

# Workshop 3 - PCB Design (Layout)

Friday 11th October

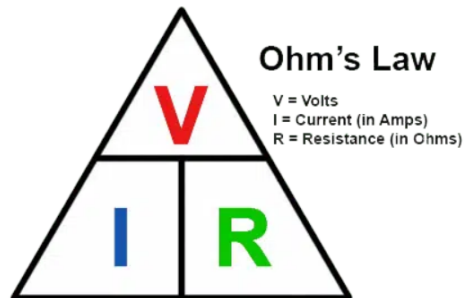
lecture

We started off by looking at Ohm's law and doing a few questions (below)

$I \times R = V$  (current multiplied by resistance = voltage)

$V \div R = I$  (voltage Divided by resistance = current)

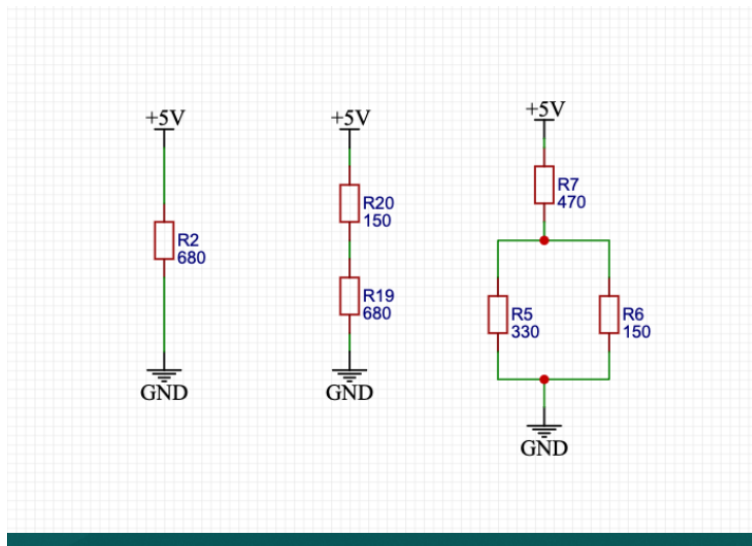
$V \div I = R$  (voltage divided by current = resistance)



