Front Panel Instructions

Link to YouTube Tutorial

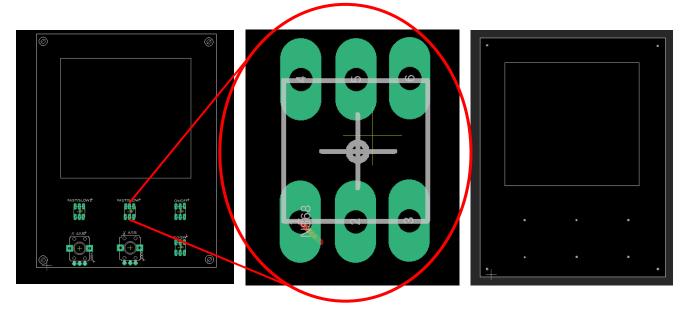
Eagle

Designing the PCB

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

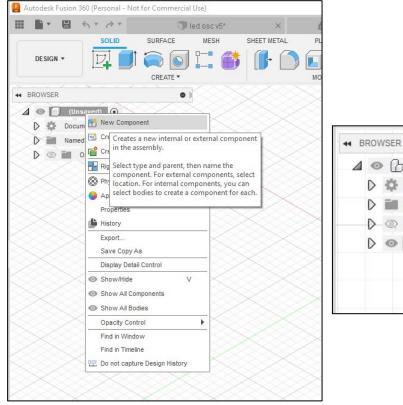
Creating a Template From the PCB

- 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.
 - o Draw/circle
 - o 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



Creating a new sketch

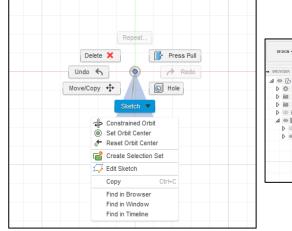
- Place curser over 'unsaved' and right click
- New Component
- Name new component
- Hit OK
- You only need to see the top plain for the sketch so hit 'top' in the box in top RHS
- Make sure that you are in the 'new component' section by clicking the small circle on the right



H BROWSER	•
 (Unsaved) Document Settings Named Views Origin 81 LED Chaser:1 	

Finding the sketch!

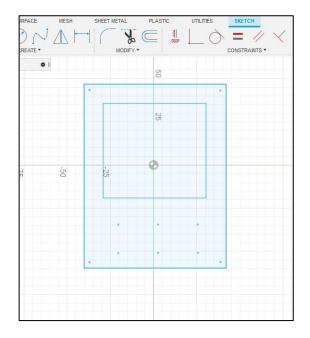
- If the 'sketch' screen isn't showing then do the following:
 - Right click centre circle
 - Click 'sketch'
 - Click on any of the shape suggestions



c	REATE *	MODIFY *		
BROWSER	•)			
🖉 🐵 🔓 81 LED Chaser v2				
Document Settings				
Named Views				
D iiii Selection Sets			50	
D 🕸 📷 Origin				
🖉 💿 🚺 81 LED Chaser:1 💿				
D 🕸 🖬 Origin				
D 🐵 🎆 Sketches				
			0	
-150	-100	-50	<u> </u>	

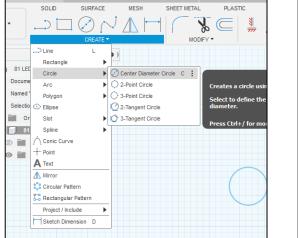
Importing the Eagle DFX file to Fusion 360

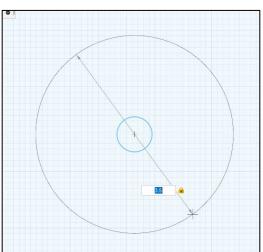
- Go to Insert/DXF/select DXF file
- File should load to screen
- Hit 'ok'



