Joystick Case Design Guide

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Joystick case manufacturing is planned using laser cutting machine with acrylic sheet of 3mm thickness, as the subtractive manufacturing process of laser cutting is faster than 3D printing. User will have the flexibility to design their own joints if a glue-free assembly for the six faces of the case is preferred.

First step of the case design process is to import the designed PCB from EsayEda software to Fusion 360. The screen shot of the Fusion design workspace is shown in Fig. 1.

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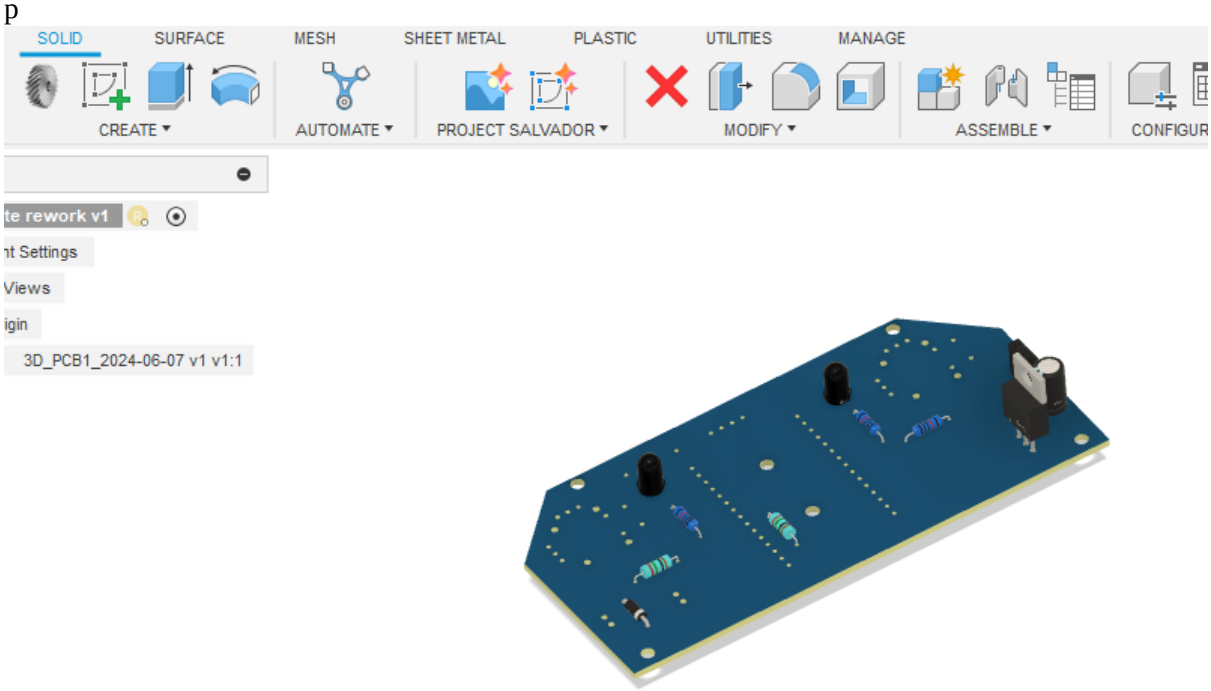

Fig. 1 Fusion workspace after importing PCB design from EasyEda

Now, make a simple case enclosing the PCB inside it. Model a solid case with no cavity initially and reduce its opacity to see the inside PCB as shown in Fig. 2a and Fig. 2b. Give fillets of around 15mm on each of the four corners.

Note: Leave enough distance between front face of the PCB and the Front face of the case, this is to accommodate the toggle switch perfectly.

