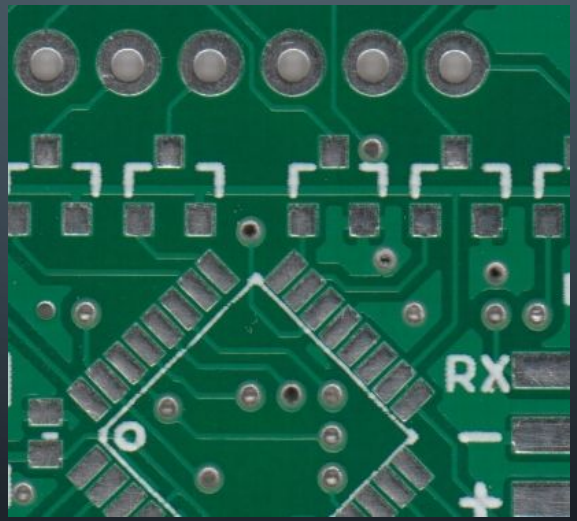


ELECTRONICS CLUB

Centre For Innovation

(iii) Soldermask

The layer on top of the copper foil is called the soldermask layer. This layer gives the PCB its iconic green color. It is overlaid onto the copper layer to insulate the copper traces from accidental contact with other metal, solder, or conductive bits. This layer helps the user to solder to the correct places.

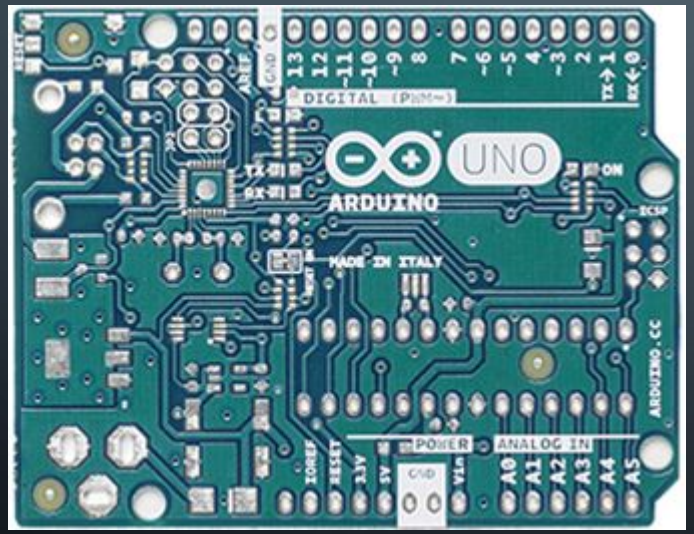


ELECTRONICS CLUB

Centre For Innovation

(iv) Silkscreen

The white silkscreen layer is applied on top of the soldermask layer. The silkscreen adds letters, numbers, and symbols to the PCB that allow for easier assembly and indicators for humans to better understand the board. (It makes life easyy for us…)



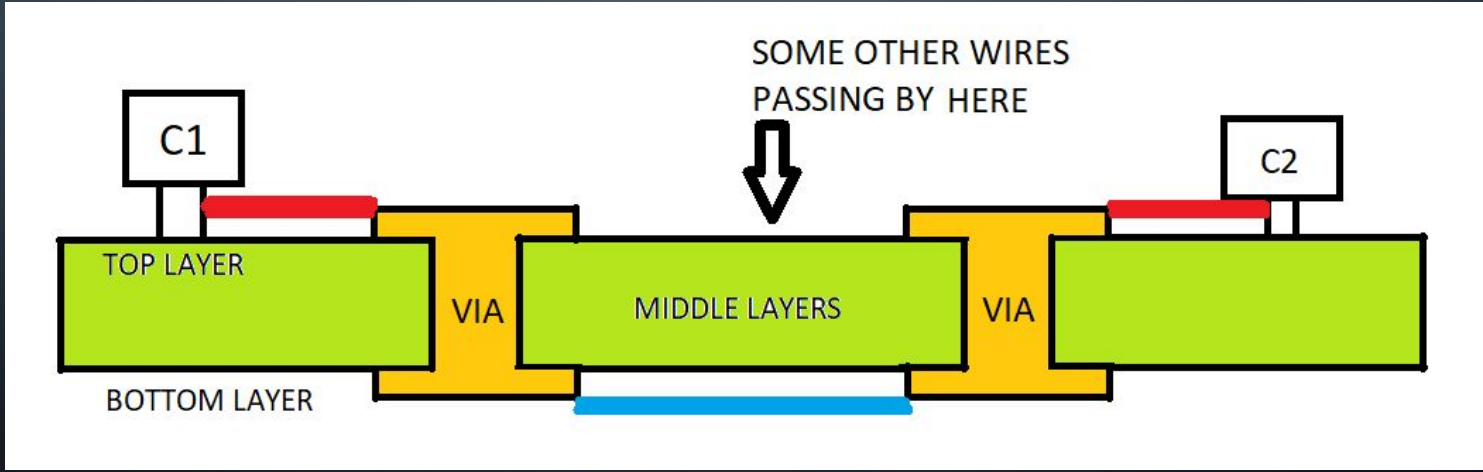
As we all know this is the famous Arduino's Silkscreen