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

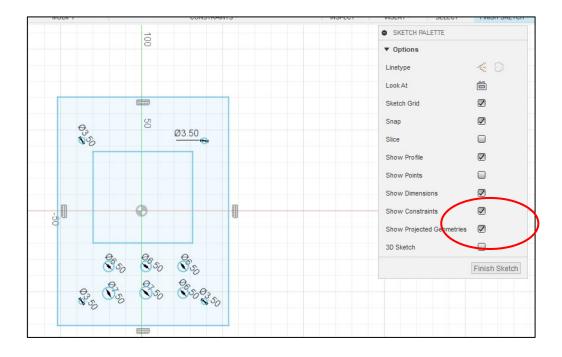




- Highlight the smaller circle (the one that was made in Eagle)
- Hit delete and remove the circle that was added to the board

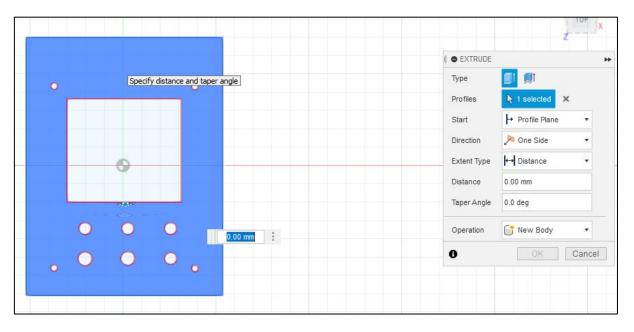
Saving and Extruding the Finished Sketch

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



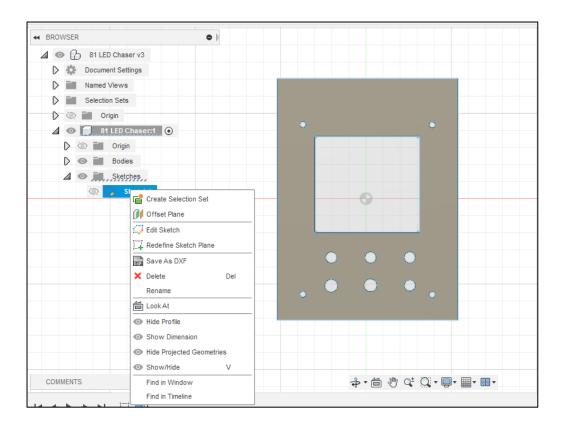
Saving and Extruding the Finished Sketch

- 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



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

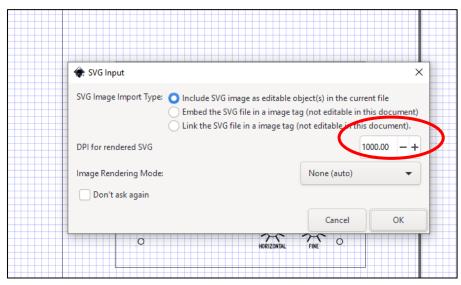


Save As DXF	×
\leftarrow \rightarrow \checkmark \bigstar Pictures \Rightarrow 81 LED Chaser	✓ O Search 81 LED Chaser
Organize 👻 New folder	
 Fusion LED Oscilloscopi OneDrive - Persor This PC 3D Objects Desktop Documents Downloads Music 	
File name: Fusion 360 Panel Save as type: DXF Files (*.dxf)	~ ~
	Save Cancel
∧ Hide Folders	