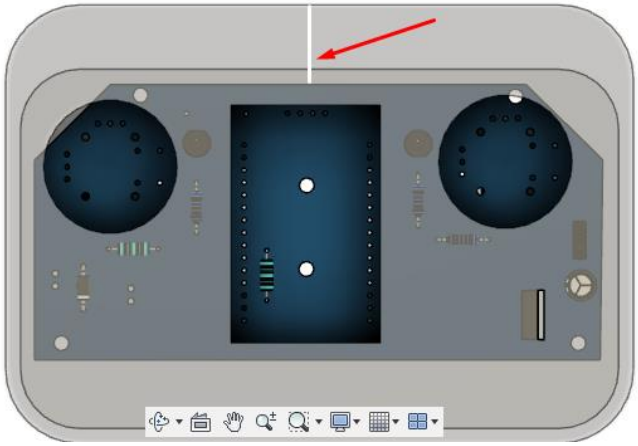

Fig. 2a. Red Arrow indicates the Space which should be Left from Top Side

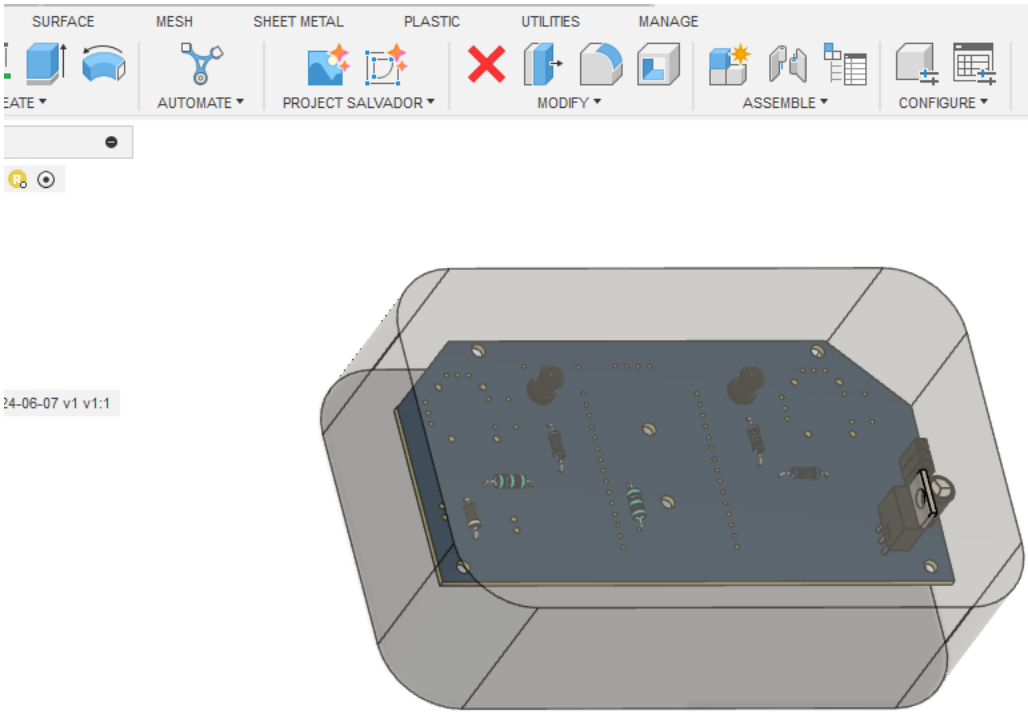

Fig. 2b. 3D Model of Joystick Case after Encompassing PCB in it

The basics dimensions which should be kept constant are as follows and also shown in Fig.3.

- Distance between the top face of the PCB and the bottom face of the top plate of case should be **15-15.4mm.** This dimension is chosen to provide adequate area for the joystick head movement. For a smaller joystick head, this dimension may further be reduced.
- Distance between bottom face of the case and the bottom face of the PCB should be **20mm.**

