



# ELECTROMANIA

---

ELECTROVERSE OF MADNESS

**PCB Design DAY - 2:** Manual PCB Routing, Fun QUIZ and  
Persistence of Vision(Fidget Spinner)

Before jumping into Manual routing, **Know your rules!**

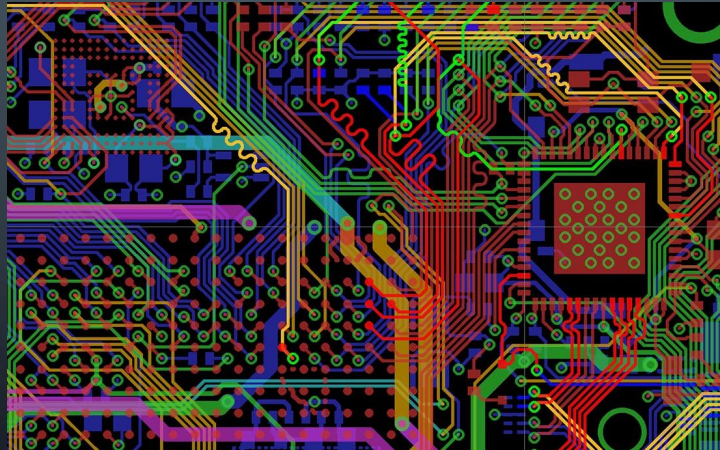
There are certain rules to be followed in the following aspects so that your **board** lasts long!



## 1) Don't rely on Auto-Router

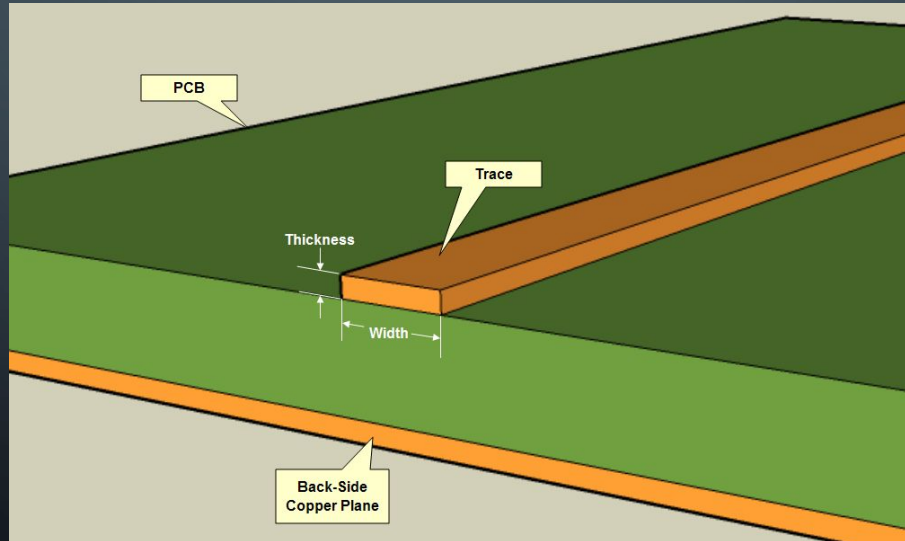
Almost Every PCB Design software has this fancy feature called **Auto-Route**. This feature automatically finds the connection routes and finishes the board for you. But it doesn't follow the guidelines like changing the width of the wire for high current connections.

Moreover you should be the master of your board and not some auto... feature. The satisfaction you get when you craft a masterpiece like this 😊.

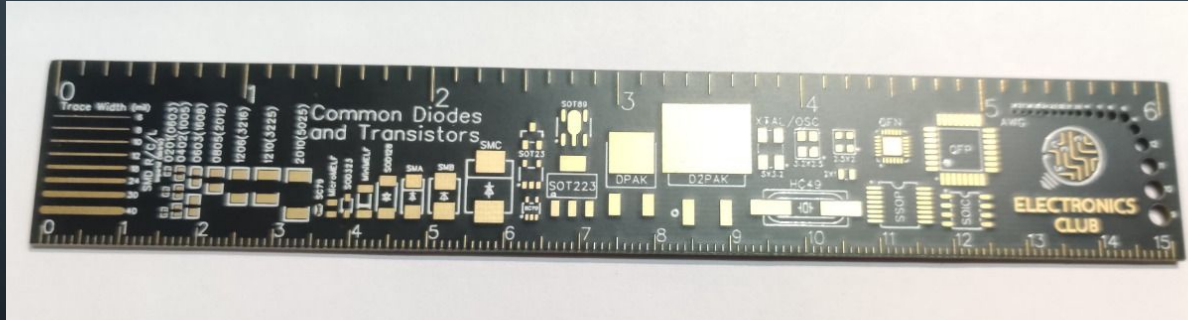
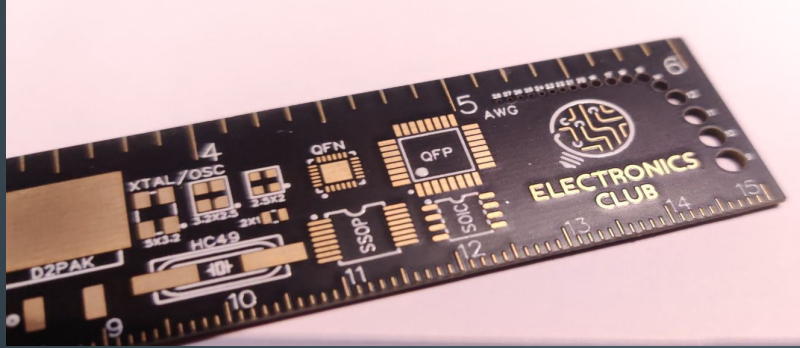


