Front Panel Instructions

[Link to YouTube Tutorial]

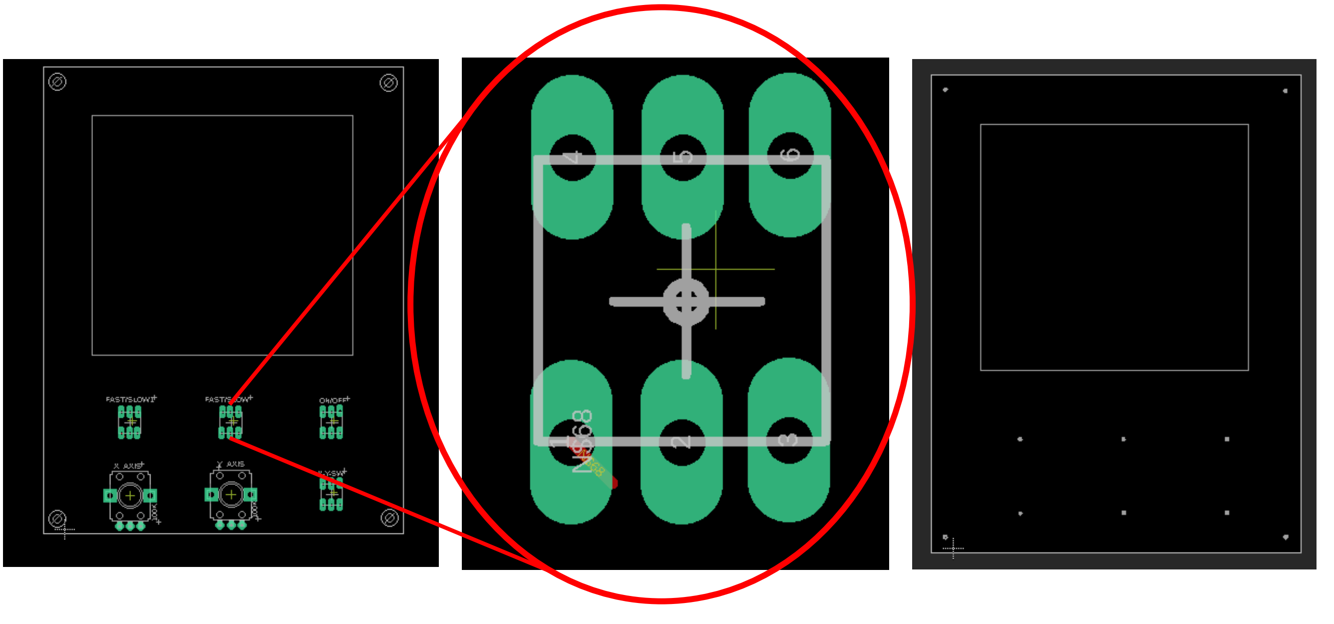
Eagle

Designing the PCB

- Design the PCB in Eagle
- Design the initial panel in Inkscape (just so you can ensure PCB design is good)

Creating a Template From the PCB

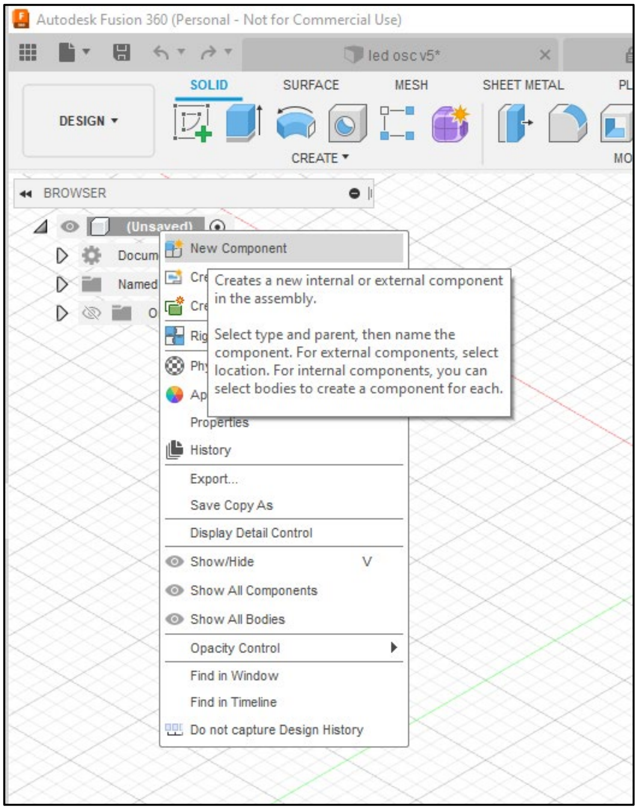
- Remove all of the parts from the schematic except for the main components like potentiometers, switches etc.
- Add small circles at the centre points of each component the board.
    - Draw/circle
    - Width - 8
    - Change grid to '1'
    - Layer – 21 place
- Delete all of the components from the schematic so you are left with just the circles created on the board
    - Delete mounting holes on the board
- Export as DXF file
    - File/export/DXF
    - Save in a folder and call it the name of the project

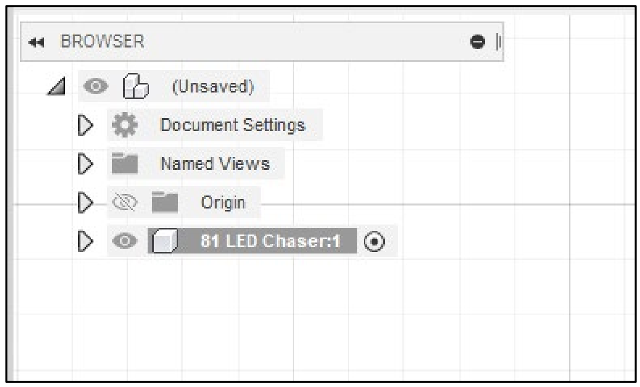


Autodesk Fusion 360

Creating a new sketch

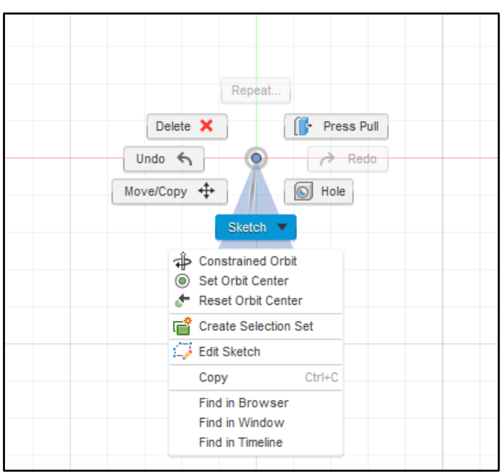
- Place curser over 'unsaved' and right click
- New Component
- Name new component
- Hit OK
- You only need to see the top plain for the sketch so hit 'top' in the box in top RHS
- Make sure that you are in the 'new component' section by clicking the small circle on the right

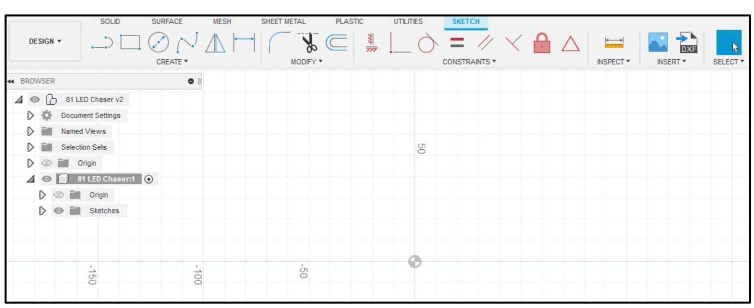




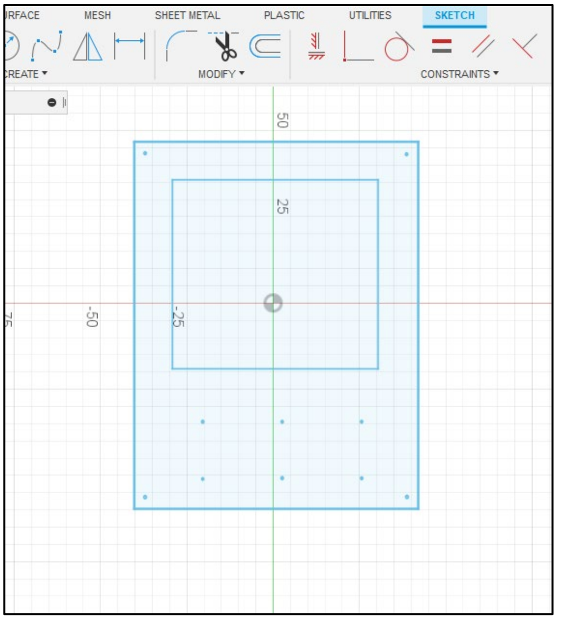
Finding the sketch!