ELECTRONICS CLUB

Centre For Innovation

(v) Via

A Via is a connection method most commonly used in multi-layer PCBs. When one layer is already congested with enough wires, we can use the bottom layer for the connection too. This is how Via works :



ELECTRONICS CLUB

Platforms for PCB Designing

Centre For Innovation

- Eagle
- KiCad
- Altium
- EasyEDA

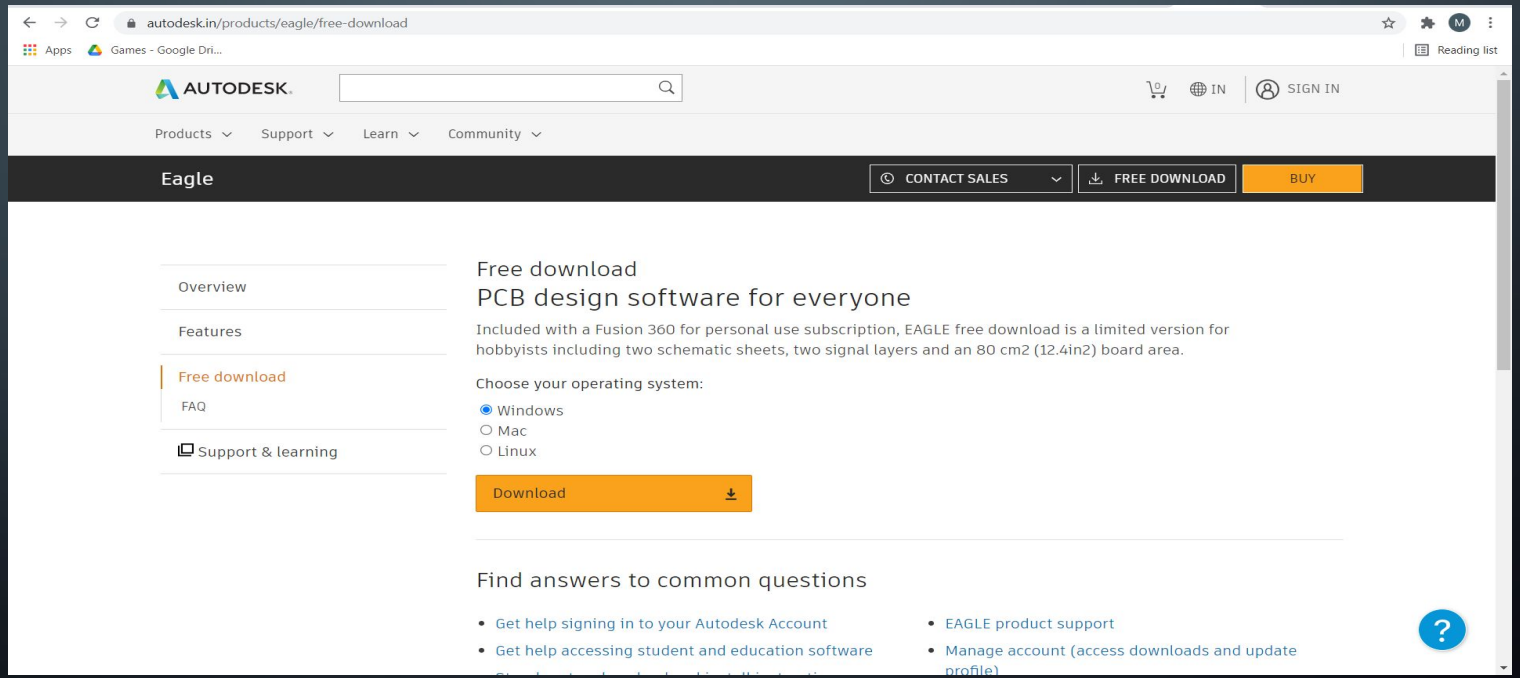
We will be using **EAGLE** platform for PCB designing.

