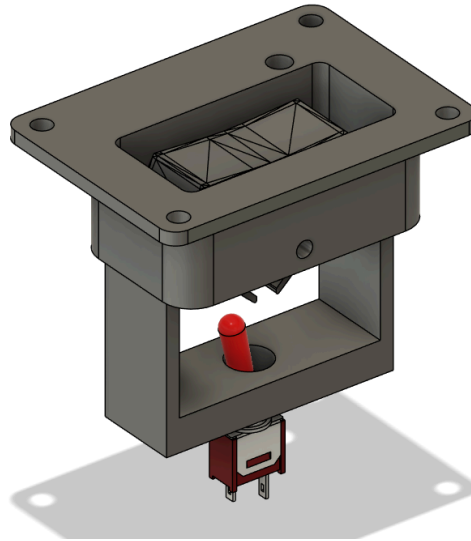


MAKING THE ELT

By StarDesigns
Discord ID: starthe2nd
4/05/24 - last updated



Bill of Materials

- All the printed parts
- 2mm metal rod Or a toothpick can be used as your hinge x1
- SPDT switch (ON OFF ON) x1
- 3mm Red LED 5V or 5V-12V, prewired makes your job way easier x1
- Arduino nano/uno/mega 2560 x 1
- Hot glue stick x 1

- 8x M3 Bolts or self tapping screws at a length of 8mm or longer

- Alternatively if you're mounting to a panel:
4x M3 Bolts - Panel thickness + 8mm for one lot.
4x M3 Bolts - 8mm or longer.

**Example - If my panel was 10mm thick $10 + 8 = 18$ mm M3 Bolts x 4
Then just 4x M3 Bolt x 8mm or longer for the other set.**

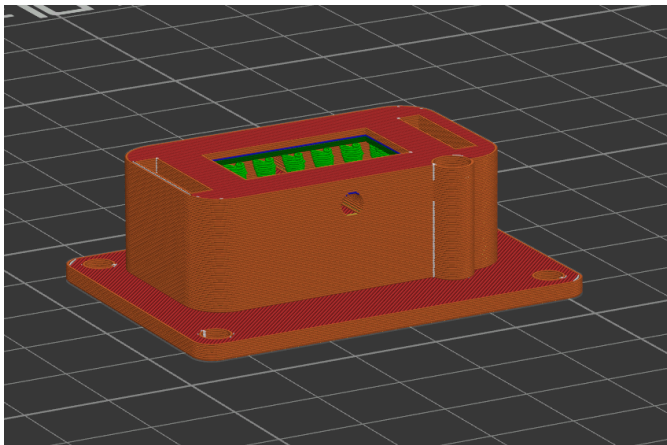
Hardware Needed

- Hot glue or any type of glue to keep the toothpick/steel rod in place and to fix the backplate to the front plate.
- Soldering iron
- 3D Printer

PRINT INSTRUCTIONS

ELT Face Plate

Print Orientation



Using this print orientation on a smooth surface bed should allow for the best visible face possible once painted. If you are using a textured build surface and do not have access to a smooth build surface I would recommend printing the object 180 degrees and having the supports on the exterior side. This would give you a smoother front layer, additionally you would have a smoother set of internal layers for the inside but these are barely visible.

Slicer Settings

Use any slicer settings you would use for an ordinary print, infill doesn't matter here. Ideally a high quality preset will be in use. Supports are required to print it in any orientation, I prefer to use tree supports as I believe they're more easily removable but you can also use traditional supports with ease.

Multi filament 3d printer users: If your machine has multifilament capability e.g. Bambulab AMS, or Prusa MMU2-3, then you are able to have a better internal layer as you can set your top Z distance to 0 and make your raft interface PLA if your part is made out of PETG, or PETG support interface if your part is PLA.

Example: [\(8\) Using pla as support material with petg #3dprinting #bambulab - YouTube](#)