## 2) Figure Out Your **Trace Widths**

When you have electricity cruising through all of your copper traces, it's going to produce some serious heat, and this is always a concern with electronics. Controlling the **width of your traces** is one of many ways to reduce how much heat builds up on your board, and the wider your traces, the less resistance that electricity will encounter when flowing through your circuit.



We've got [Electronics Club](#) merch for That!

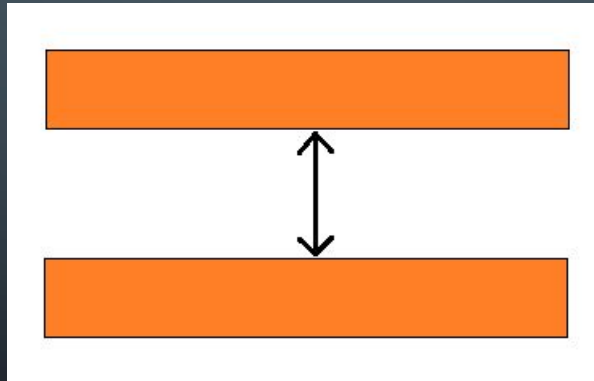


This is a custom ruler made by [Electronics club](#) to find common trace widths and RLC, IC components



### 3) Leave Enough Space Between Traces

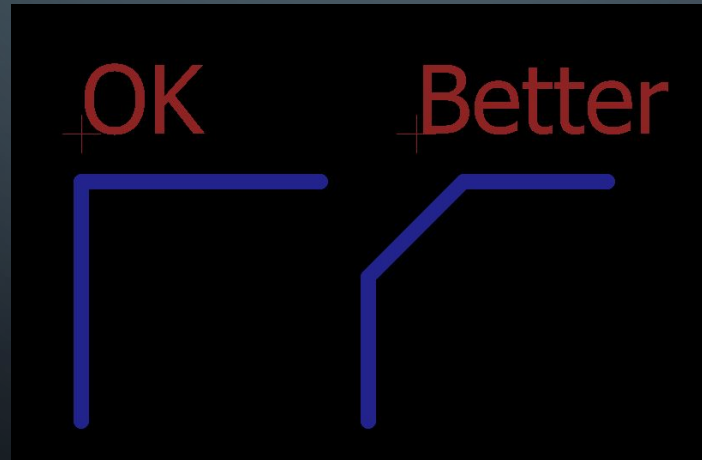
It's important to leave enough room between all of the traces and pads on your PCB layout. Why? If you pack things too close together, then you'll run the risk of a short circuit developing when your board gets manufactured and traces unintentionally can connect. Recommended distance to leave between your pads and traces is **6 mil -10 mil (0.006" to 0.010")**.



#### 4) Avoid Using 90 Degree Trace Angles

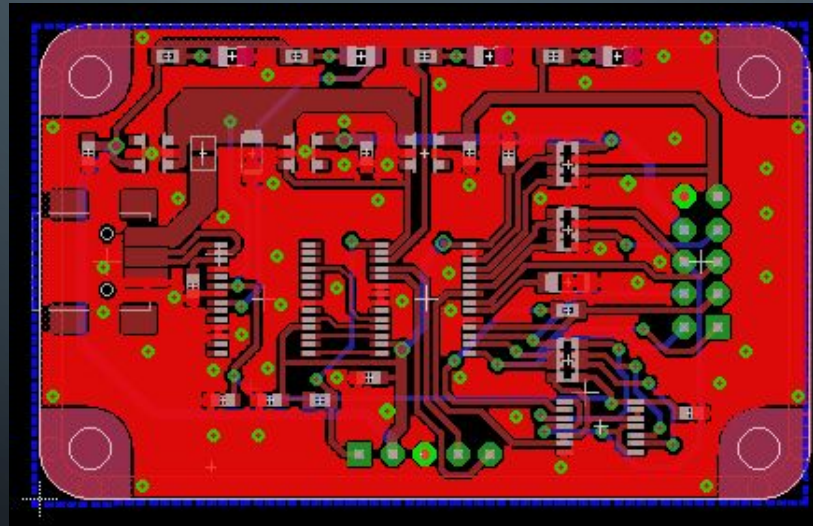
90 degree traces are very tough to etch on the board and the machine might end up etching the outside corner of that 90-degree angle narrower than your standard trace width.