$$V = I \cdot R \quad \text{(volts = amps times ohms)}$$

$$I = \frac{V}{R} \quad \text{(amps = volts divided by ohms)}$$

$$R = \frac{V}{I} \quad \text{(ohms = volts divided by amps)}$$



Series

$$R = R_1 + R_2$$

Parallel

$$R = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2}}$$

Ohm's Law

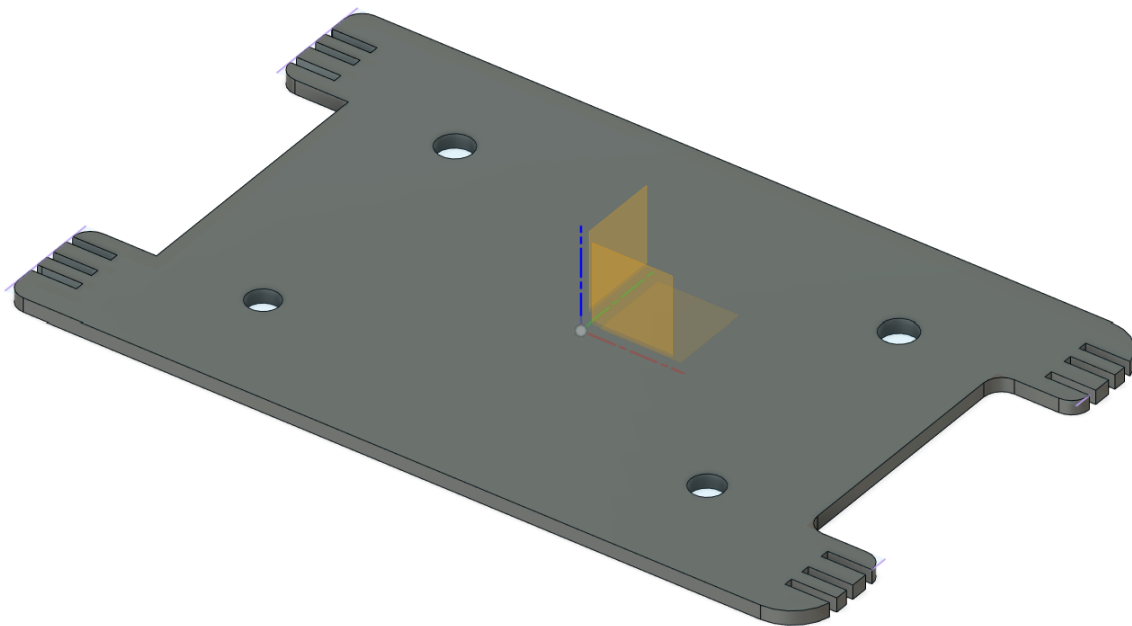
$$V = IR$$

2nd one is  $150 + 680 = 850$

$$5 / 830 = 0.006$$

voltage between the two resistors would be  $0.006 \times 680 = 4.09$

After this we went into EasyEda and was shown how to import a shape from Fusion 360. The process was simple enough, we build the basic design (see below) and then we create a new sketch on the surface of the design. Finish the sketch and then right click on sketch in the right hand drop down and save as a DXF file.

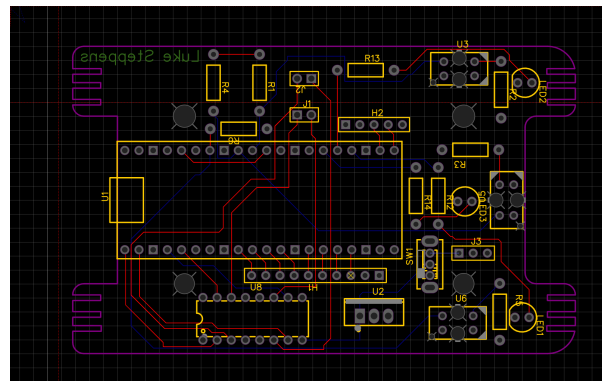
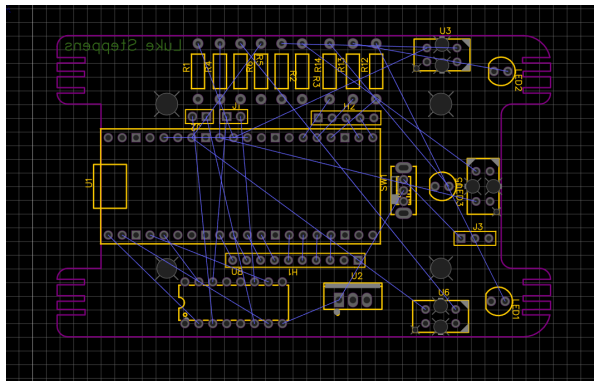


I gave Algernon front and back feet using the mirror function.

Within easyEDA we press the PCB button and we are given the below style schematic, then we import the DXF file and place all the components within it. We were advised to place components in places that make sense, such as the TCRT5000's needing to be near the edges as they will need to detect light, and items that have connecting tracks to be close to one another, although it is not always possible. This makes the puzzle-like element of connecting the tracks (known as routing) much less work.