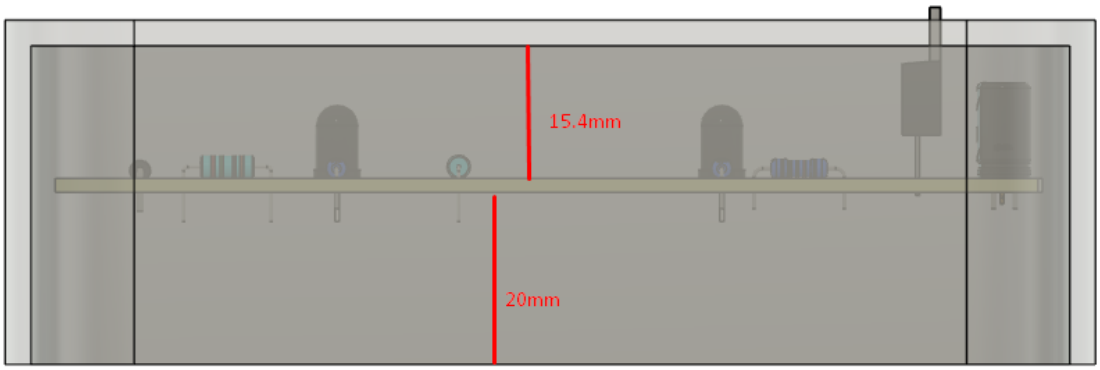

Fig. 3. Basic Dimensions to be kept Constant

Joystick Head Opening

For making holes for the joystick head, the centre of the circle has to be coincided with the centre of the joystick footprint. The Diameter of the circle is recommended to be in the range of **25mm to 29.5 mm.** The schematic of a PCB board in which the holes inside the red circle are the joystick's footprint as shown in Fig. 4 and by taking it as the reference, the centre of the joystick opening is marked as shown in Fig. 5.

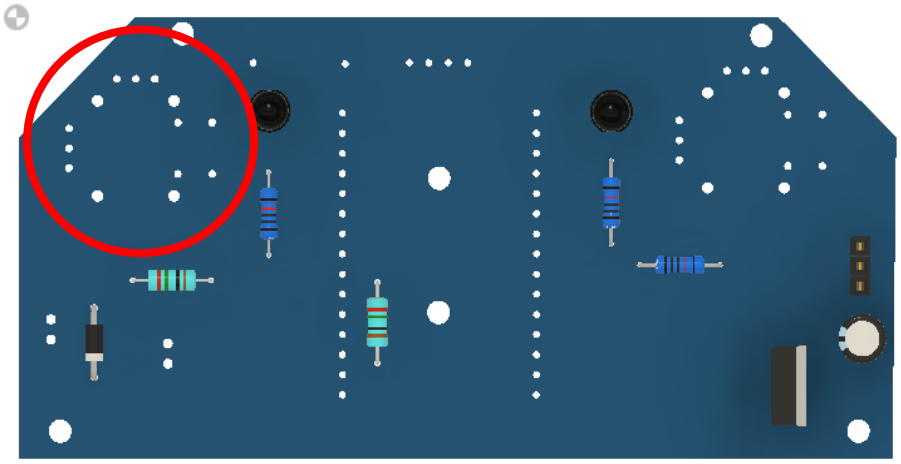

Fig. 4 Joystick Footprint on PCB Board

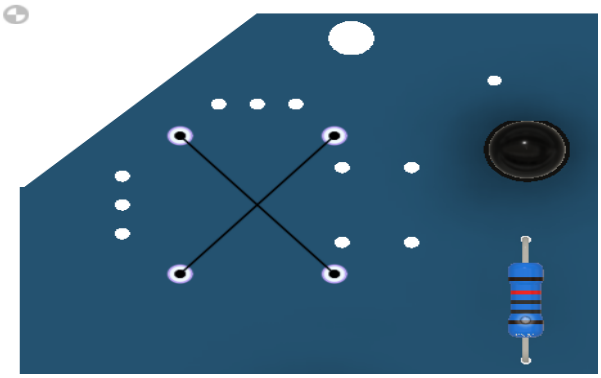

Fig. 5 Make Diagonal as shown to get the Centre Point

- Make a sketch and project the four holes using Project Feature (**Short cut – P**). These four holes are the mounting leads of the joystick, so even if the joystick is not symmetrical the four mounting points will always be at the centre of the joystick head.
- Sketch two lines as shown in black as shown in Fig. 6, like the diagonals of a virtual rectangle. The intersection point will be the centre of the circle and will later serve as the sketch for making opening of joystick head on the upper face of the case.