ELECTRONICS CLUB

Installation

Centre For Innovation

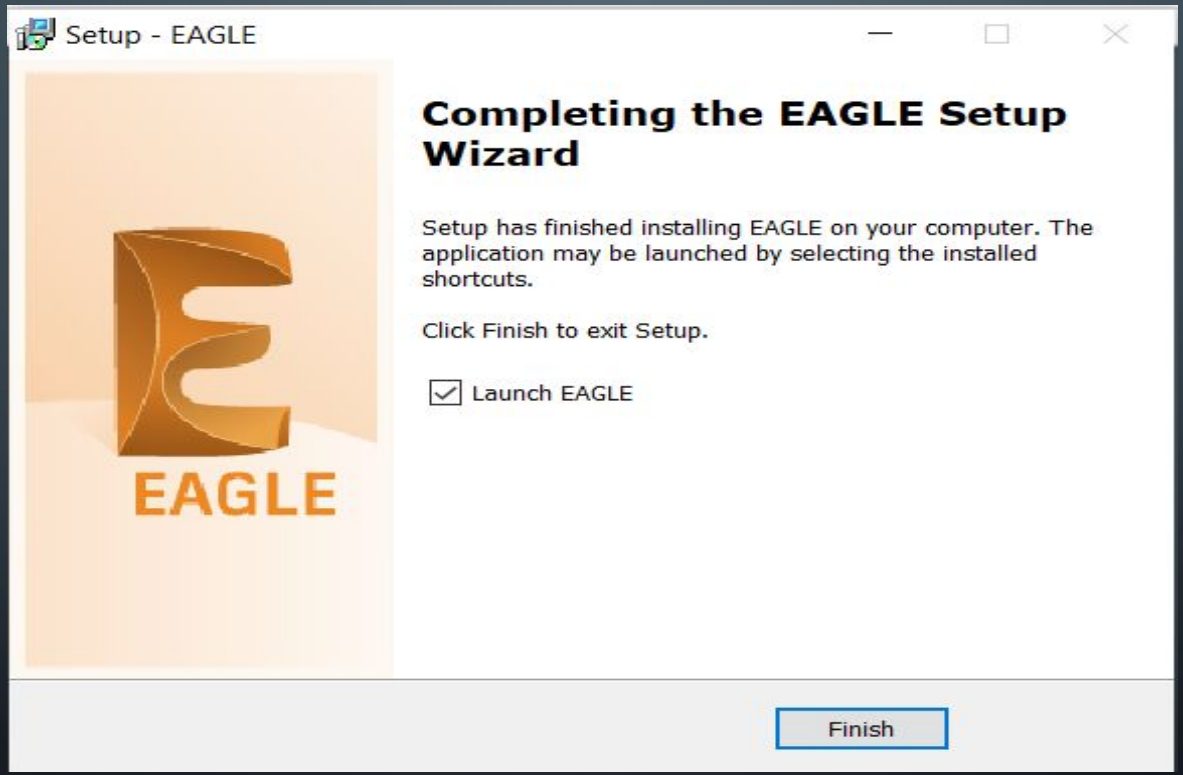
Download EAGLE setup from Autodesk. Follow the instructions to install it.
It is free for insti students, if registered with smail id.



ELECTRONICS CLUB

Centre For Innovation

Once the installation is complete.You are ready to launch the platform!!!!