Note: Shorter tracks = less resistance

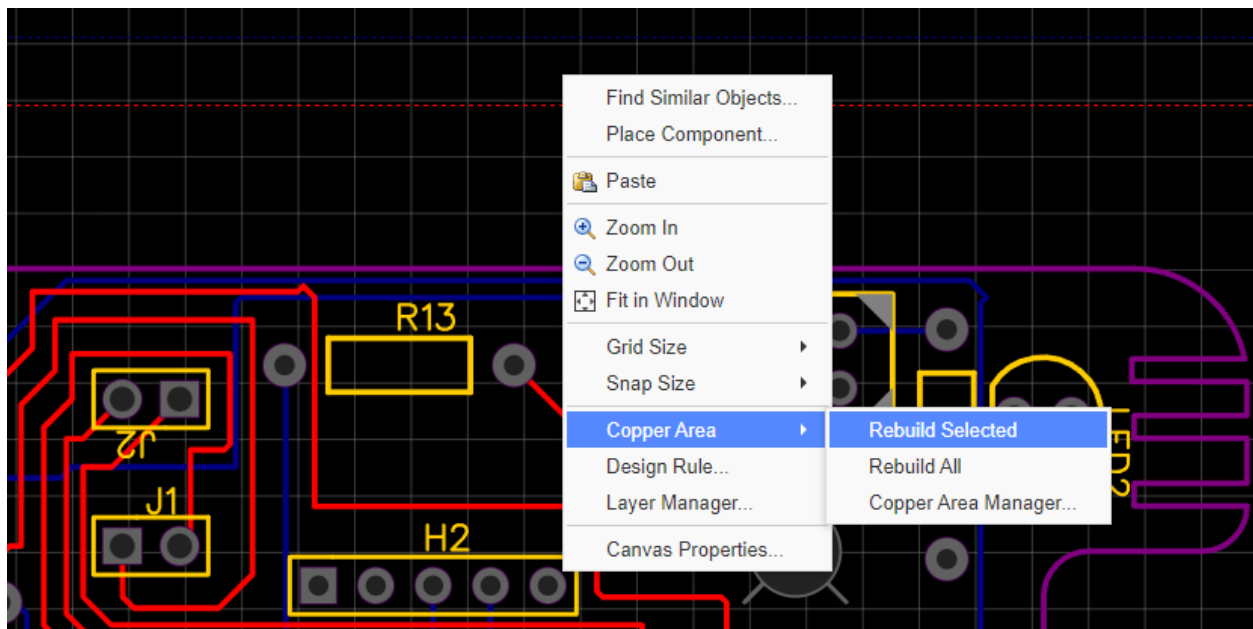
Another way to make the routing procedure simpler and more fun is to utilise more than one layer of the board (top and bottom in our cases) You can see red tracks on one side and blue on the other. You can always use the VIA option with creates a bridge over the same colour track buy going through to the other side of the board then back again. I havent utilised this as I wanted try and figure it all out as a test for myself.



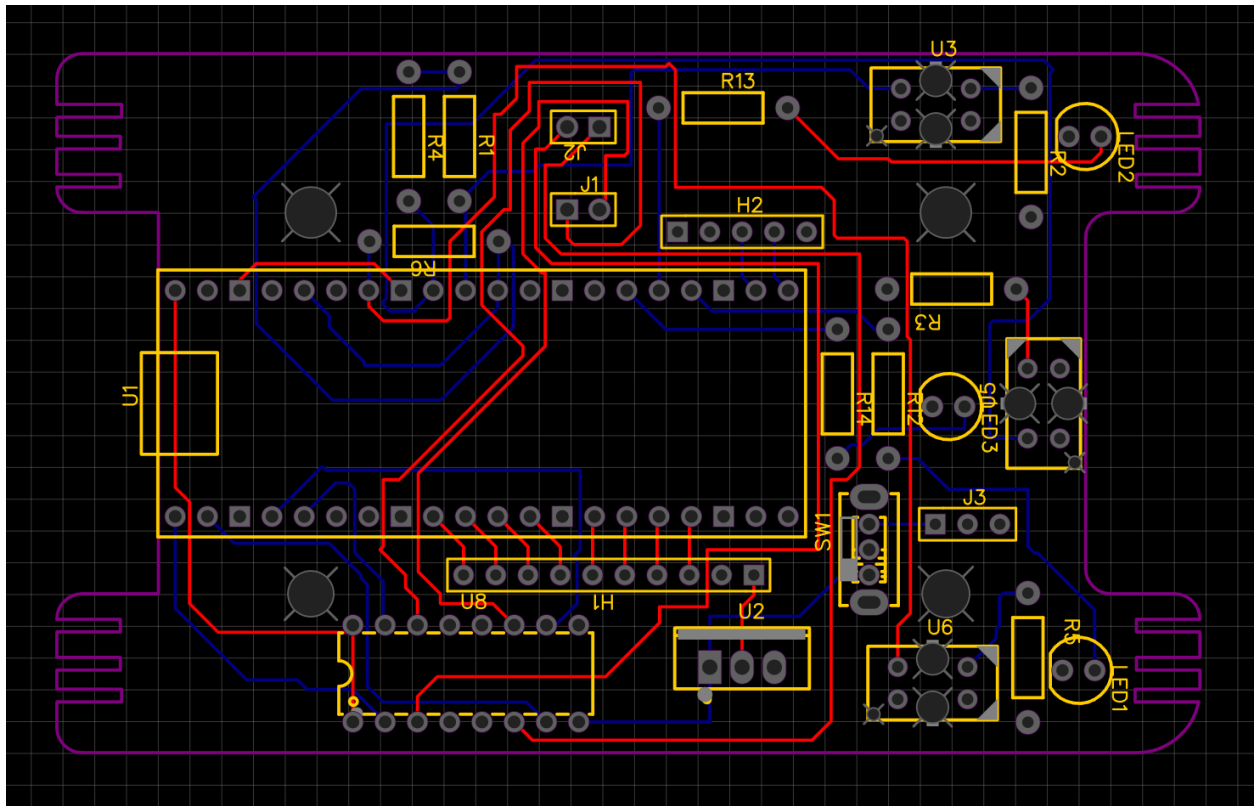
We were shown how to add a copper layer within the PCB that way there is no need to connect ground connections, of which there are many. Alongside this we did the same with connecting all +5V removing more manual connections and saving time.