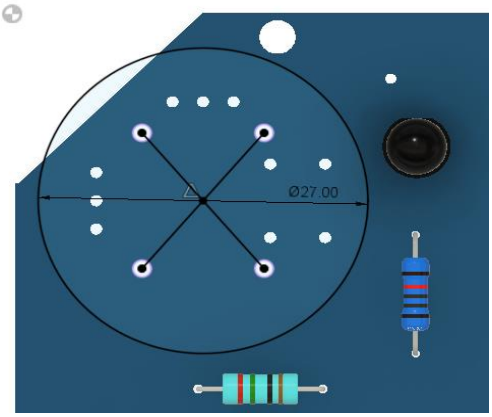

Fig. 6 Circle for Making opening in Transmitter Case

- Make the circle from the centre whose diameter will range from **27mm-29.5mm** depending on the PCB design. In this way the opening is exactly at the centre of the joystick head.
- Similarly make an opening for the other joystick. **Don't use mirror commands,** because the PCB may not be symmetrical.
- Make openings for the **OLED display wires, Toggle switch (6mm Dia) and the battery** by projecting the respective element. The cad model of these elements can be downloaded for taking the exact dimension of the opening.

After completing the necessary openings, the controller case design is shown in Fig. 7. The rectangular opening at the top is given for the ESP 32. ESP 32 (including its micro-USB port) is designed to be projected outside of the top face for better functioning and heat dissipation of the controller device.

Fig. 7. 3D Model of Remote Case after giving All the Openings

Next step is to make the opening for the battery holder at the bottom face of the case as shown in Fig. 8. For battery holder opening, download battery holder cad model shared on moodle. After this, make two sketches, one for top plate and other for the bottom as shown in Fig. 9a and Fig. 9b respectively and save as DXF.