- If the 'sketch' screen isn't showing then do the following:
    - Right click centre circle
    - Click 'sketch'
    - Click on any of the shape suggestions





Autodesk Fusion 360

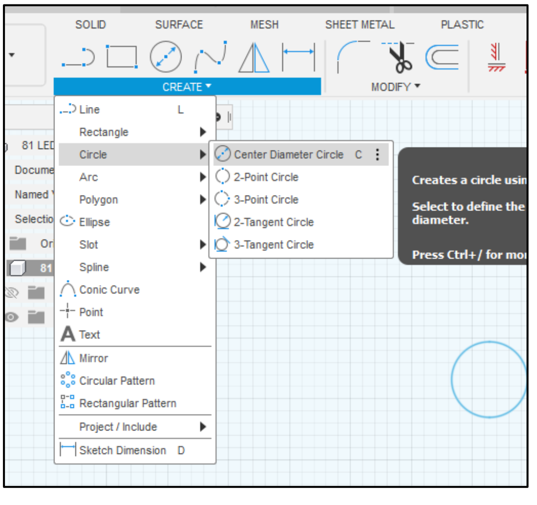


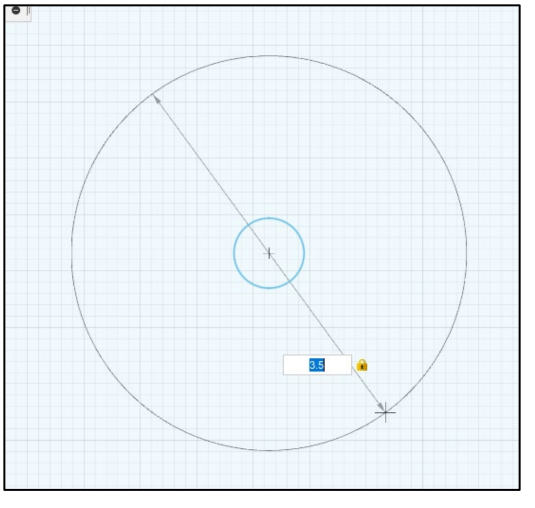
Importing the Eagle DFX file to Fusion 360

- Go to Insert/DXF/select DXF file
- File should load to screen
- Hit 'ok'

Creating larger circles for the components

- Next you need to enlarge the circles
  - Go to one of the circles on the DFX file
  - Go to create/circle/centre circle diameter
    - Note (you can also hit 'C' on the keyboard as a short cut
  - Place cursor in the centre of the circle and type in the size you want it to be
  - Hit – enter



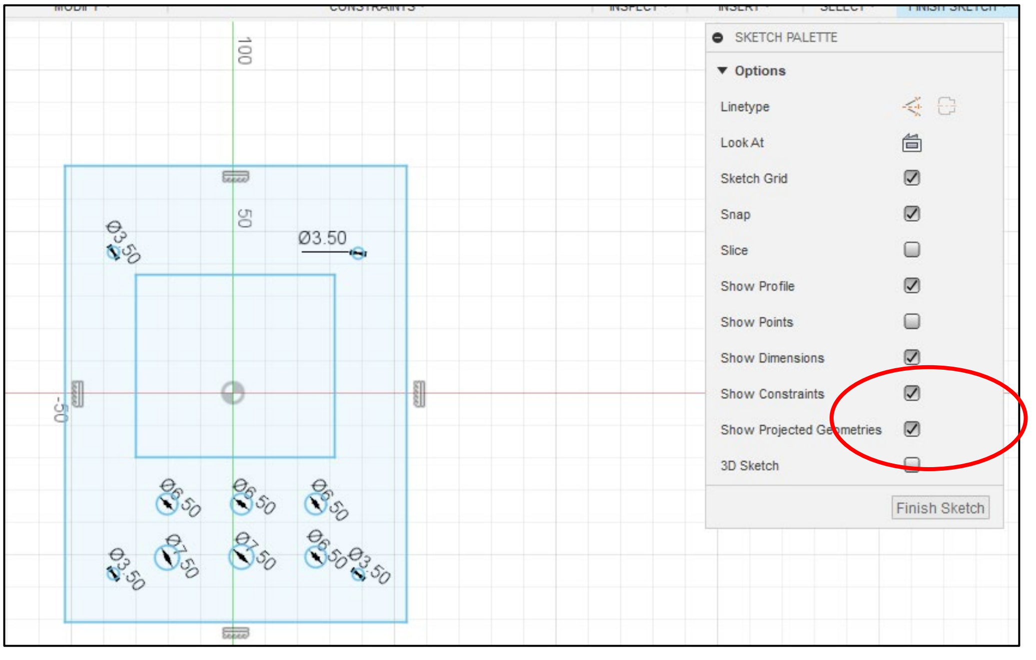


- Highlight the smaller circle (the one that was made in Eagle)
- Hit - delete and remove the circle that was added to the board

Autodesk Fusion 360

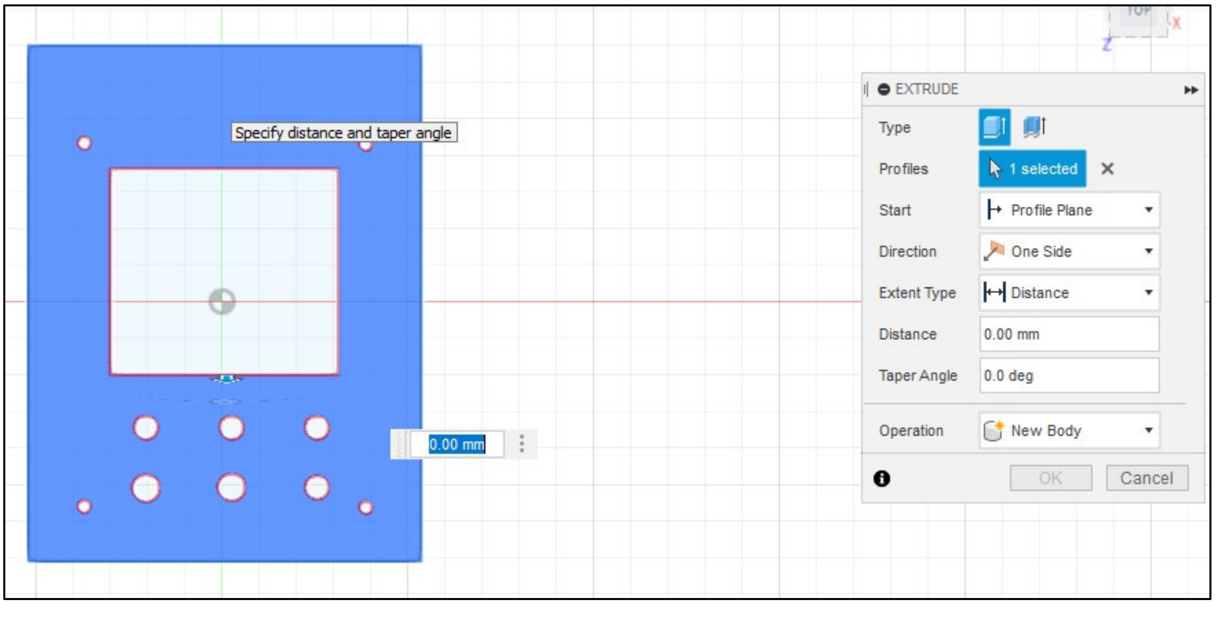
Saving and Extruding the Finished Sketch

- Once you have completed the sketch you need to do the following
- Go to 'finish sketch'



Saving and Extruding the Finished Sketch

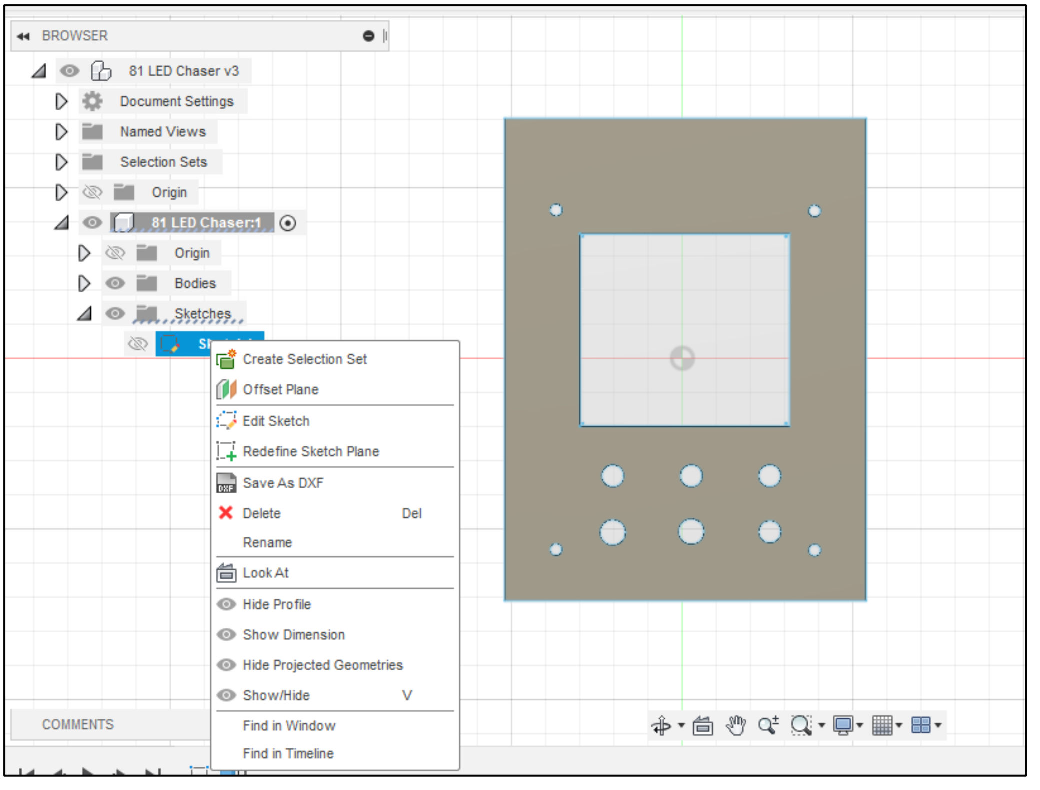
- Hit – extrude
- Put cursor onto the sketch and click the mouse
    - The sketch will turn blue
- Add the distance you want the sketch to extrude (1.6mm is good for PCB's)
- Note you don't have to do this but it looks good!
- Hit - ok