Another problem is that 90 degree sharp turns can cause high accumulation of voltage at that point causing dielectric breakdown. (This occurs only in very high power circuits and does not affect our low voltage PCBs much). USE 45°



5) Create a **ground** plane for all the connections to the ground.

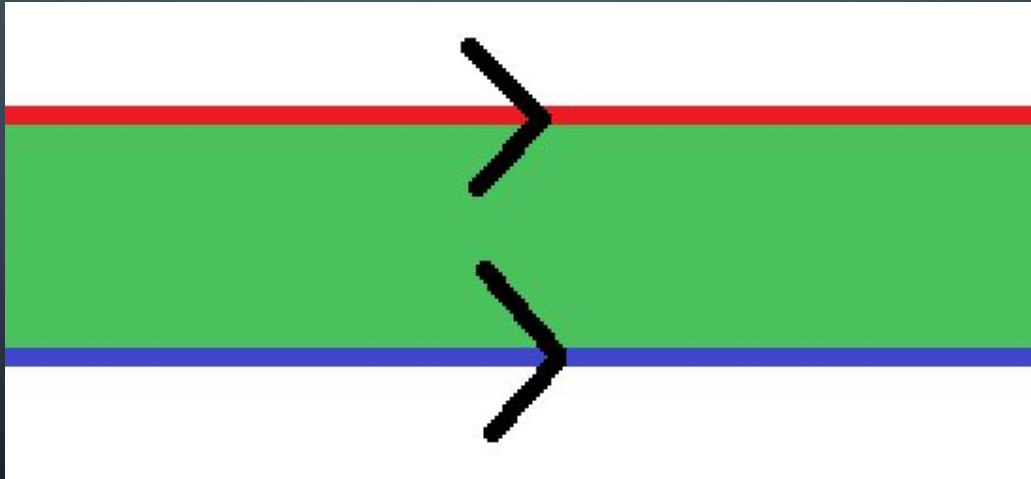
This will reduce the complexity of routing and will reduce heat dissipation significantly. This is a very good practice.





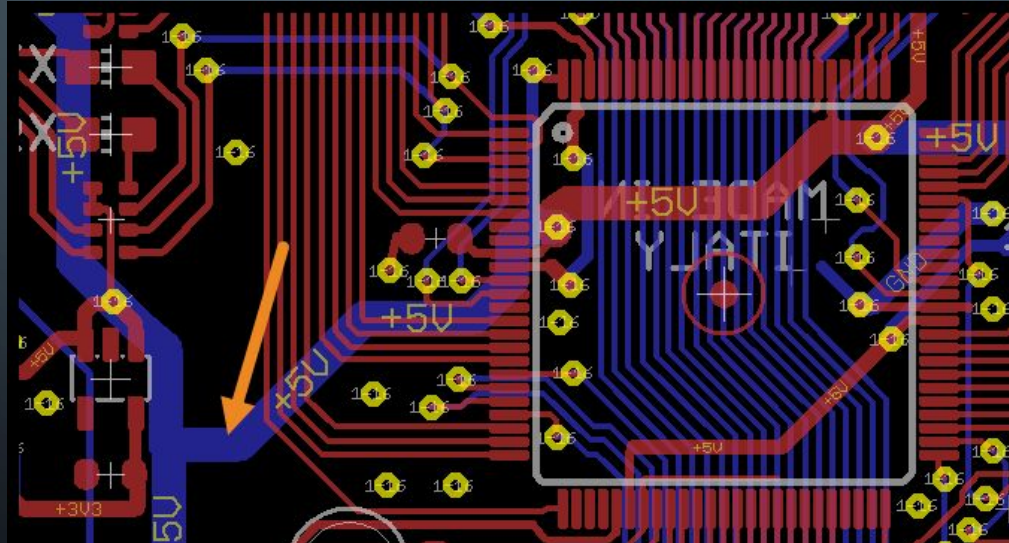
6) Do not have parallel connection in **Top** and **Bottom** layer.

Having parallel connection in **Top** and **Bottom** layer will kind of create a capacitance and affect the circuit performance.