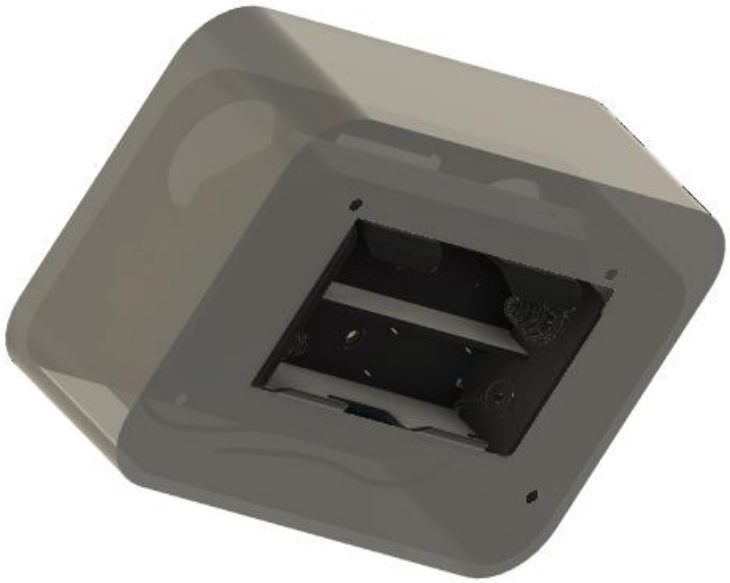

Fig. 8. Bottom View of Remote Case after giving all the Openings

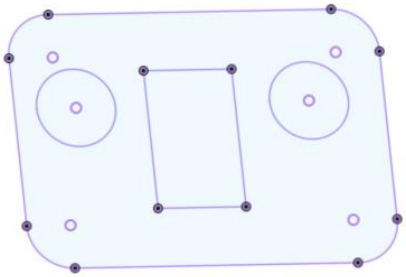

Fig. 9a Top face plate sketch

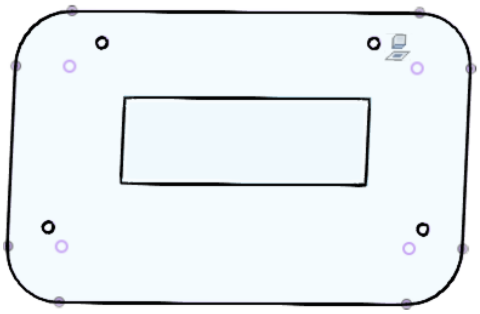

Fig. 9b Bottom face plate sketch

The link given below is used to create finger joints for assembling the box. The screen shot of the UI of the link is shown in Fig. 10. Take end-to-end dimensions of the box created as is shown in Fig. 11. Enter the dimensions in the dedicated input fields. Also enter the value of corner radius same as the design value used in Fusion 360 and keep rest of the settings same. Then click on generate button and download the dxf.