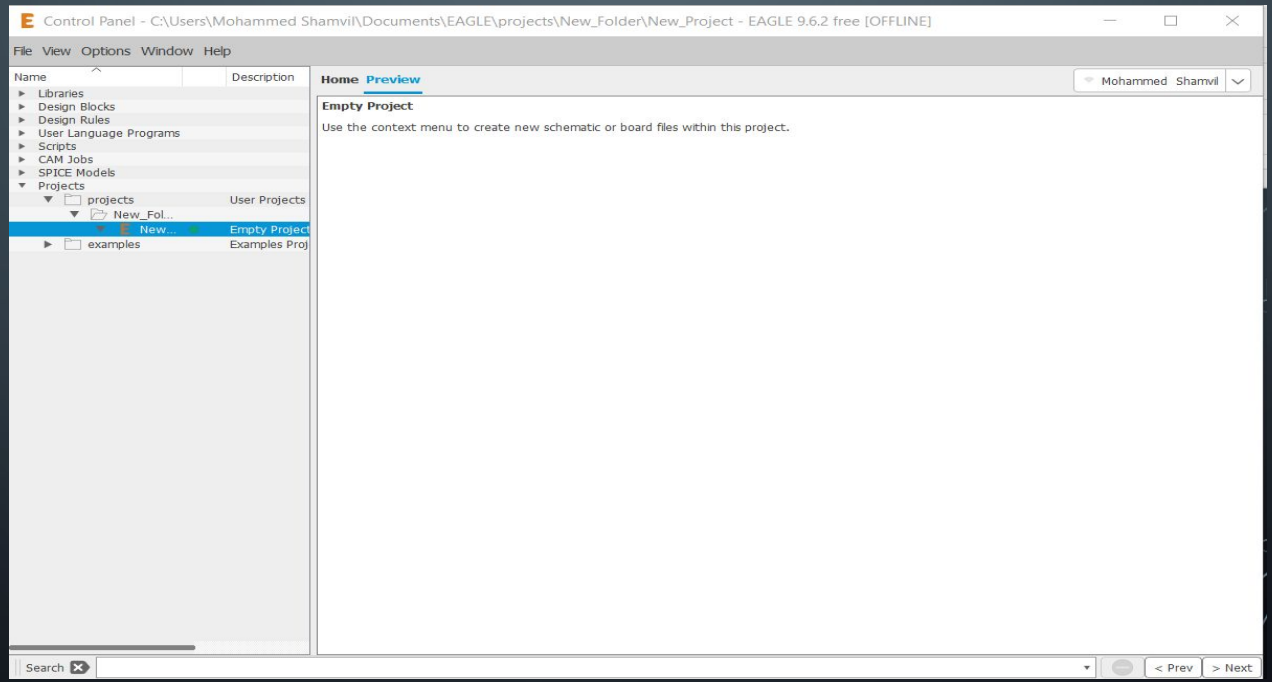


ELECTRONICS CLUB

Let's switch to the Eagle platform!!!!

Centre For Innovation

And learn the basic UI and tools of Eagle !

