Front Panel Instructions

Link to YouTube Tutorial

https://www.youtube.com/watch?v=UOQezMJ560o&t=4321s

<u>Eagle</u>

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



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



D	ESI	IGN -	•			~√ <u>∧</u>	$\left - \right $	MO	¥. C	 LÒ	CONSTRA	//		INSERT *	SELECT
BRO	ows	SER				•									
4 <	•	ß	81	LED Chaser v2											
D	1	¢.	Dec	ument Settings											
D	i,	in i	Nam	ed Views											
D	i	in i	Sele	ction Sets						5					
D		@ i	in i	Origin											
4	1	0		81 LED Chasers	1 💿										
	D	8	2	Origin											
	D	> <	Ð	Sketches											
										0					
				150		100		6	5						

• ||

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



Autodesk Fusion 300 (Personal - Not for Commercial Use)	
🛃 Save As DXF	×
$\leftarrow \rightarrow$ \checkmark \bigstar eictures \rightarrow 81 LED Chaser	✓ ♂ Search 81 LED Chaser
Organize 👻 New folder	■ ▼ ()
 Fusion LED Oscilloscopi OneDrive - Persor This PC 3D Objects Desktop Documents Downloads Music Pictures 	
File name: Fusion 360 Panel	~
Save as type: DXF Files (*.dxf)	×
∧ Hide Folders	Save Cancel
	• • •

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				Rackground col	lour X
A0		841.0) x 1189.0 mm					
A1		594.0) x 841.0 mm				RGB HSL H	SV 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					
Orientation		74.0	105.0				В:	255 — +
Custom size							in the second	•
custom size								100 — +
Width: 297	.00000	- +						
Resize page	ge to conter	ıt						
Scale							► O 🖊	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
D' 1	L			Ma	ke sure	this is v	white	Show border shadow
Display								Border color:
🕑 Use anti	allasing							

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

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 rector ordpines me	~
Β	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	
	Fabrication Outputs	>	Graphics	Ctrl+Shift+F	Import scale: 1.000000	i
•6	Board Setup				Group item:	_
	Page Settings					
Ð	Print	Ctrl+P			DXF Parameters Default line width: 0.200000 mm	
Pī	Plot				Default units: Millimeters ~	
Ċ	Close		· · · · · ·		OK Cancel	
1 con						

<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	×	Bitmap Information Bitmap size: 0000 0000 pixels
$\leftarrow \rightarrow \checkmark \uparrow$ 🔤 « Pictures » LED Oscilloscope » v Ö	> Search LED Oscilloscope	Bitmap PPI: 0000 0000 PPI
Organize 🔻 New folder	≂ - □ 2	BPP: 0000 Bits
S1 LED Chaser Camera Roll LED Oscilloscop OneDrive - Persor This PC J 3D Objects Desktop	LINCOR	Output Parameters Lock height/width ratio Size: 0.0 0.0 mm Load Bitmap Export to File Export to File
Documents Downloads Music Pictures Video V		Output Format Symbol (kicad_sym file) Foreit (kicad_mod file) Postscript (.ps file) Orawing Sheet (kicad_wks file)
File name: LED Oscilloscope - Inkscape File V	mage Files ("bmp;"-png:"-jpg: " Open Cancel	Image Options Black / White Threshold: 0 50 100

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



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



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



ciass wide		ICIdSS SIZES			~	Zoom Aut						1
🛗 Footj	print Properties										\times	1
General	Clearance Overrides a	nd Settings	3D Models									Ľ.
		,										
			Text	Items	Show	Width	Height	Thickness	Italic	Layer		1
Refe	rence designator	G <mark>***</mark>			\checkmark	1.524 mm	1.524 mm	0.3 mm		F.Silkscreen	-	
Value	2	LOGO				1.524 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





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

٩	Kicad Start Boa	rd — PC	B Editor								
File	e Edit View	Place	Route	Inspect	Tools	Preferences	Help				
٦	► Ľ 15	[***]	Add Foo	tprint			Α	2	2	Fil	6
Tra	ick: use netclass	•	Add Vias	5		Ctrl+	Shift+V) mm	n (0.0	020	in)
		e	Add Fille	ed Zone		Ctrl+	Shift+Z				
29. 21			Add Rul	e Area		Ctrl+	Shift+K				
nij		ሇ	Add Mic	rowave 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 🗸 🗸				
	Plot footprint values	Scaling:	1:1 ~				
B.Adhesive	Plot reference designators	Plot mode:	Filled \sim				
✓ F.Paste ✓ B.Paste	Force plotting of invisible values / refs	Use drill/place file origin					
F.Silkscreen	Plot Edge.Cuts on all layers	Mirrored plot					
✓ B.Silkscreen ✓ F.Mask	Sketch pads on fabrication layers	Negative plot Check zone fills before plotting					
B.Mask	Do not tent vias						
User.Comments	SVG Options						
User.Eco2 v	Units: Millimeter V Precision: 6						
Output Messages							
Show: 🗌 All 🛛 Errors	● Warnings ● Actions	✓ Infos	Save				
Run DRC (7 known DRC	violations; 0 exclusions)	Plot Close Ge	enerate Drill Files				