INKSCAPE

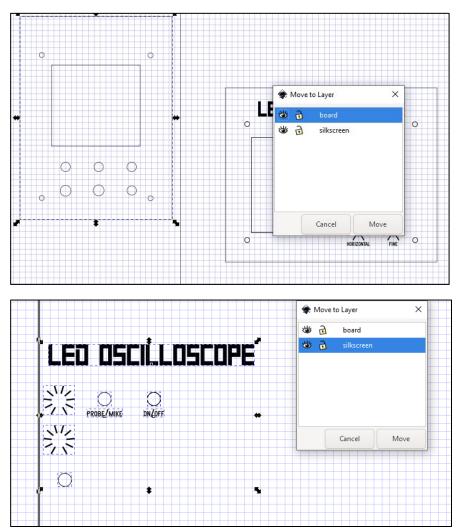
Adding the DFX file to Inkscape

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



Layers

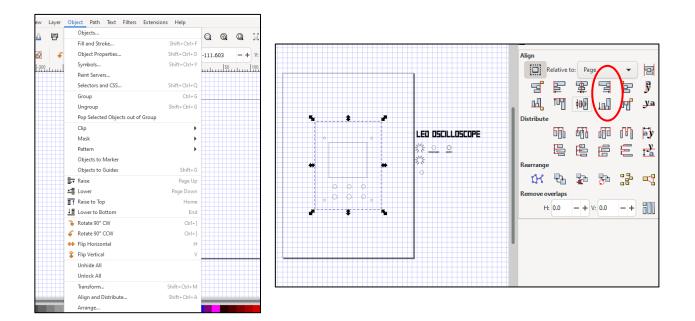
- Save the DXF file to one layer(call it board)
- Save any graphics to another layer (call this silkscreen)



INKSCAPE

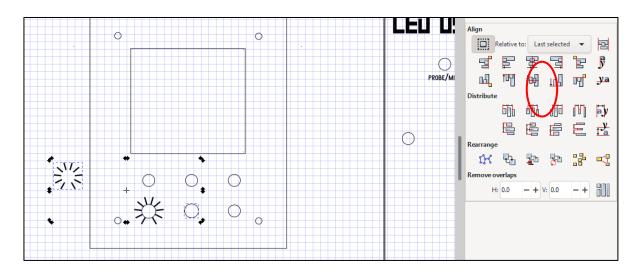
Aligning Board to Centre of Page

- 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

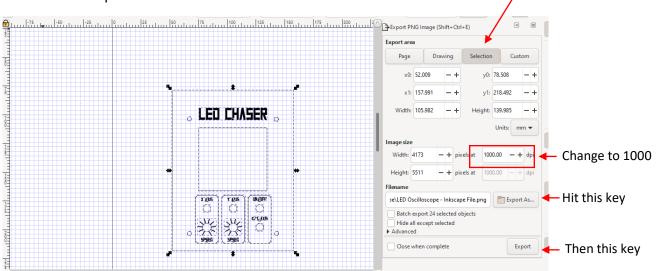
Making the Background White

- 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

Page	Guides	Grids	Snap	Colour	Scripting	Metadata	License	
General								
Display units:	mm 🔻							
Page Size								
A4		210.0	x 297.0 mm					
US Letter		8.5 x	11.0 in					
US Legal		8.5 x	14.0 in					
US Executive		7.2 x	10.5 in				Sackground cold	our ×
A0		841.0	x 1189.0 mm					
A1		594.0	x 841.0 mm				RGB HSL HS	V CMYK Wheel CMS
A2		420.0	x 594.0 mm					
A3		297.0	x 420.0 mm				R:	255 — +
A5		148.0	x 210.0 mm				G:	255 - +
A6		105.0	x 148.0 mm				0.	
Arr Orientation:		74.0	105.0				B:	255 — +
Custom size							and the second sec	•
_							A:	100 — +
Width: 297	.00000	- +						
Resize page	ge to conter	ıt						
Scale								RGBA: ffffffff
Scale x: 1.0	. 00000	- +				Scale y:	1.00000 -+	
Viewbox								
Background								Border
Checker	board backg	round						Show page border
Background	color:							Border on top of drawing
-	L			Ma	ke sure	this is v	white	Show border shadow
Display								Border color:
🔽 Use anti	aliasing							

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'