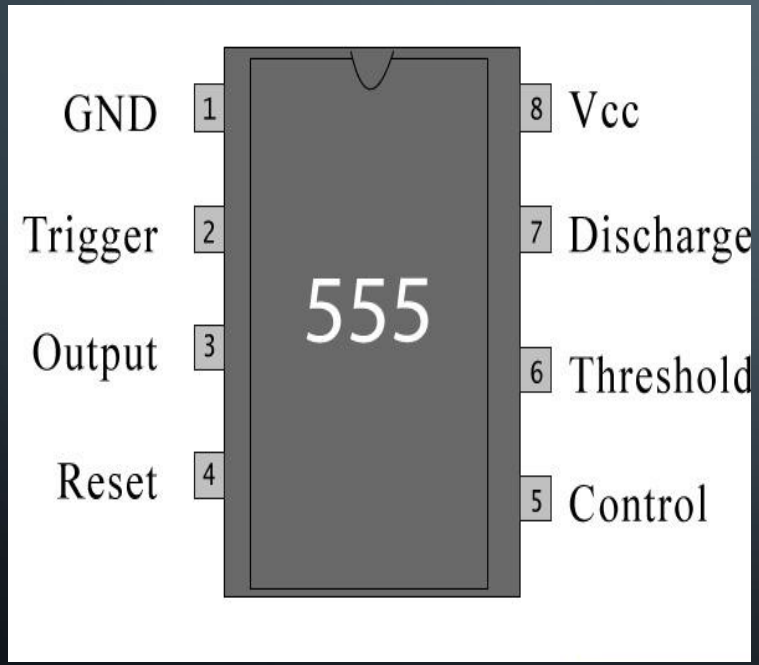


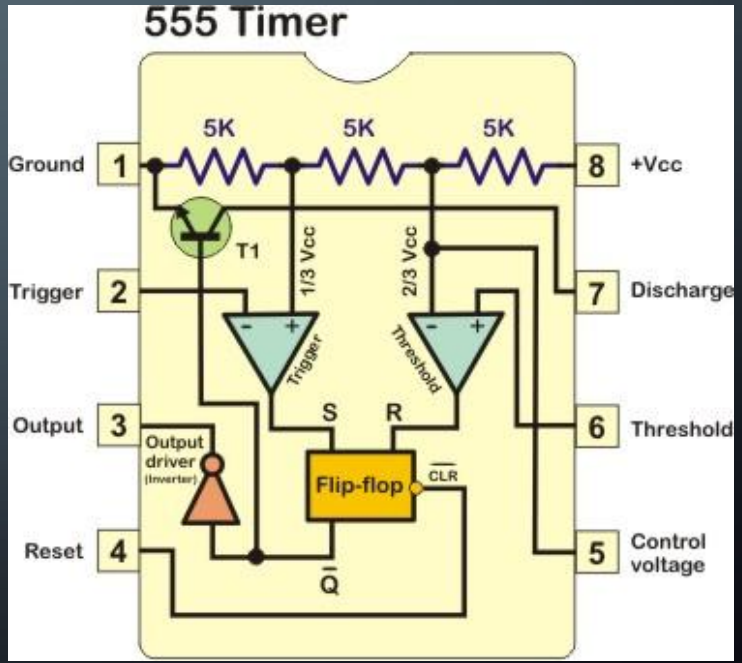
ELECTRONICS CLUB

Centre For Innovation

555 Timer IC

The 555 timer IC is an integrated circuit(chip) used in a variety of timer delay,pulse generation and oscillator applications.





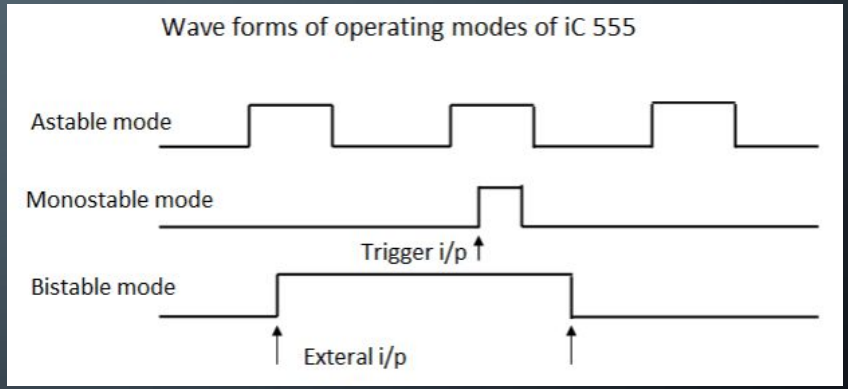
ELECTRONICS CLUB

Centre For Innovation

Different Modes of Operation

The 555 timer IC has three main operating mode

- Astable Mode
- Monostable Mode
- Bistable Mode(Schmitt Trigger)



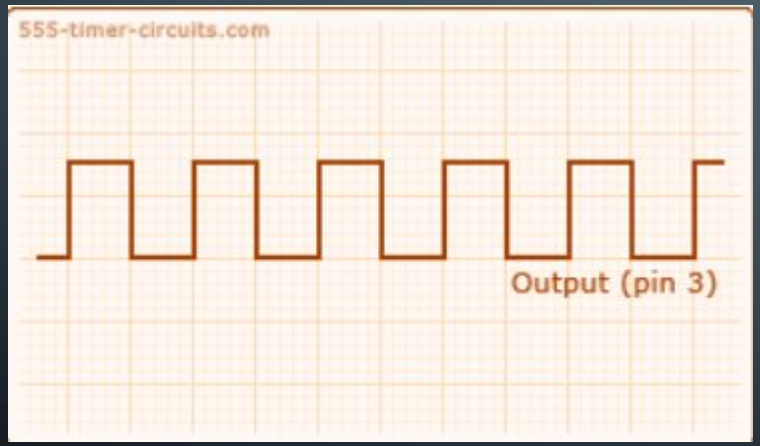
We will be seeing the output of each mode briefly