In order to make it easier for myself I decided to make the track thinner so they could navigate through the components easier. This was a mistake and when finalising the schematic and pressing the DRC button (Design Rule Check) I was greeted with over 40 errors, I attempted to fix them all one by one but was unable to claim victory and deleted them all and started again.

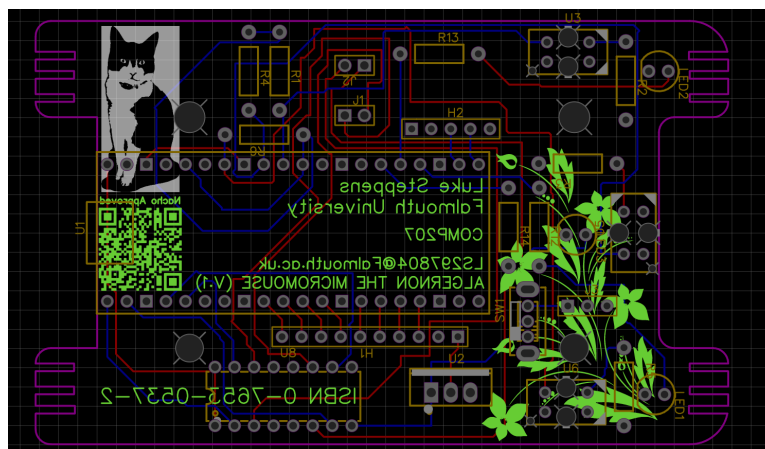
When editing you have to remake the copper layer mentioned above for it to fully update to any changes you have made with the tracks.



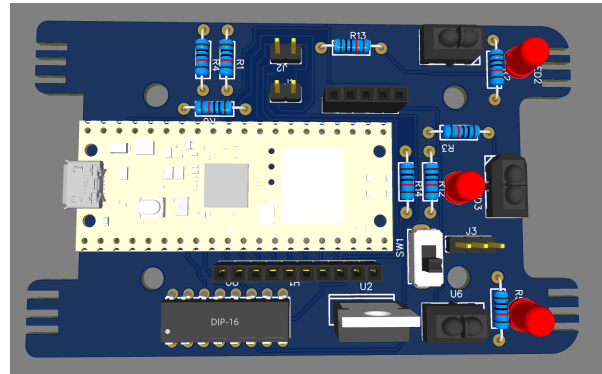
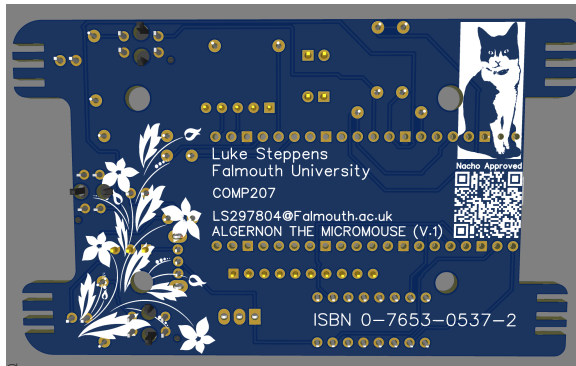
When re routing the whole board I found my style aligning with the advice given, which was to not make arbitrary lines and try to have them run alongside one another.



And Finally we were shown how to add pictures and writing to our PCB to personalise them



This is the final product



COMP207-W3-WKSP-PCB\_DESIGN\_LAYOUT-2.pdf