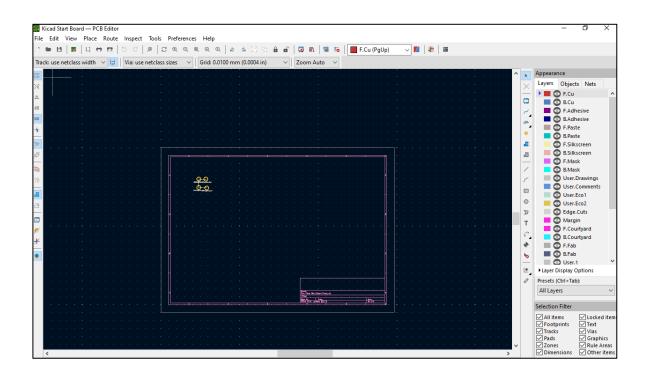


Selection should be on

<u>KICAD</u>

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

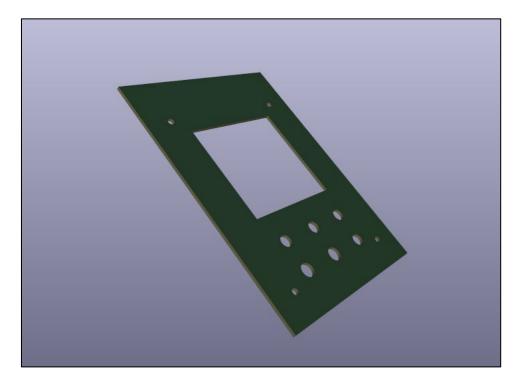
- File/Import/Graphics
- Make sure that the graphic layer is on 'edge cuts'
- Select the Fusion 360 file and hit 'ok'

🕜 LE	D Chaser - Kicad — PC	B Editor			Import Vector Graphics File	×
File	Edit View Place F	Route Insp	pect Tools Preference	es Help	import vector ordpines rice	~
B	Save	Ctrl+S	20000		File C:\Users\marcus\Pictures\81 LED Chaser\Fusion 360 Panel.dxf	
В	Save Copy As		use netclass sizes 🛛 🗸	Grid: 0.0100 mm (0.000	Placement	
0	Rescue				Interactive placement At X: 0.000000 Y: 0.000000 Units mm	
Ò	Import	>	Netlist			
ō	Export	>	Specctra Session		Import Parameters Graphic layer: Edge.Cuts	1
	Fabrication Outputs	>	Graphics	Ctrl+Shift+F	Import scale: 1.000000	
•6	Board Setup				Group items	_
Ľ	Page Settings		· · · · · ·			
Ð	Print	Ctrl+P			DXF Parameters Default line width: 0.200000 mm	
P	Plot				Default units: Millimeters V	
	Close		· · · · · ·		OK	
1000						

<u>KICAD</u>

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

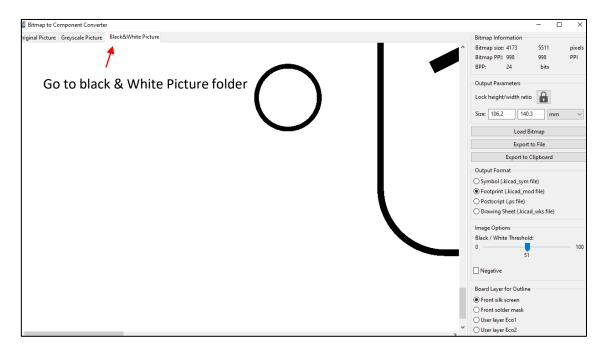
- First go to image converter in the front page of Kicad
- You now need to import the Inkscape front panel image
- Go to 'Load Bitmap'

Bitmap to Component Converter		– 🗆 X
Choose Image ← → ✓ ↑	∠ Search LED Oscilloscope	Bitmap Information Bitmap size: 0000 0000 pixels
Organize New folder		Bitmap PPI: 0000 0000 PPI BPP: 0000 bits
LED Oscilloscopi OneDrive - Person Footprint Gerber LED Os	cilloscope cape File EED Oscilloscope front panel	Output Parameters Lock height/width ratio Size: 0.0 0.0 mm ~ Load Bitmap Export to File Export to Clipboard
	Image Files (*.bmp;*.png;*.jpg;' V Open Cancel	Output Format Symbol (kicad_sym file) Footprint (kicad_mod file) Drawing Shet (kicad_wks file) Image Options Black / White Threshold: 0 100 50 100
		□ Negative

<u>KICAD</u>

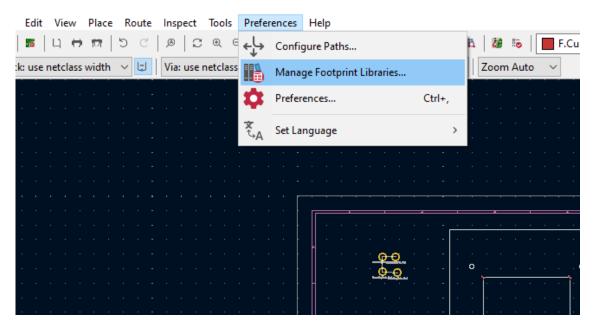
Adding Silkscreen Graphics onto the Board - Continued