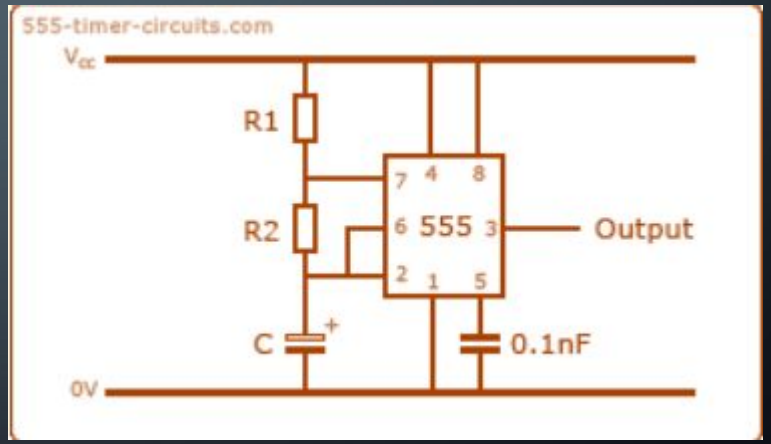
ELECTRONICS CLUB

Centre For Innovation

Astable Mode

This mode has no stable state.The output continually switches between high and low without any intervention from user called a “Square Wave”.



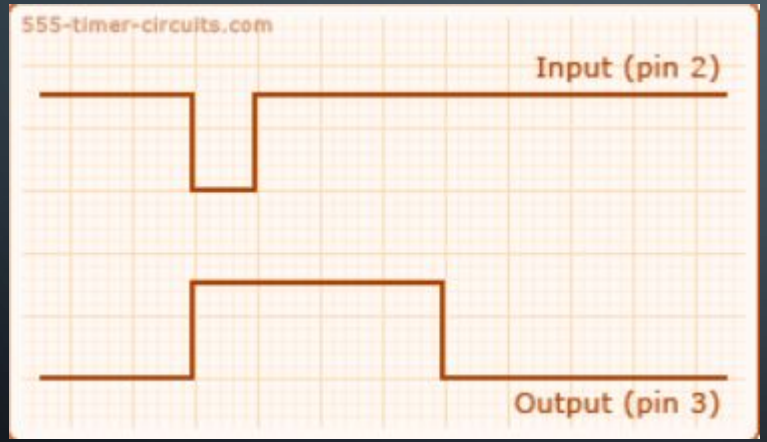


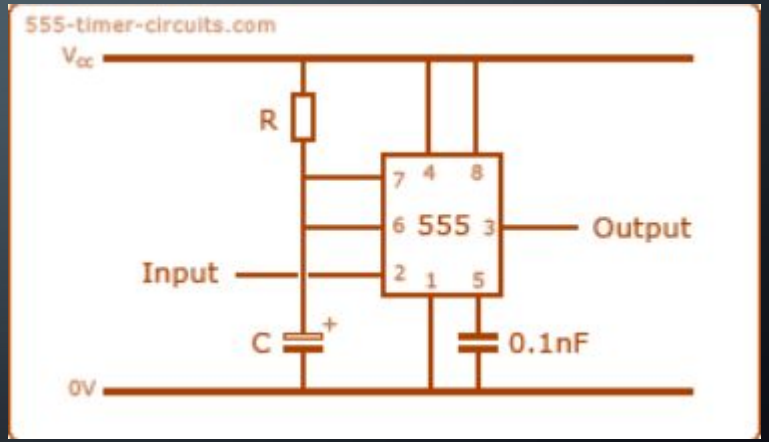
ELECTRONICS CLUB

Centre For Innovation

Monostable Mode

A Monostable Circuit produces one pulse of a set length in response to a trigger input such as a push button. The output of the circuit stays in low state until there is a trigger input hence the name Monostable meaning one stable mode.