Link: https://boxes.hackerspace-bamberg.de/RoundedBox?language=en

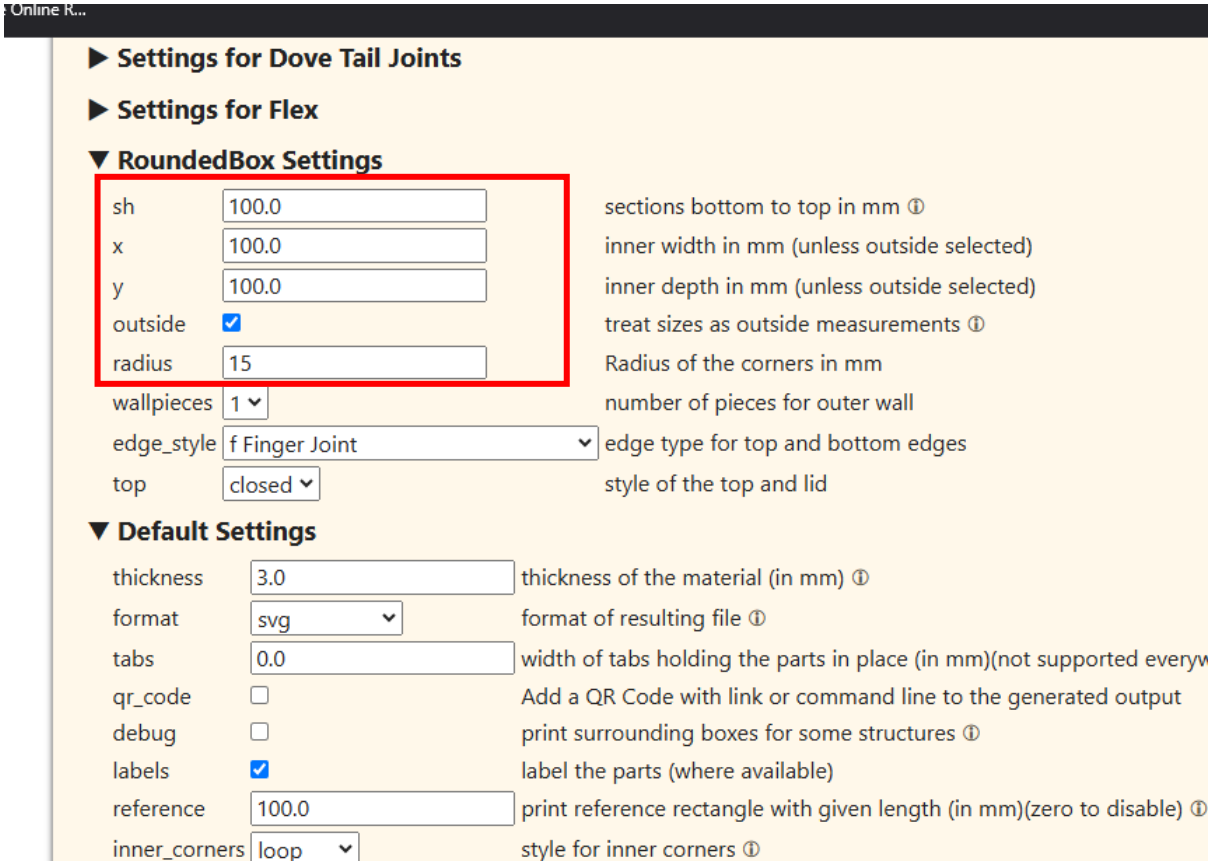

Fig. 10 User Interface of the boxes.py, keep rest of the settings same, just enter values in (sh,x,y and radius and change format from svg to dxf)

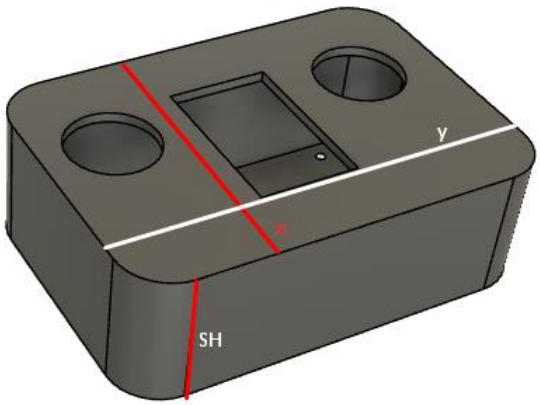

Fig. 11. Red and White Lines indicates the Dimension to be taken for Further Processing

Now, there are 3 dxf files in total to process, 1st top plate, 2nd bottom plate and 3rd whole box with finger joints and kerfing flexes, named as top plate.dxf, botton plate.dxf and FlexBox.dxf respectively as shown in Fig. 12. It is better to make a separate folder and save all the 3 DXFs in that folder which is to be used for laser cutting.

Fig. 12. Folder with all the 3 files required for making Laser Cut Case

Now open the LaserCad software and import the downloaded dxf file named 'FlexBox'. The screen shot is as shown in Fig. 13. The two upper blocks will be the top and bottom plate respectively.