- 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

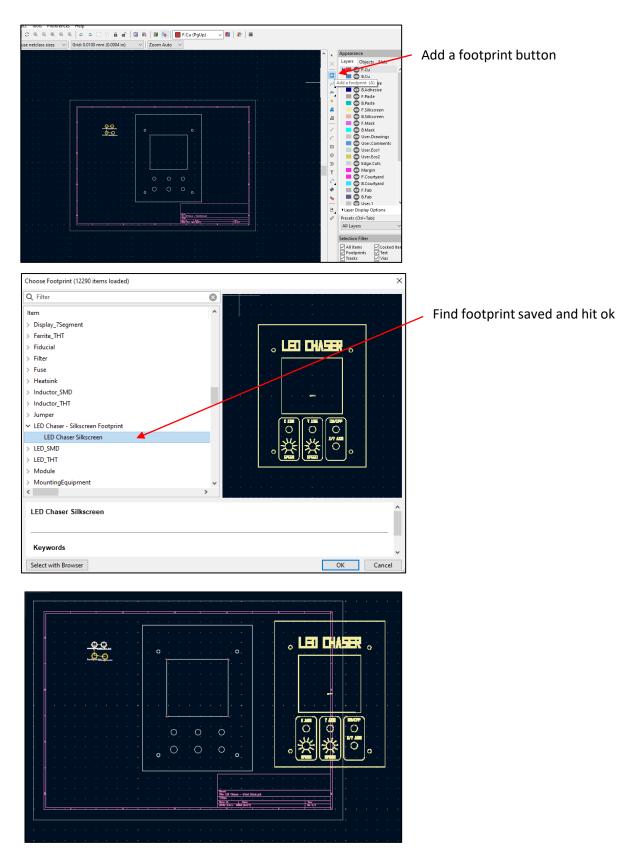
- Go to preferences/manage footprint library
- Next go to project specific library
- Hit the folder button
- Load <name of project> Silkscreen Footprint
 - Just select the folder don't load what is inside of it
- Hit ok



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



KICAD

Removing G*** From the Silkscreen

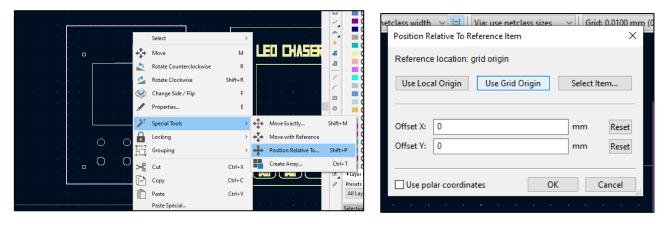
- Double click into the silkscreen
- Un-tick the box next to the G***
- Hit Update footprint from library
- Hit Update



eneral Clearance Overr	ides and Settings 3D Models						
	Text Items	Show W	idth Height	Thickness	Italic	Layer	
Reference designator	G <mark>***</mark>	1.524	4 mm 1.524 mm	0.3 mm		F.Silkscreen	
Value	LOGO	1.524	4 mm 1.524 mm	0.3 mm		F.Silkscreen	