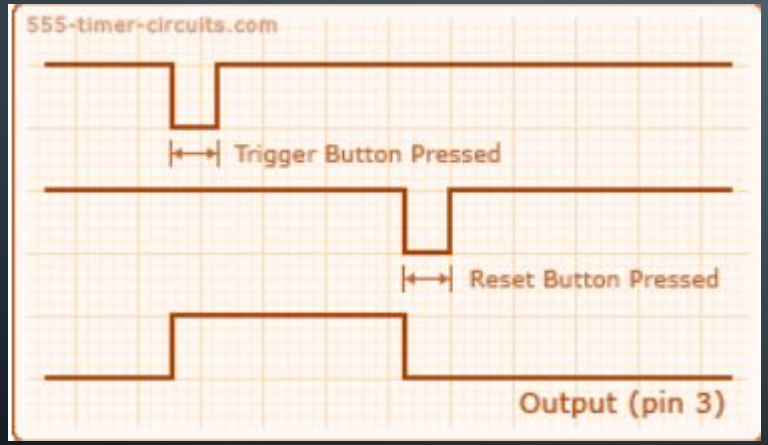


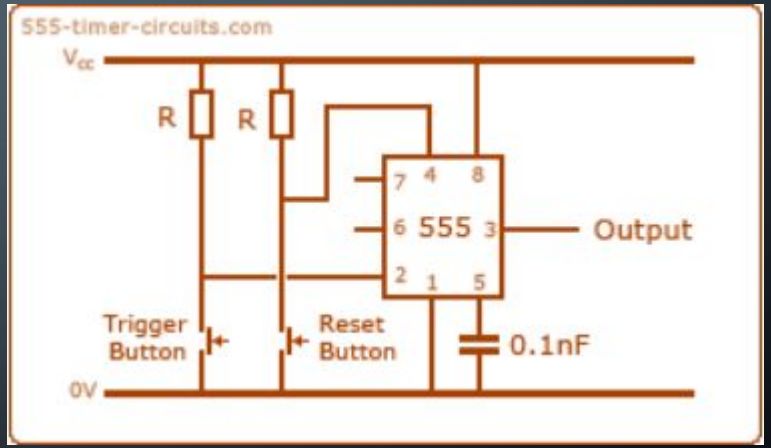
ELECTRONICS CLUB

Centre For Innovation

Bistable Mode

A Bistable Mode(or a Schmitt Trigger) has two stable states high and low.Taking the Trigger input low makes the circuit go into the high state.Taking the Reset input low makes the circuit go into the low state.