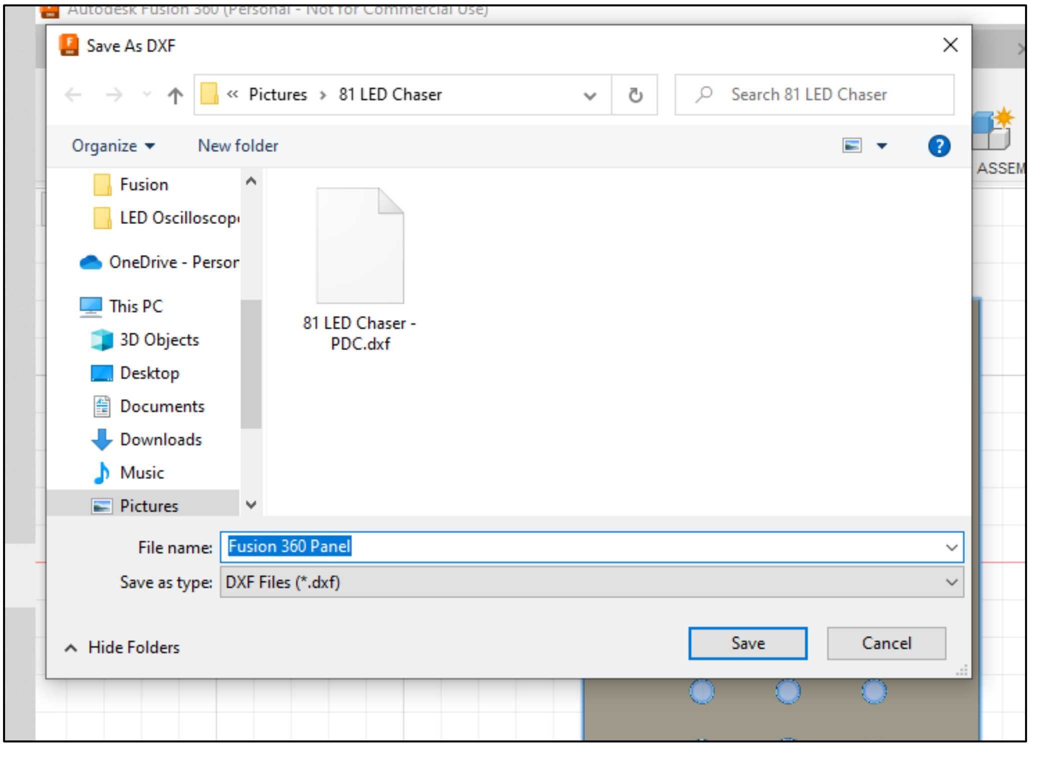


Autodesk Fusion 360

Exporting the sketch as a DFX File

- Right click on sketch dropdown
- Go to – save as DFX
- Name the sketch and save
- Note – you can go back into the sketch by double clicking the sketch dropdown

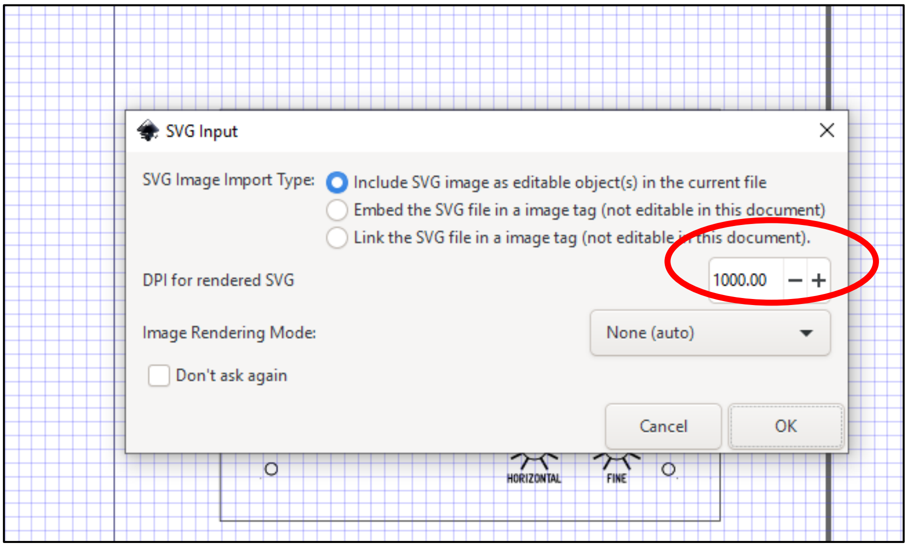




INKSCAPE

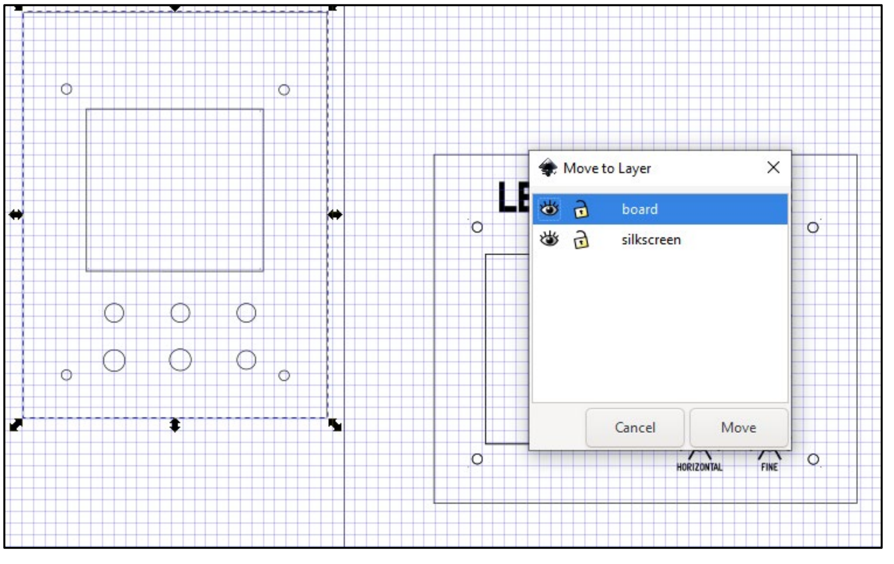
Adding the DFX file to Inkscape

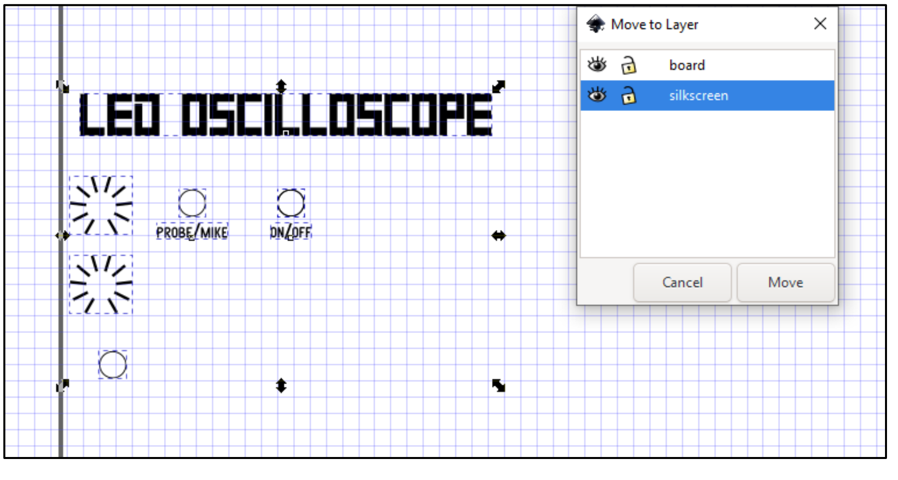
- Import the Fusion 360 DXF file to Inkscape
- Change DPI to 1000



Layers

- Save the DXF file to one layer(call it board)
- Save any graphics to another layer (call this silkscreen)