## 7) Make Your **Power** and **Ground** Traces Wider

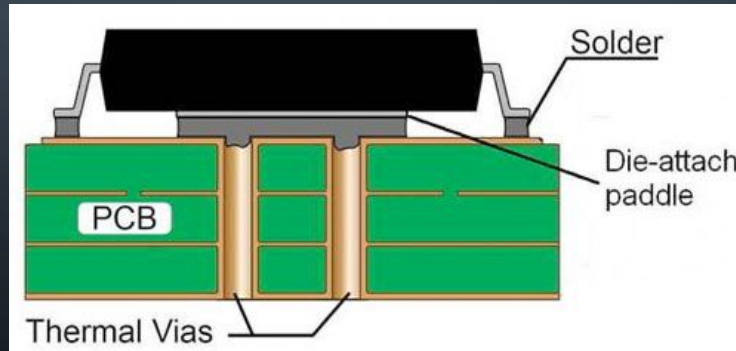
Both **power** and **ground** traces will have more current flowing through them, and if you don't make them wider than average, you'll get a ton of heat trying to flow through those skinny spaces, which can end up burning wires and ruining your board.



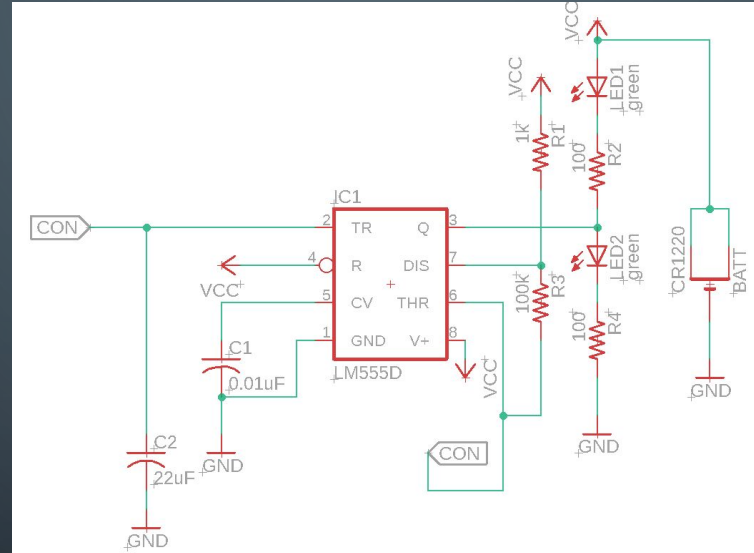
## 8) Use Vias

**Vias** are not only used for connections across layers, they are also used to dissipate heat from one layer to another. Adding **Via** moves the heat from one side of the board to another.

This comes in handy if you ever need to channel heat away from one of your super powered components, like an integrated circuit (IC). By placing multiple vias below the die of an IC, you'll be able to lower the operating temperature of a component.

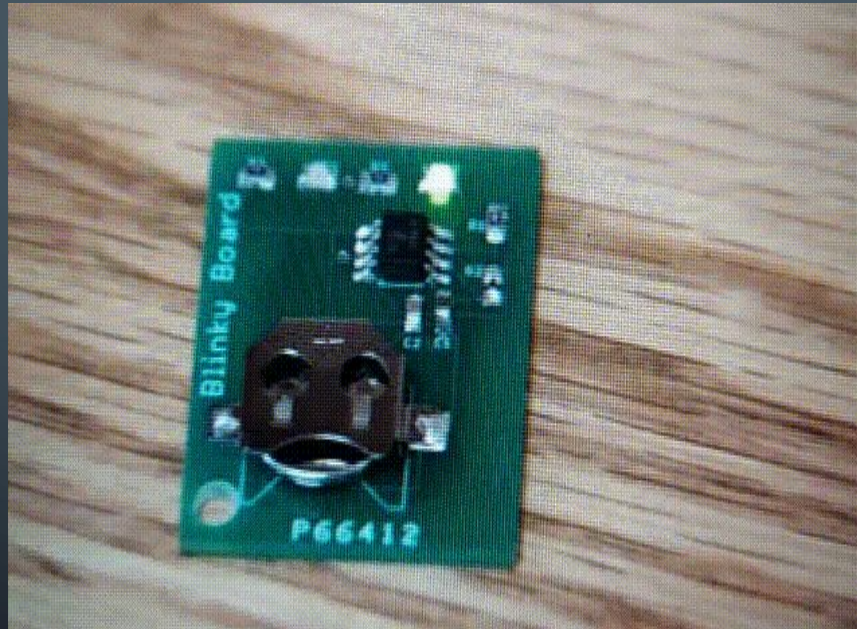


Now That you have made your **first schematic**, And you know the **Rules of the game**,  
Now lets see how to **manual route** it in a board layout.



Let's switch to **Eagle** now!

So when we get the **board manufactured**, This is how it looks like







# The Main Project of our session is the **POV-Fidget Spinner**



But wait,wait,wait.....It's Trivia Fun time now!