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

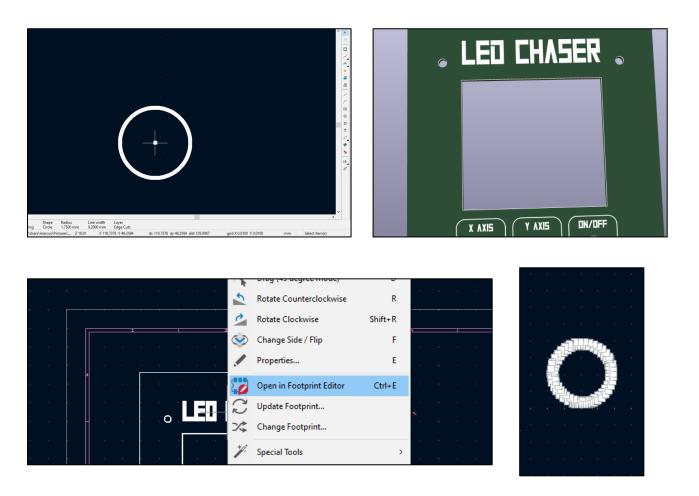




KICAD

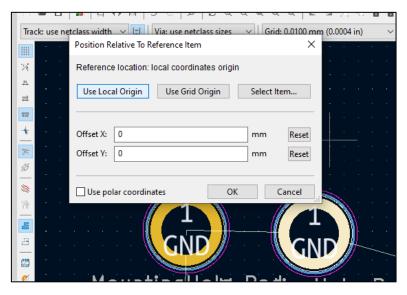
Editing the Footprint to Remove Borders, Holes etc

- 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





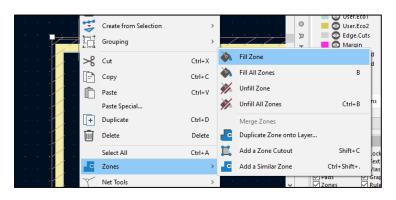
Adding Copper to Top & Bottom Layer

- 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

Kicad Start Boar	d — PC	BEditor					
File Edit View	Place	Route Inspect	Tools	Preferences	Help		
) 🛏 🗎 🗖	[••••]	Add Footprint			А	1	4 [6] 6
Track: use netclass	0	Add Vias		Ctrl+	Shift+V) mm	(0.0020 in)
	e	Add Filled Zone		Ctrl+	Shift+Z		
in .		Add Rule Area		Ctrl+	Shift+K		
<u>vil</u>	ሇ	Add Microwave Sł	nape		>		

Copper Zone Properties \times Tick - F.Cu Layer -Net F.Cu Filter Hide auto-generated net names Sort nets by pad count B.Cu <no net> Click on - GND Change to - 0.2 Change to - Solid Electrical Properties Fill General Fill type: 0.2 Solid fill Zone name: Clearance: mm \sim -0 0.254 Zone priority level: 0 Minimum width: mm Orientation: deg Hatch width: 1.016 mm Pad connections: Solid ~ Shape Hatch gap: 1.524 mm Constrain outline to H, V and 45 degrees Thermal relief gap: 0.508 mm Smoothing effort: 0 * Locked Thermal spoke width: 0.508 mm * Smoothing amount: 0.10 Hatched \sim Outline display: Always Remove islands: \sim Corner smoothing: None \sim Minimum island size: 0 0 Fillet radius: mm Export Settings to Other Zones OK Cancel

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'

 \sim

- Hit Plot. This will create your gerber files
- Next hit Generate Drill Files
- You may need to change the Drill Units to Millimetres
- Close

let

- Go to Gerber file and place all of the files in a zip folder
- Test by loading the gerber files into JLCPCB

			~		
lot format: SVG ~	Output directory:/Gerber/				
Include Layers	General Options				
F.Cu 🔺	Plot border and title block	Drill marks:	None 🗸 🗸		
B.Cu F.Adhesive	Plot footprint values	Scaling:	1:1 ~		
B.Adhesive	Plot reference designators	Plot mode:	Filled \sim		
✓ F.Paste Ø B.Paste	Force plotting of invisible values / refs	Use drill/place file origin	1		
F.Silkscreen	Plot Edge.Cuts on all layers	Mirrored plot			
✓ B.Silkscreen ✓ F.Mask	Sketch pads on fabrication layers	Negative plot			
B.Mask	Do not tent vias	Check zone fills before p	Check zone fills before plotting		
User.Drawings	SVC Ortiger				
User.Eco1	SVG Options Units: Millimeter V Precision: 6				
User.Eco2 🗸	Units: Millimeter V Precision: 6 💌				
Output Messages					
		_			
Show: All Frrors	Warnings O Actions	✓ Infos	Save		
Run DRC (7 known DRC	Cviolations; 0 exclusions)	Plot Close G	enerate Drill Files		