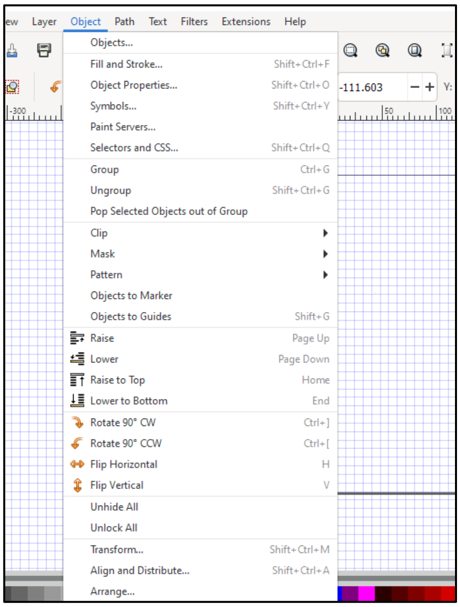


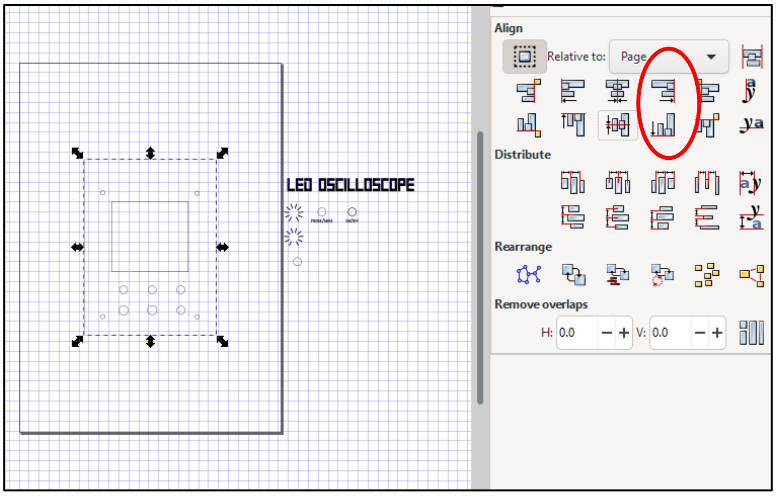


INKSCAPE

Aligning Board to Centre of Page

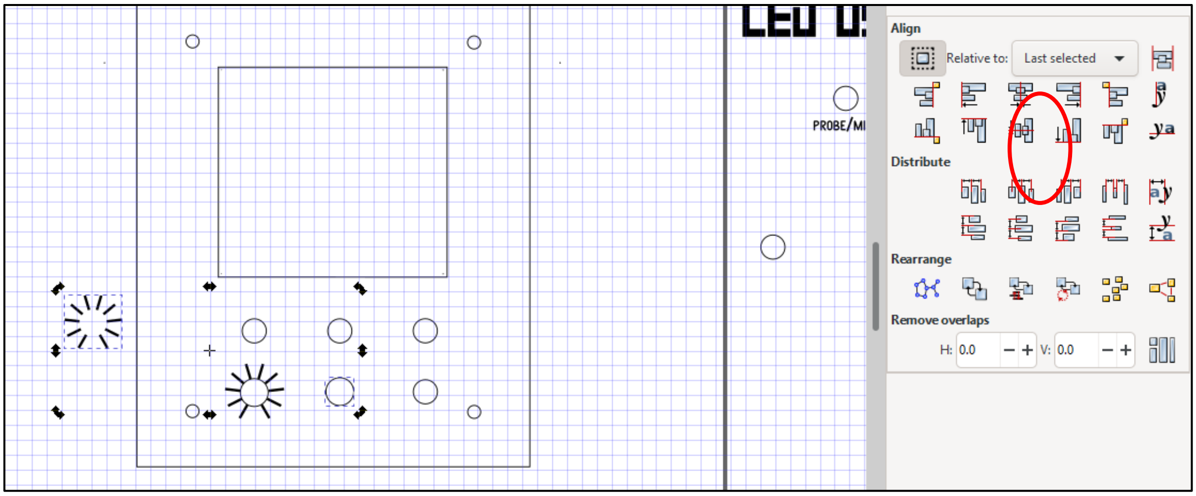
- Go to – Object/Align & Distribute
- Click on board
- Align relative to page
- Hit the centre on vertical and centre on horizontal





Aligning the Pot Dials to the Holes

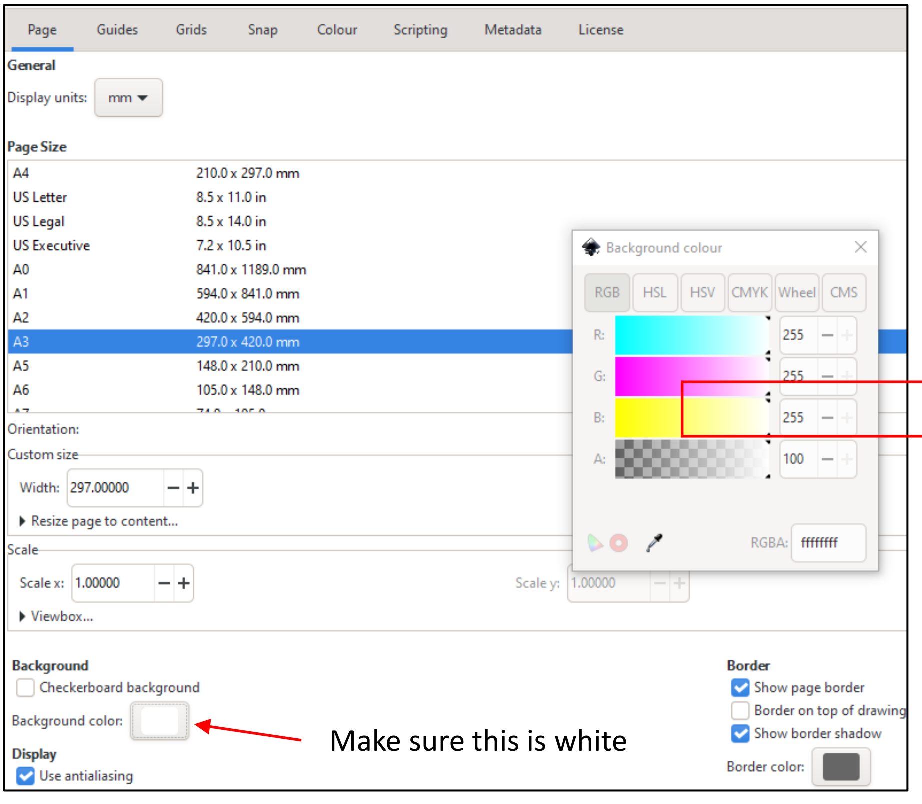
- Go to – Object/Align & Distribute
- Click onto the dial ad then the hole you want to dial to go around
    - You might have to 'un-group' the panel to be able to highlight sections
- Hit the centre on vertical and centre on horizontal
- You may have to adjust slightly to ensure the dials are aligned with the holes



INKSCAPE

Making the Background White

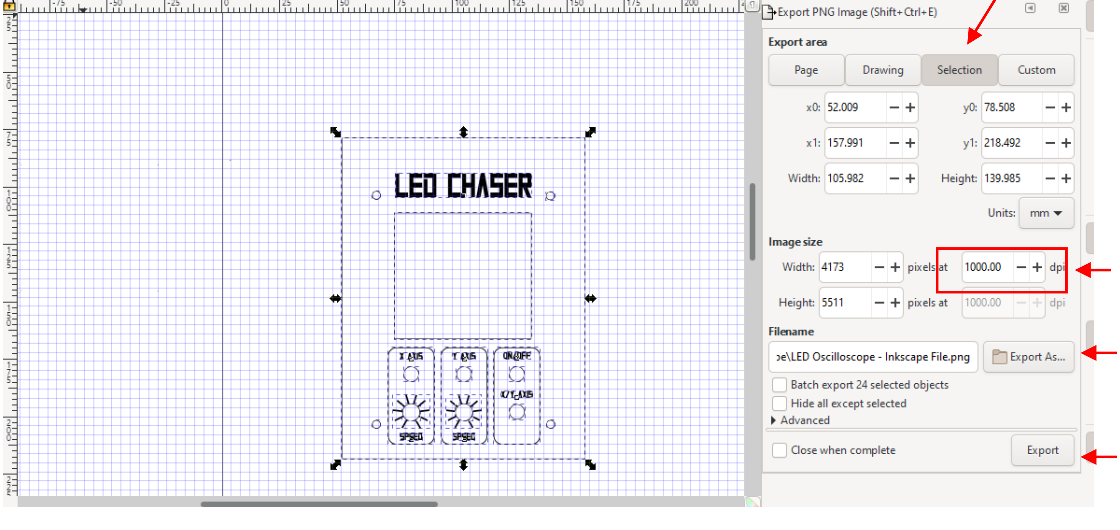
- File/Document Properties
- Then go to background colour
- Make sure you are on RGB
- Change 'A;' to from transparent to white by moving the toggle to the right



Saving & Exporting a PNG Image

- File/Export PNG Image
- Make sure image size (dpi) is set to 1000
- Highlight the full image
- Make sure 'export area' is on selection
- Hit 'Export As' and save into folder
- Then hit 'Export'