It is an **Learn-Along PCB Trivia**.

You not only get to revise all the concepts taught, but also get to learn some new and cool stuff.



We'll be conducting the quiz on **Kahoot** platform

- 1) Search for '**Kahoot**' on google/any search engine
- 2) Click this website

 <https://kahoot.it>

**Play Kahoot! - Enter game PIN here!**

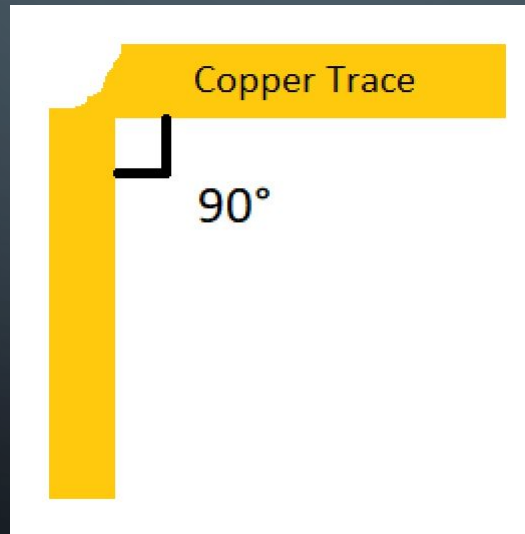
Join a game of kahoot here. Kahoot! is a free game-based learning platform that makes it fun to learn – any subject, in any language, on any device, for all ages!

- 3) Enter the **game pin** that we display on the screen now

# SOLUTIONS

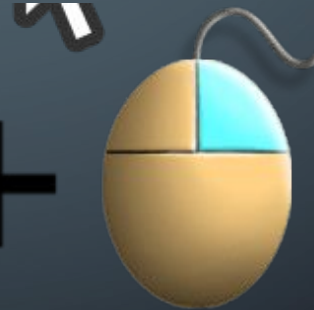
**Q1:** Why are  $90^\circ$  bends not preferred in PCB routing in low current circuits ?

**ANS:** It becomes tough to manufacture, the outside corner of that 90-degree angle has the likelihood of being etched narrower than your standard trace width



**Q2:** After grouping certain components and how do you move them using the mouse ?

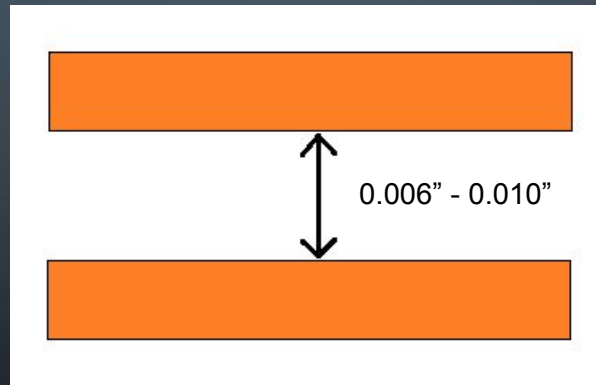
**ANS:** ctrl + right click drag





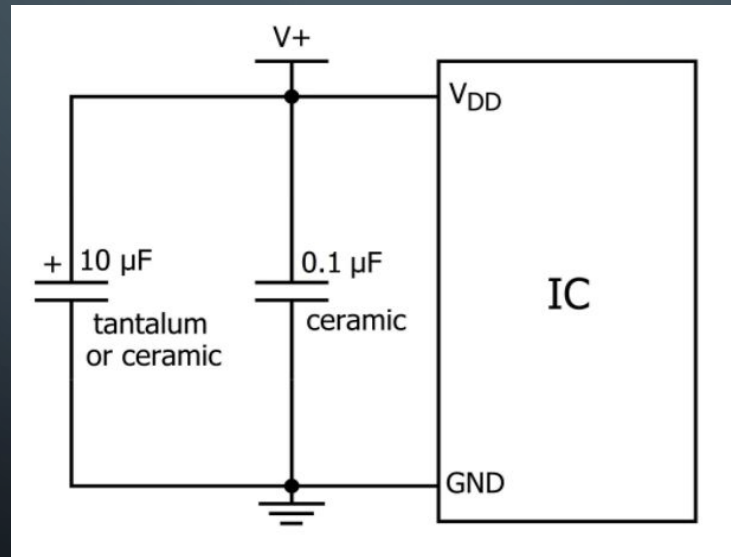
**Q3:** How much minimum space should generally be left between traces so that they come without manufacture defect ?