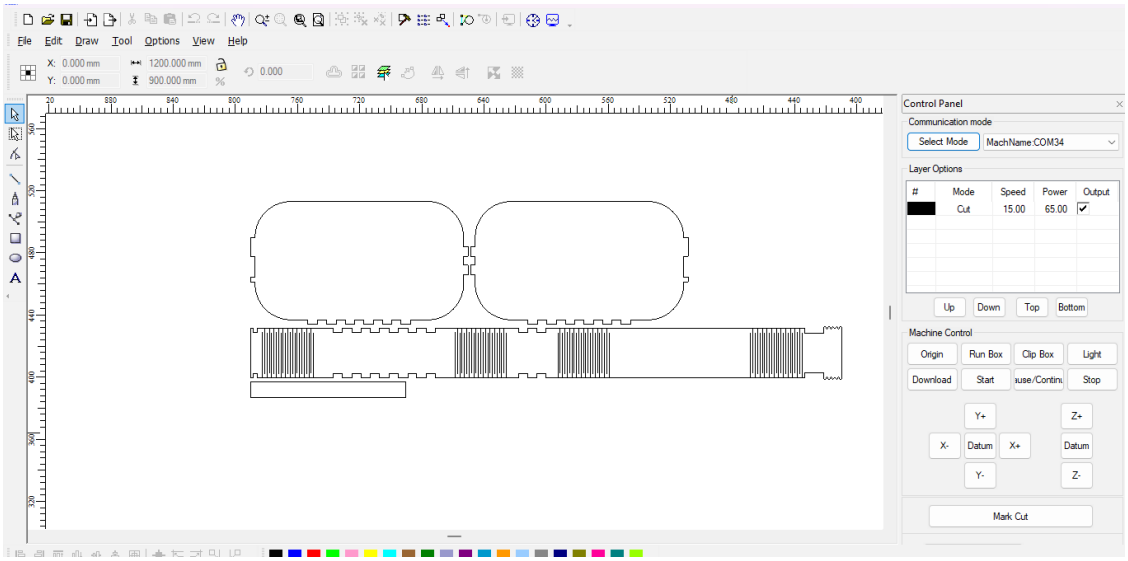

Fig. 13 DXF of Flex Box after importing it into Laser CAD

Next step is to align the DXFs from fusion 360 to each of the block. The top plate alignment process is shown in Fig. 14 and Fig. 15 respectively. Align the boundary with at least two sides of the fingers (from downloaded dxf). Here, alignment is such that Fusion file in red colour boundary match with flex box in black colour boundary. After alignment delete the outer boundary of the Fusion sketch and the downloaded sketch block with finger structure will become the new boundary as shown in Fig. 15.

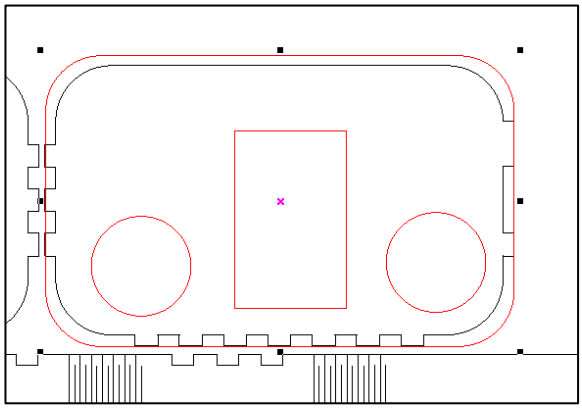

Fig. 14 Sketch in Red is top Top Face Plate Sketch imported from Fusion 360

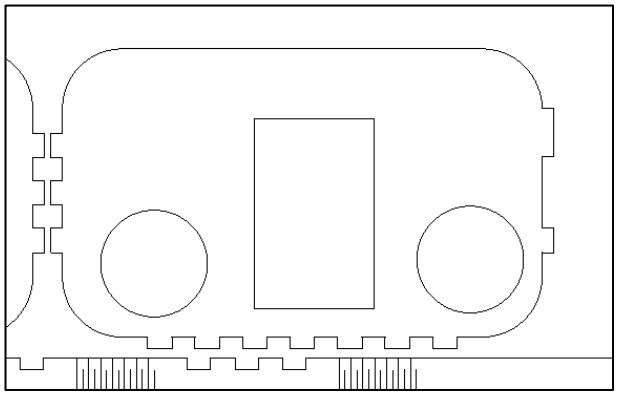

Fig. 15 DXF after deleting the Boundry of Top Face Plate Sketch, while leaving the ESP and joystick openings

Repeat the same kind of alignment with Bottom Plate as shown in Fig. 16. Not to change the orientation of any of the sketch, as all the sketches are already adjusted with respect to each other.

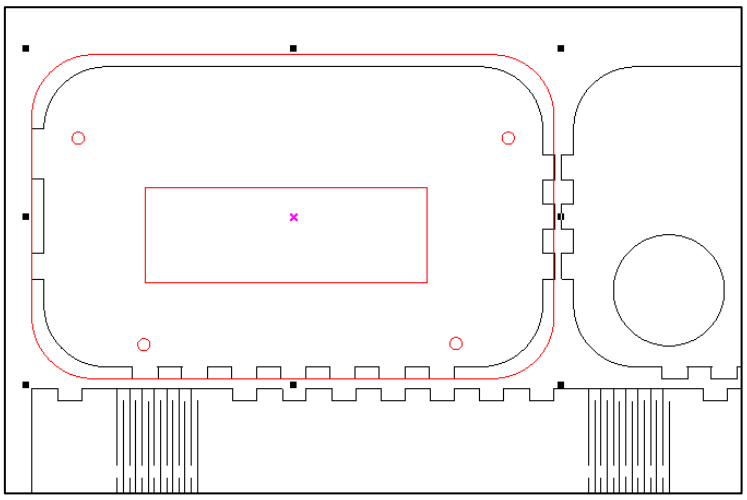

Fig. 16 Sketch in RED is Bottom Face Plate Sketch

The final modified sketch is shown in Fig. 17. The red rectangle can be deleted from the sketch. Enter the speed and power values as 17mm/sec and 60 watts respectively for 3mm sheet in the control panel (screen shot is shown in Fig. 18) and download the ***.ud5** file. Now the file is ready to be laser cut and save it into pen drive with extension **ud5**.