Selection should be on



Change to 1000

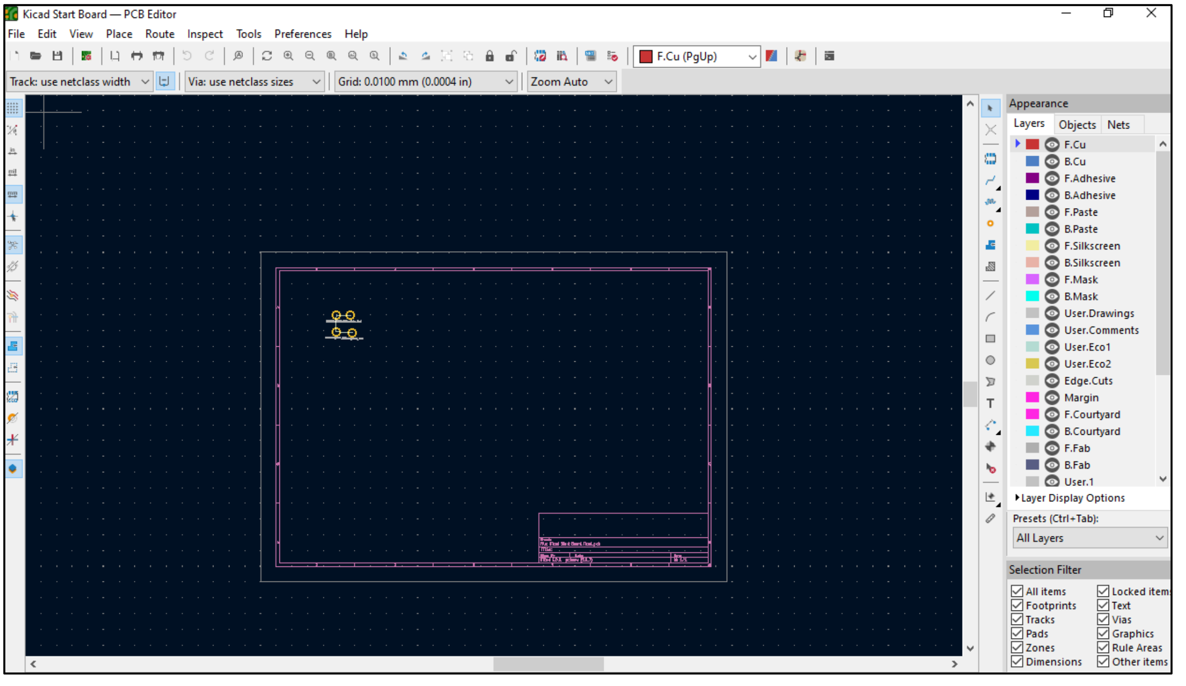
Hit this key

Then this key

KICAD

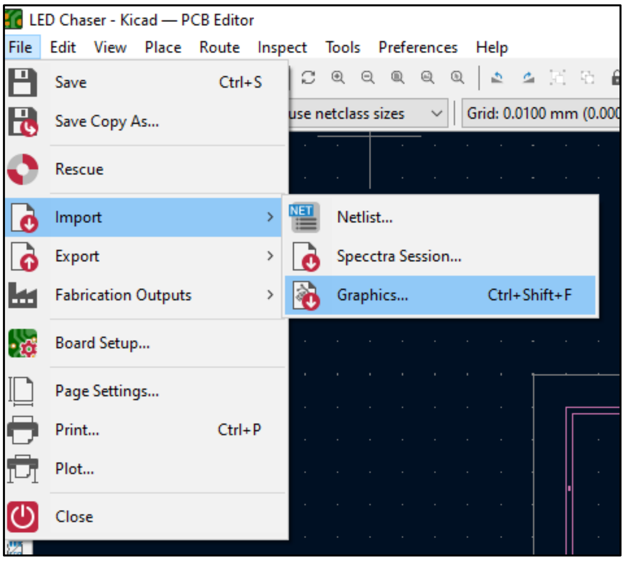
Opening the Start Board

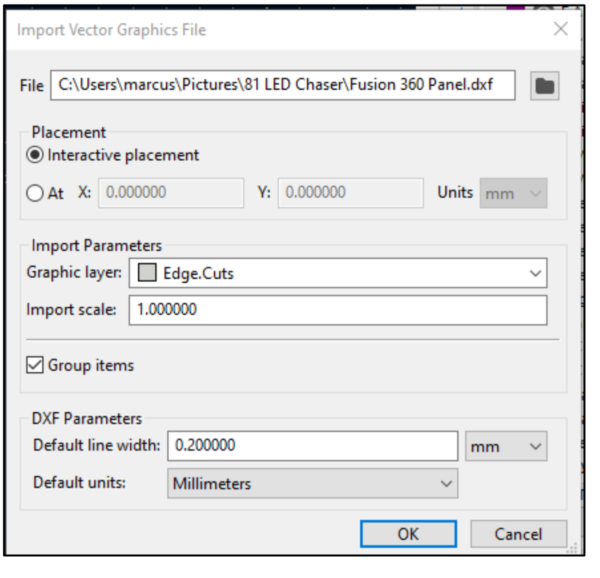
- Open Kicad software
- File/Open project/pictures/kicad/kicad start board
- Save as Pictures /<folder name>/Kicad



Importing the Fusion 360 File to Kicad

- File/Import/Graphics
- Make sure that the graphic layer is on 'edge cuts'
- Select the Fusion 360 file and hit 'ok'

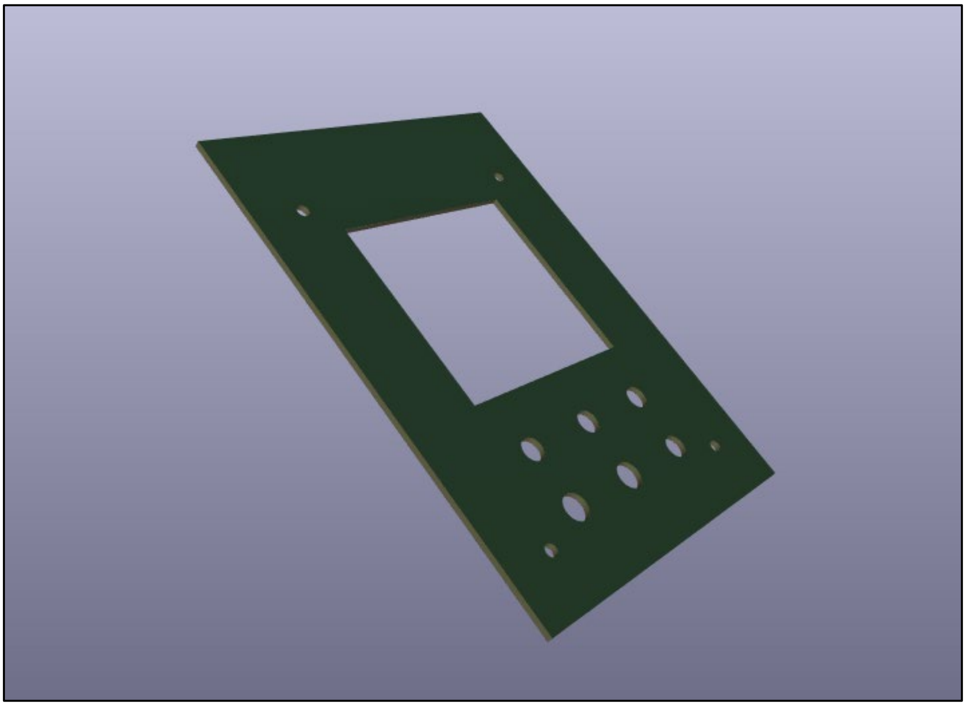




KICAD

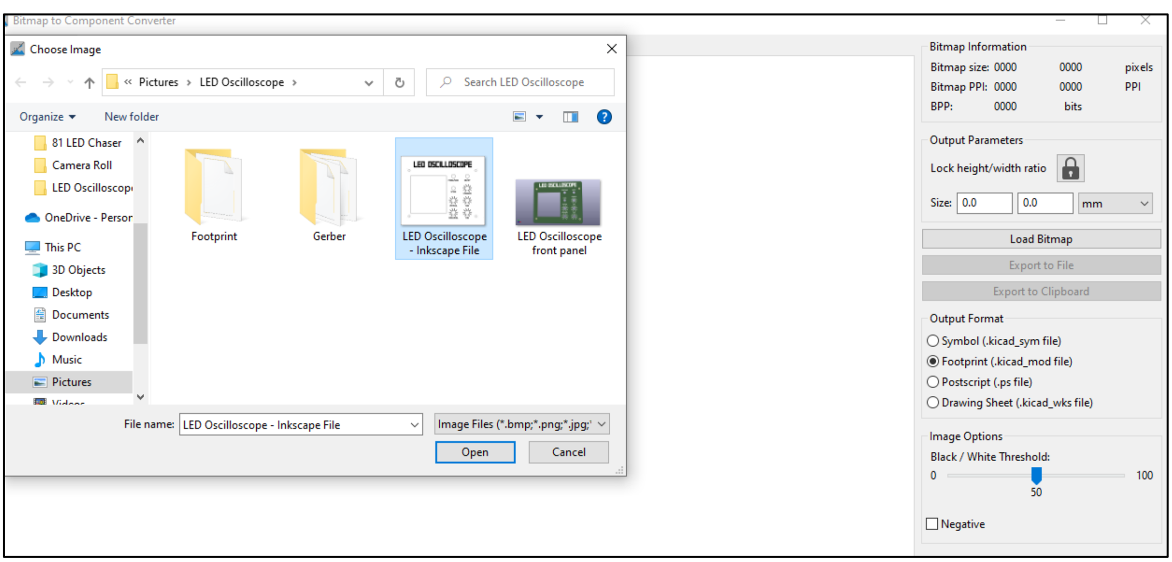
Checking the board

- Check file by going to view/3D viewer
- If the holes are not showing up in the 3D view then run DRC to identify any issues
- You may have to go back into Fusion 360 and clean up any issues
- Save revised file and Run 3D viewer again once issues have been fixed