**ANS:** 0.006" - 0.010"



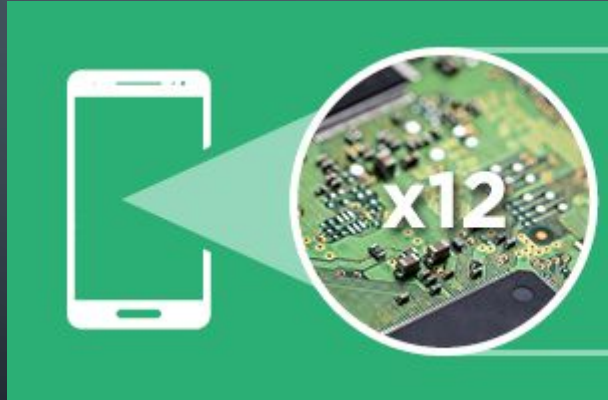
**Q4:** Decoupling capacitors must be as far as possible to the respective IC to which they are connected to, True or False ?

**ANS:** False, as the decoupling capacitor is used to filter out the noises coming from the IC, So the nearer it is, the better



**Q5:** Around how many layers do PCBs of mobile phones have ?

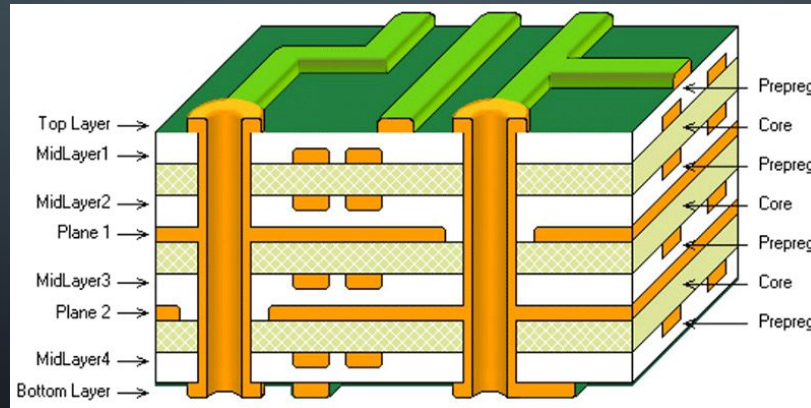
**ANS:** 12



**Q6:** Which of the following is NOT an advantage of having a multilayered PCB ?

**ANS:** Lower production time

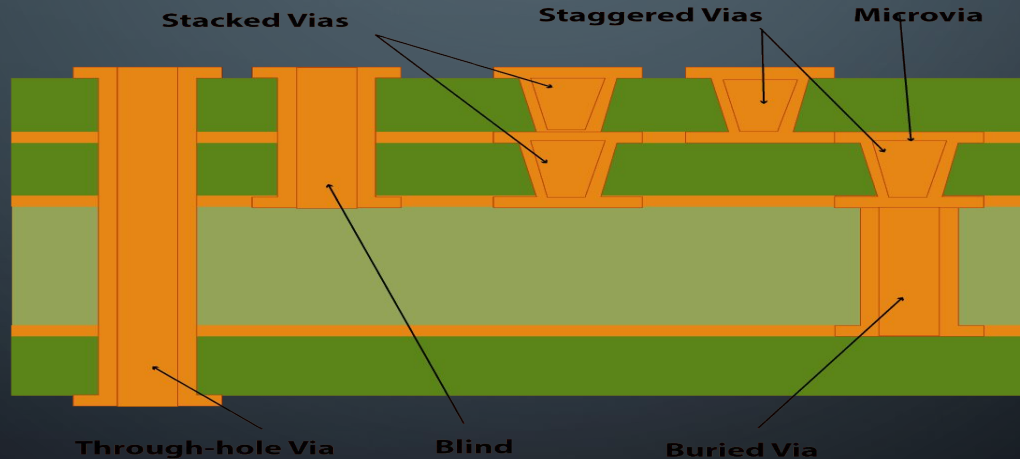
**Fact:** Making Multi-Layer PCB is a time-taking process





**Q7:**What is the role of Via in PCB layout designing?

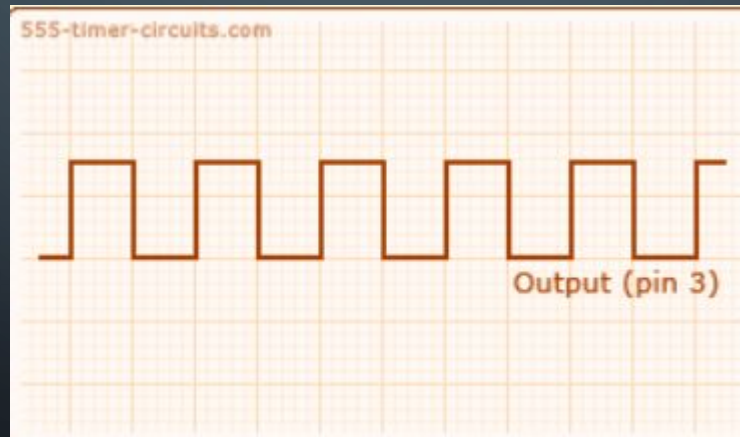
**ANS:** Via is used to connect the different layers in PCB routing