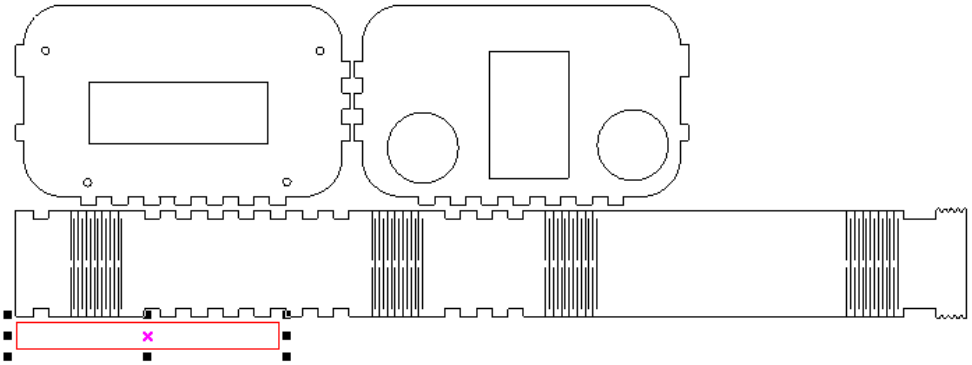

Fig. 17 The RED Rectangle can be deleted

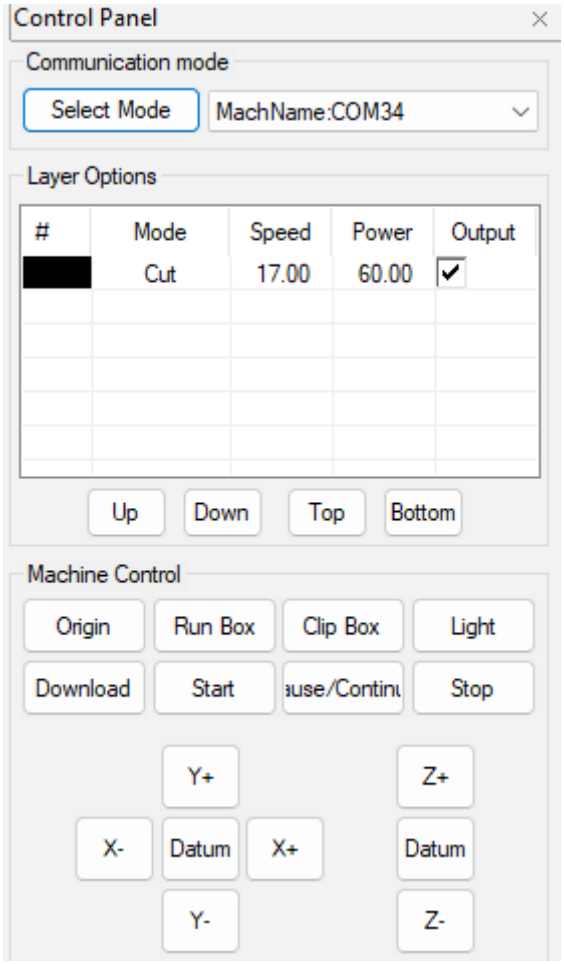

Fig. 18 Speed/ Power Setting for Laser (for 3mm Acrylic Sheet)

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References

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2) https://boxes.hackerspace-bamberg.de/ClosedBox?language=en

3) https://lastminuteengineers.com/joystick-interfacing-arduino-processing/