Adding Silkscreen Graphics onto the Board

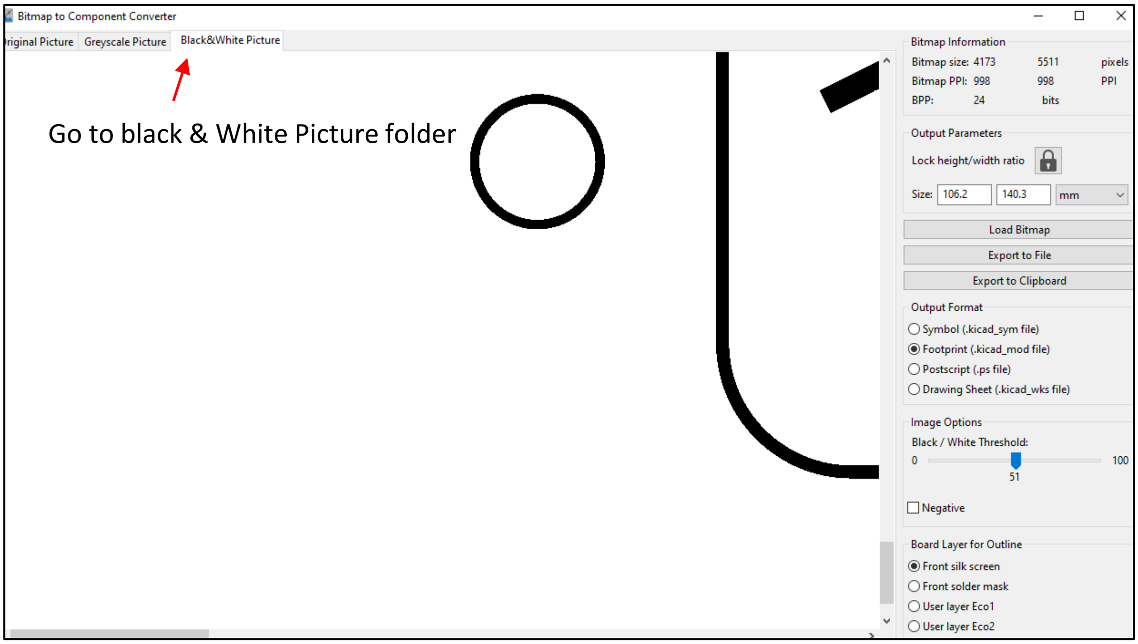
- First go to image converter in the front page of Kicad
- You now need to import the Inkscape front panel image
- Go to 'Load Bitmap'



KICAD

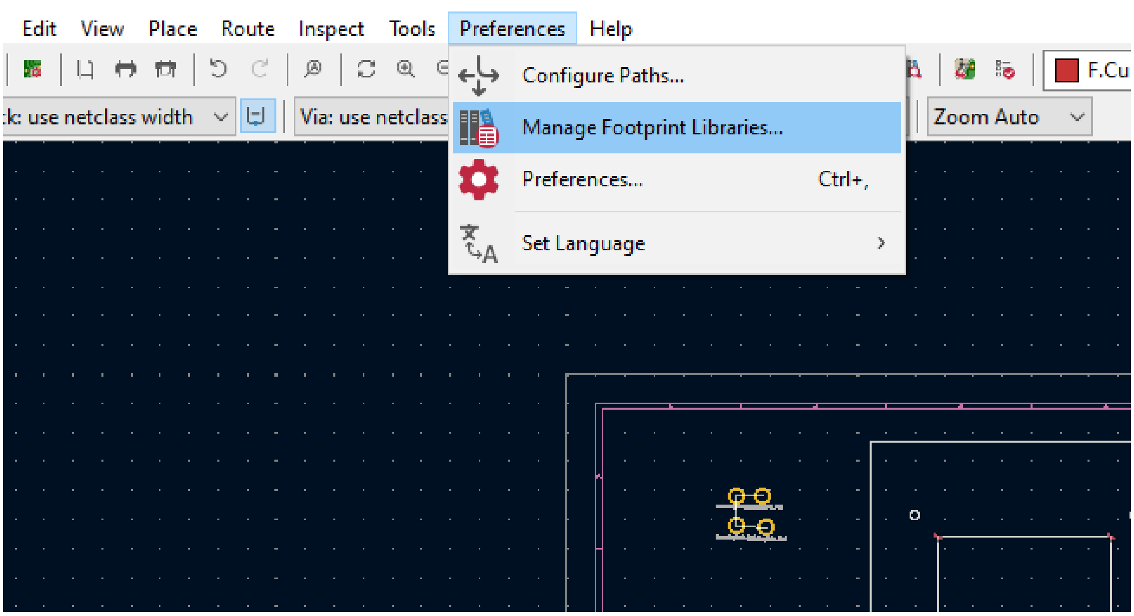
Adding Silkscreen Graphics onto the Board - Continued

- Go to ‘black & white picture’
- Export to file
- Create a new folder and call it - <name of project> Silkscreen Footprint
- Save file in folder



Adding silkscreen as a new footprint

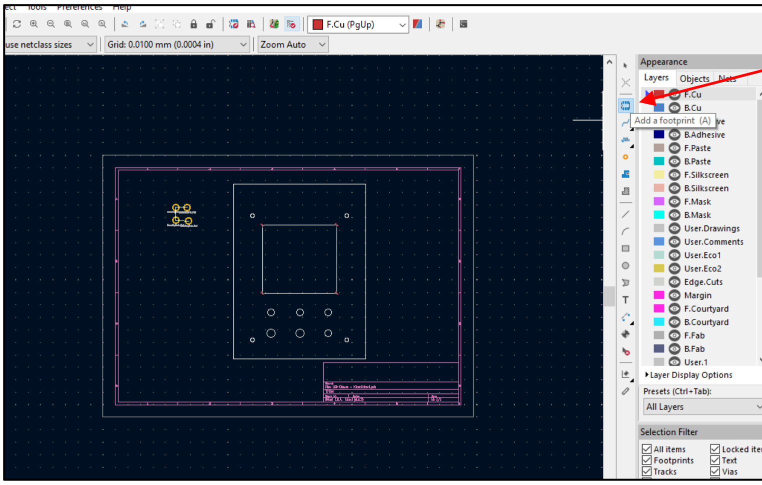
- Go to preferences/manage footprint library
- Next go to project specific library
- Hit the folder button
- Load <name of project> Silkscreen Footprint
    - Just select the folder – don’t load what is inside of it
- Hit - ok



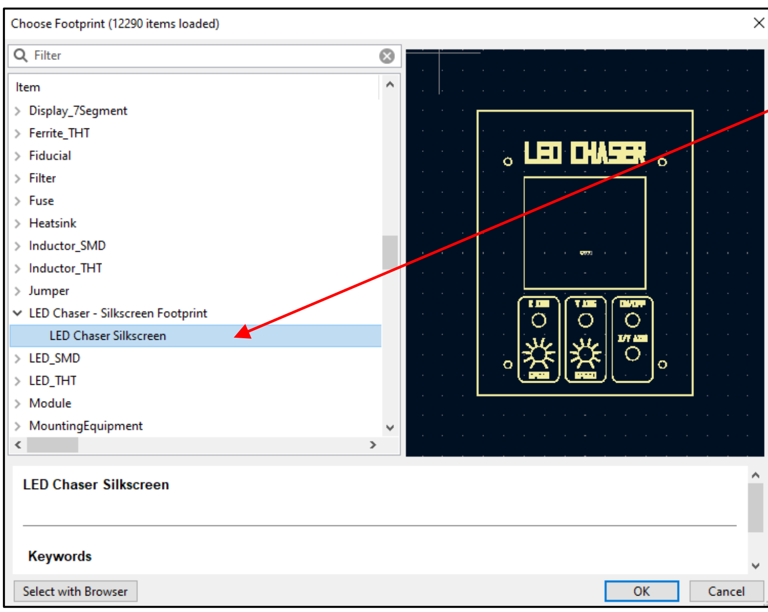
KICAD

Adding silkscreen as a new footprint - continued

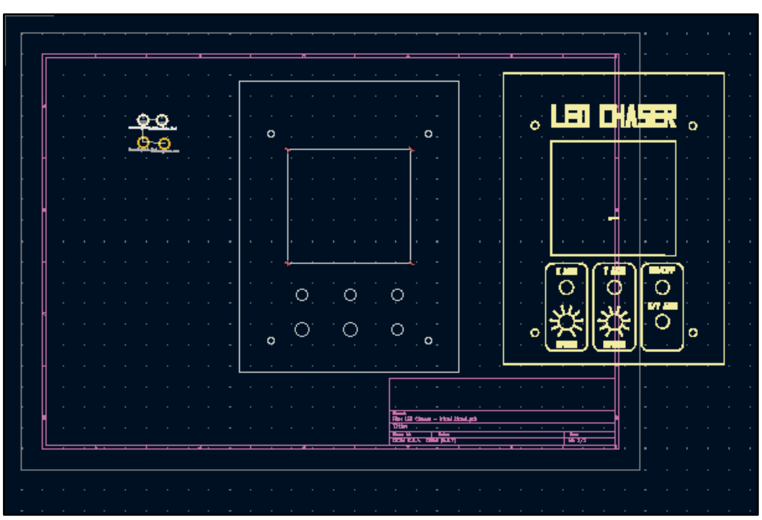
- Go to – add a footprint which is on the right hand side of Kicad
- Find the folder that the footprint was saved in
- Open folder and select footprint