**Q8:** In which mode does the project of 555 Timer that we designed works?

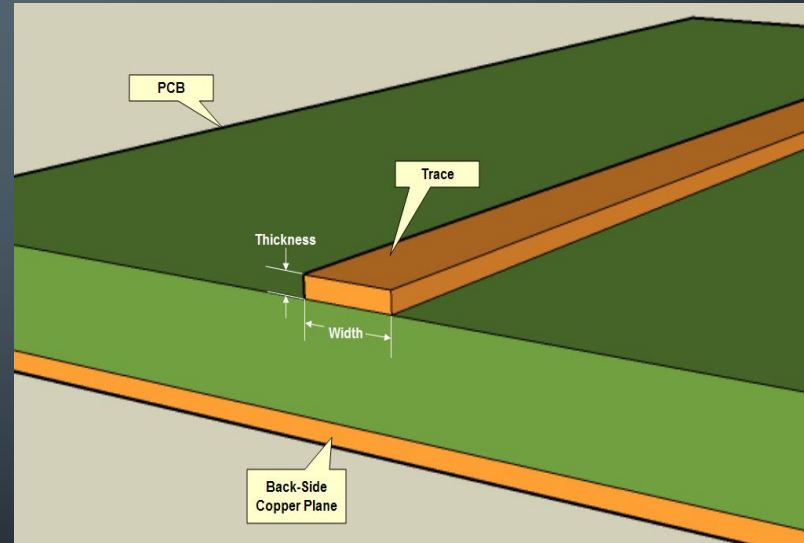
**AND:** Astable



**Q9.** The width of the traces and resistance faced by the electric current is directly proportional TRUE or FALSE?

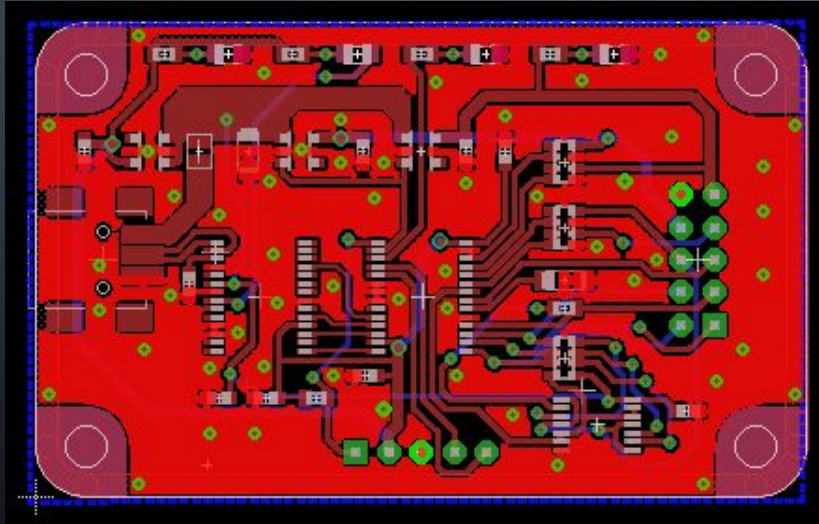
**ANS:** False

The wider your traces, the less resistance that electricity will encounter when flowing through your circuit



**Q10.** Which tool is used to create a Ground Plane in Layout Designing?

**ANS:** Polygon



GGs everyone!! Congratulations to the Podium finishers!

Now let's get back to our fidget spinner project!

First let's learn what is POV

# Persistence of Vision

It is the retention of a visual image for a short period of time after the removal of stimulus that produced it.

It is due to the persistence of vision that we continue to see the object in its position for about  $\frac{1}{16}$ th of a second after it is removed.

**Example:** When a burning stick of incense is moved fast in a circle, a circle of red light is seen.





# Some Examples of Persistence of Vision

The well-known **Flip-Book** uses the principle of persistence of vision



A series of drawings converted to an **Animation**

If the image on the left is **rotated** at a good enough speed, we see the animation on the right