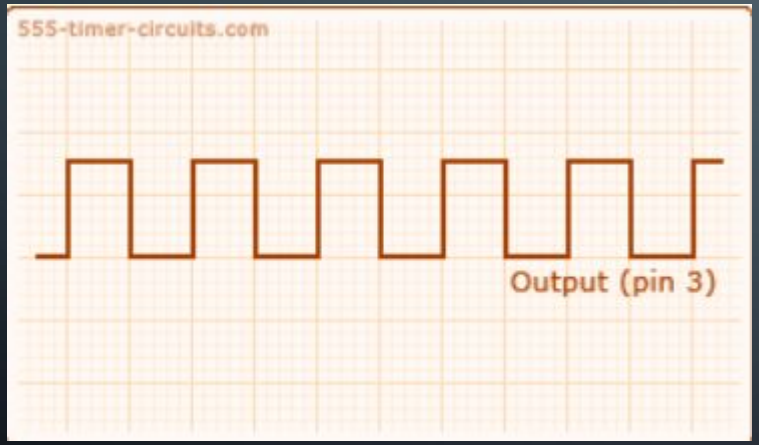


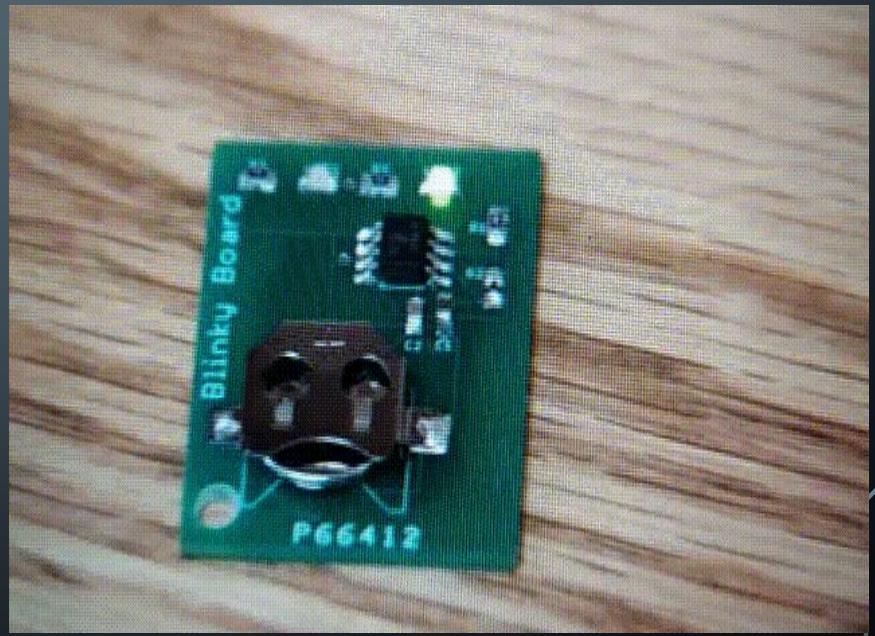
ELECTRONICS CLUB

Centre For Innovation

Today we will be working on the schematic of the Astable mode of the 555 Timer circuit

The final result will be blinking of two LEDs each one for the low and high of the wave output.





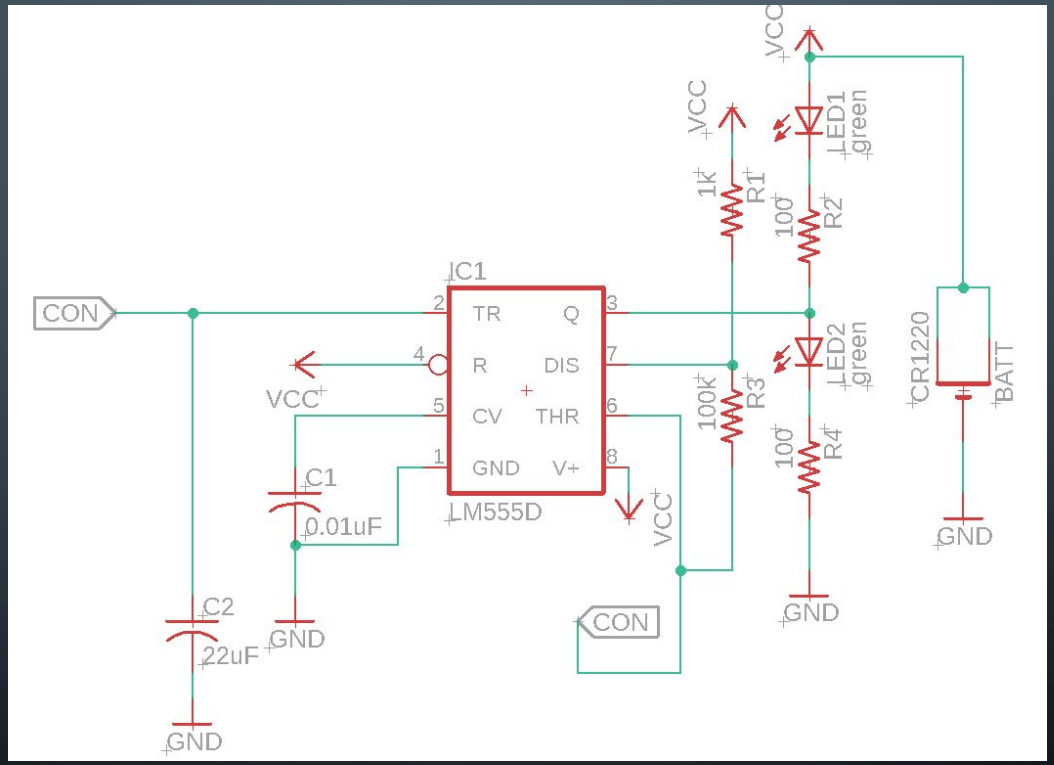
Before we move to the Schematic drawing for Blinky Board Project ,we need to have Adafruit external library

You can get the Adafruit library from the given link:
https://github.com/adafruit/Adafruit-Eagle-Library/zipball/master

ELECTRONICS CLUB

Centre For Innovation

Now let's switch to EAGLE to make our first schematic!



ELECTRONICS CLUB

Centre For Innovation

Thank you guys So much for attending the session

We'll meet again tomorrow on Day-2!!

Mohammed Shamvil
CH20B066

Arun Palaniappan
ME20B036