Add a footprint button



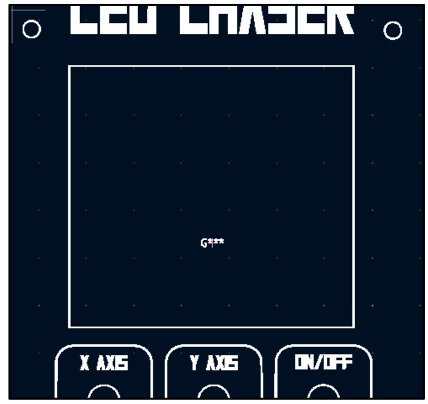
Find footprint saved and hit ok

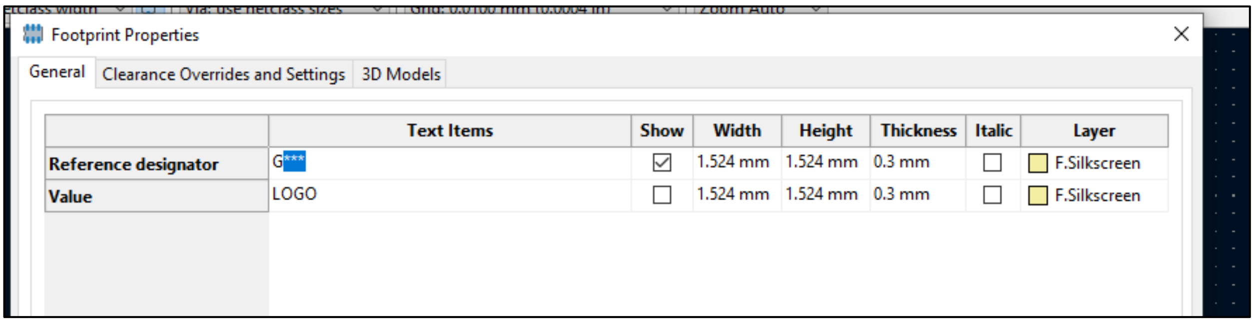


KICAD

Removing G*** From the Silkscreen

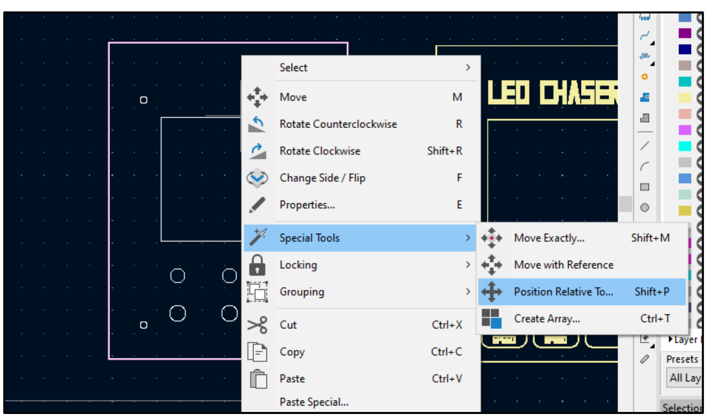
- Double click into the silkscreen
- Un-tick the box next to the G***
- Hit – Update footprint from library
- Hit – Update

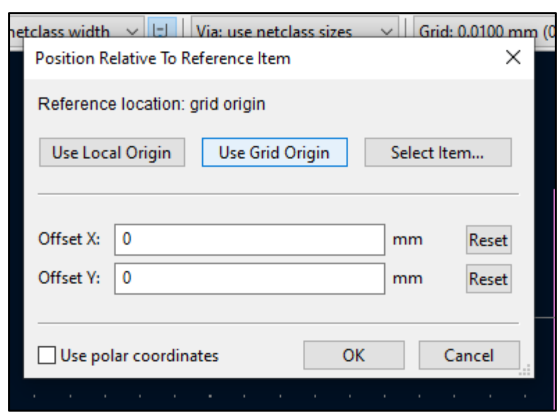


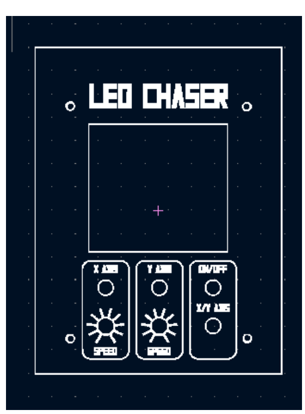


Aligning the Board and Silkscreen up Using Position Relative to…

- Click on the board and right click
- Go to – Special Tools/Position Relative to…
- Hit – User Grid Origin
- Hit - ok
- Do the same thing for the silkscreen
- The silkscreen should be directly on top of the board



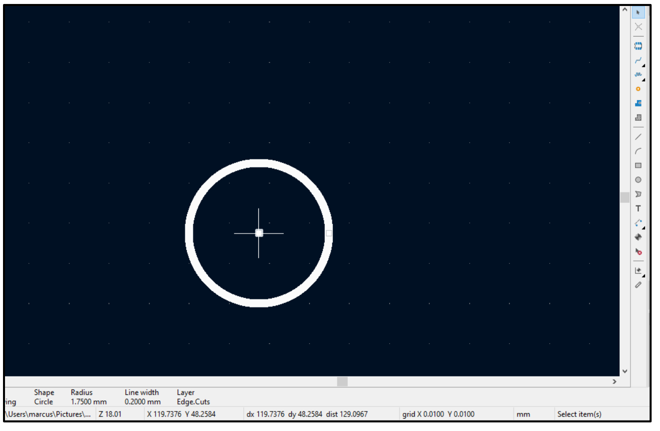


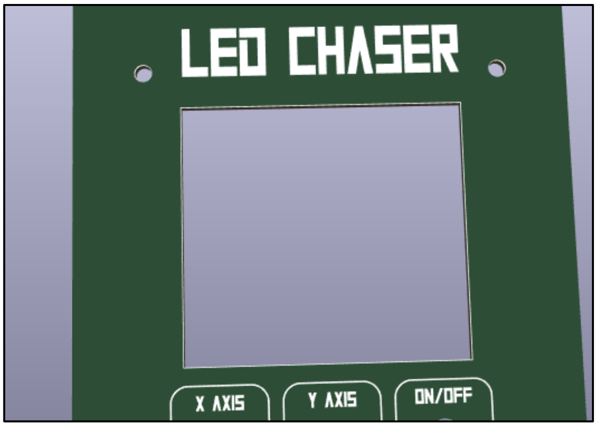


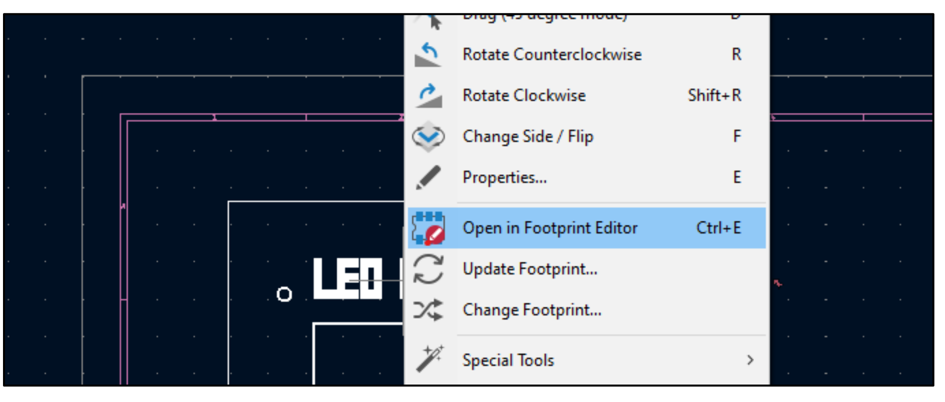
KICAD

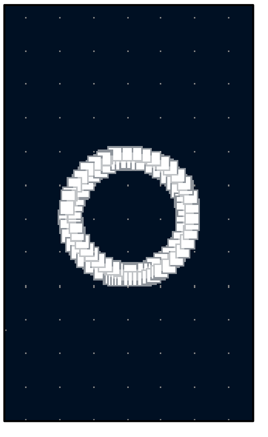
Editing the Footprint to Remove Borders, Holes etc

- Click onto the silkscreen and right click
- Hit Open in Footprint Editor
- Click onto the object that you want to remove (i.e. – white circle around the mounting holes)
- Right click and cut object (or hit delete)
- Go to File/Save as/Save/Overwrite