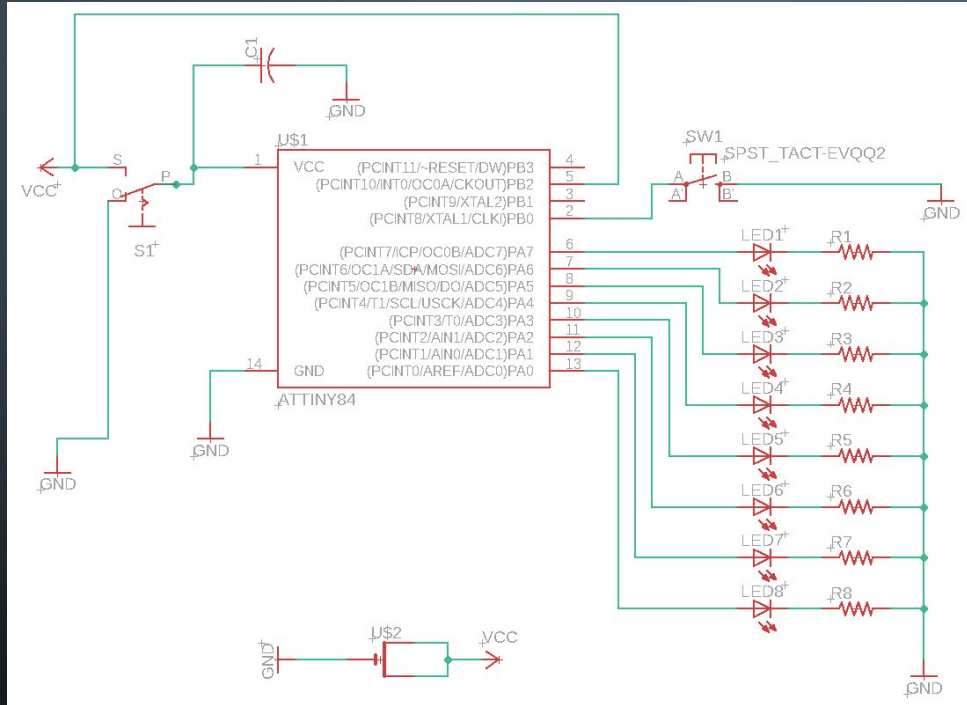
We hope you all have installed the library from the [link](#) we sent you yesterday through [Whatsapp](#)

Name of Library: **Fidget Spinner Example**

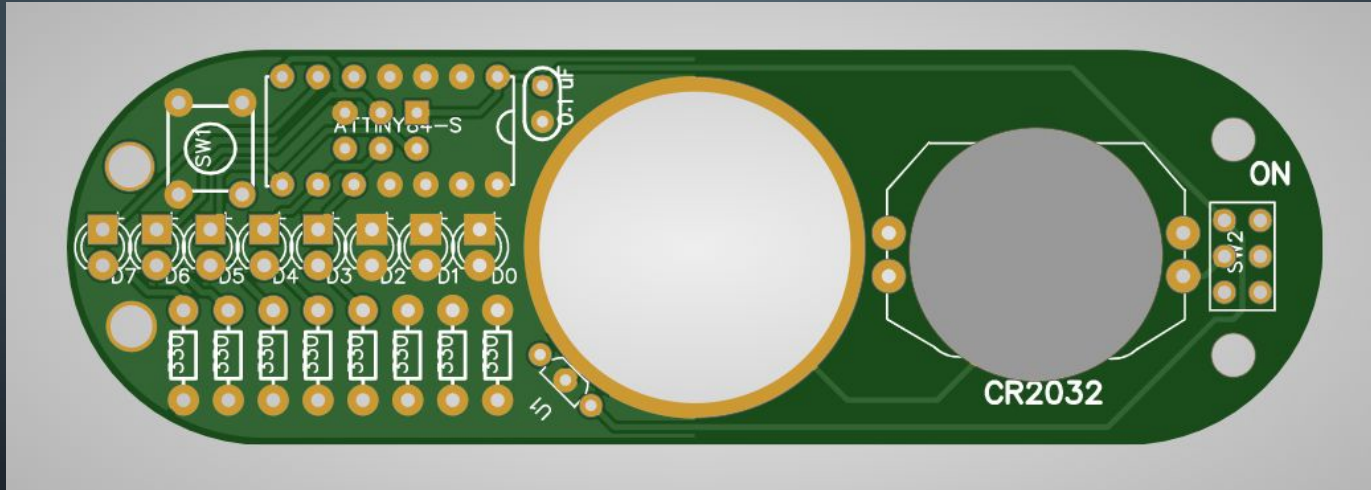
Link:

[https://drive.google.com/uc?export=download&id=1HSI1iC5QcFApjglntuQw8xZwrw-\\_1CwC](https://drive.google.com/uc?export=download&id=1HSI1iC5QcFApjglntuQw8xZwrw-_1CwC)

Now let's switch to **EAGLE** to make the schematic of the POV Fidget Spinner!



Now that we have finished the schematic of our **Fidget Spinner**, Let us show you how to make the board into **Fidget Spinner shape**.



**Thank you guys So much for  
attending the session**

**We'll meet again tomorrow on Day-3!!**

**Arun Palaniappan  
ME20B036**

**Mohammed Shamvil  
CH20B066**