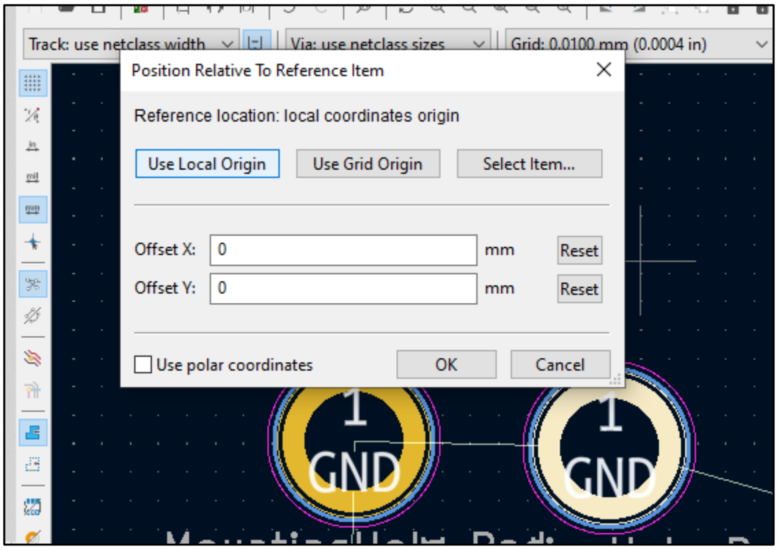


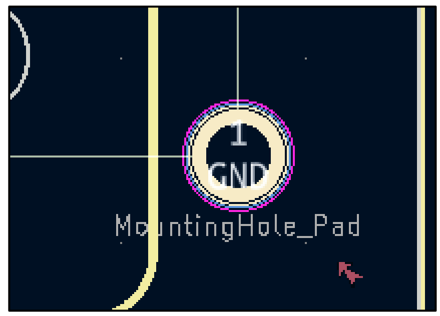




Adding the Copper Mounting Holes to the Board

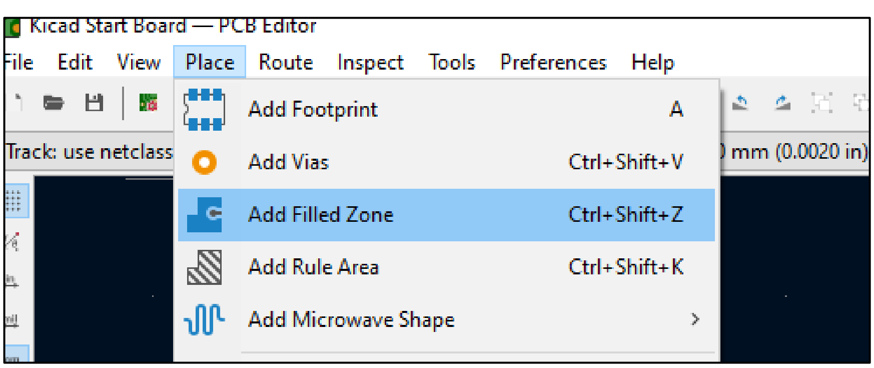
- Create a local origin grid by clicking onto the side of the circle
- Place the cursor in the centre square
- Hit the space bar
- Click onto a copper ring and right click
- Go to – Special Tools/Position Relative to…
- Hit – Use Local Origin
- Hit - ok



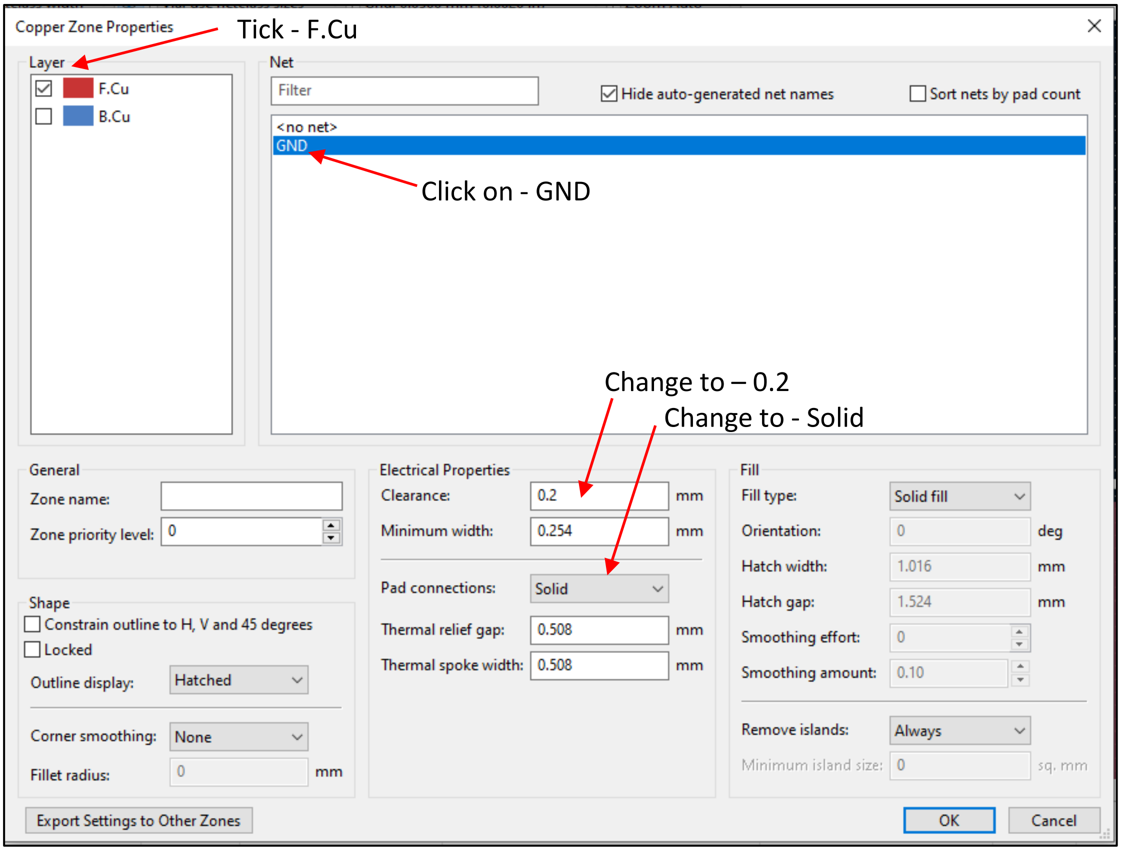


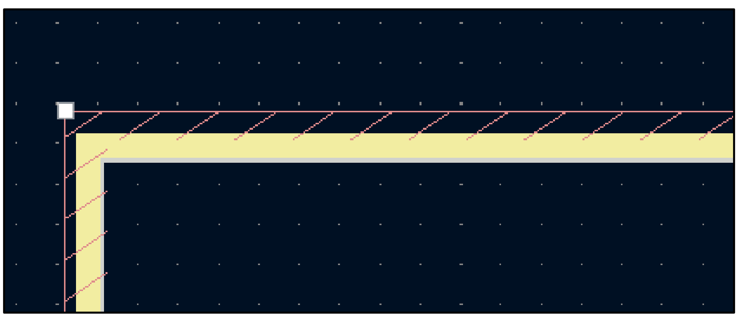
Adding Copper to Top & Bottom Layer

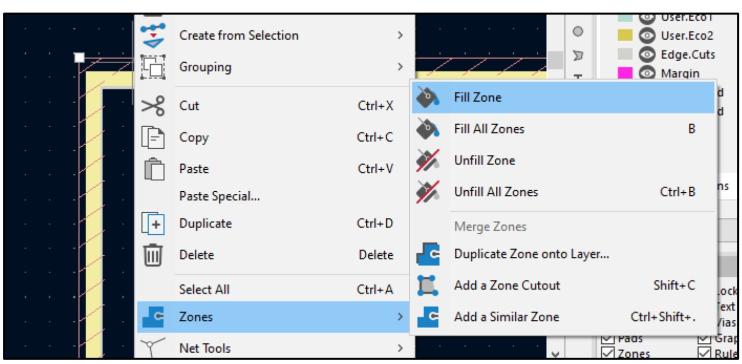
- Go to place/Add Filled Zone
- Click near the corner of the panel – properties pop up will appear
    - Tick F.cu
    - Hit – GND
    - Change Clearance to 0.2
    - Change Pad Connections to Solid
    - Hit ok
- Draw a square around the panel
- Right click the border and go to Zones/Fill Zones
- To add a copper layer to the bottom do the following:
- Right click the surround and go to Zones/Duplicate Zones onto Layer
- Add a tick to B.cu
- Hit Ok
- Go to fill all zones and hit ok
- To check if it worked go to 3D viewer/preferences/preferences/General/
- Un-tick – show solder mask layers



Adding Copper to Top & Bottom Layer - Continued







Making Gerber Files

- Go to – File/Plot
- Make sure that plot format is in - Gerber
- Go to – Output directory and create and save a file in the project folder – Call it 'gerber files'
- Hit – Plot. This will create your gerber files
- Next hit – Generate Drill Files
- You may need to change the Drill Units to Millimetres
- Close
- Go to Gerber file and place all of the files in a zip folder
- Test by loading the gerber files into JLCPCB