Print Color

Intending to laser engrave text onto the front face

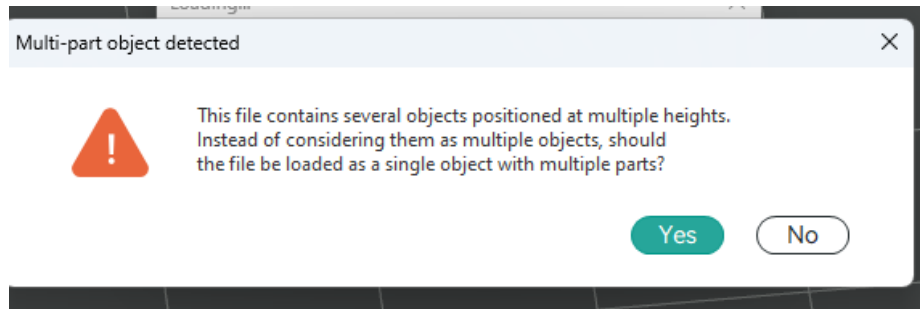
Print the faceplate in white filament or the filament choice you want to have showing underneath. Note here that backlighting using this method will not be possible, although there is an alternative solution of making the front face out of 3mm opal acrylic using a CO2 laser or a CNC, but assuming you have either machine you should also have the skillset to modify the provided fusion 360 file in order to make the necessary modifications, documentation and a configuration for this is in WIP.

Not intending to laser engrave text onto the front face

Print in black to provide the least work for you. There is a chance if you print in black and your bed level is very good on a smooth surface that you will not require sanding or painting for your first layer. You can also optionally print in white as well but there is little purpose for doing so.

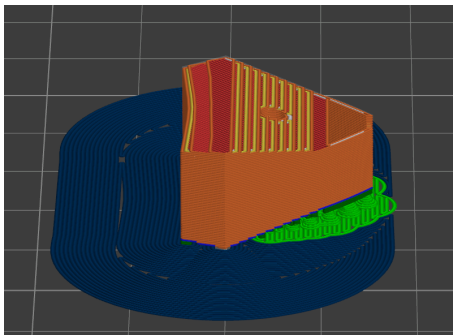
ELT Switch

Notice:



If you get this error press "Yes" this will be fixed eventually in a future revision.

Print Orientation



Print either this way or 180 degrees of this. This part was designed pretty poorly for 3D printing and will be fixed in a future revision when I have time. Requires supports

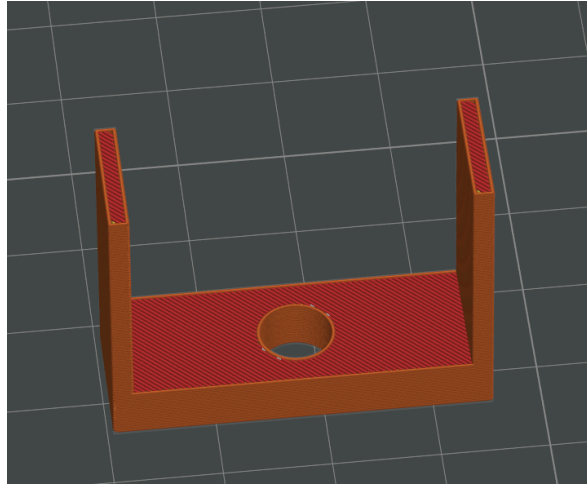
Slicer Settings

This model requires support settings to be on. Infill should be denser than usual, 50% or above should give you a very long life. Bed might need glue here or a good stick if you use a textured plate. A nice slow preset to make a nice even wall distribution for the front where your fingers would feel or a low layer height would be great to improve overall feel but not required whatsoever.

Multi Material Users: Can also use the interface material here if you use PLA then use PETG as your interface or if you're printing these out of PETG use PLA as your interface material.

ELT Back Plate

Print Orientation



Just the easiest method of printing, that's all.

Slicer Settings

Use any slicer settings you would use for an ordinary print, infill doesn't matter here.

Print Color

Match the filament color whatever the print color of the ELT Face Plate will be. If you intend to enclose the unit it will be pitch black inside of the ELT enclosure and therefore the Back plate will not be seen whatsoever but for good measure it is just easiest to use the same color. Using a separate color probably will be fine also.

ELT Shroud

This Part is a WIP, this is not designed yet to be placed in a panel although I believe it should work fine. There is also no place to put an arduino inside the case to save on length. There is a hole on the back cover to route out wires for the electronics to an arduino separately on your panel.

Print Orientation

No image required, just place it vertical, any direction both are supportless.

Slicer Settings

Use any slicer settings you would use for an ordinary print, infill doesn't matter here.

Print Color

Black, should not be visible though and definitely won't be visible if you mount it to a panel, but black is most "realistic"

ELT Shroud Back Cover

Print Orientation

No image required, any direction all don't require support material, however I strongly recommend laying it flat on the bed for print stability.

Slicer Settings

Use any slicer settings you would use for an ordinary print, infill doesn't matter here.

Print Color

Black, should not be visible though and definitely won't be visible if you mount it to a panel, but black is most "realistic"

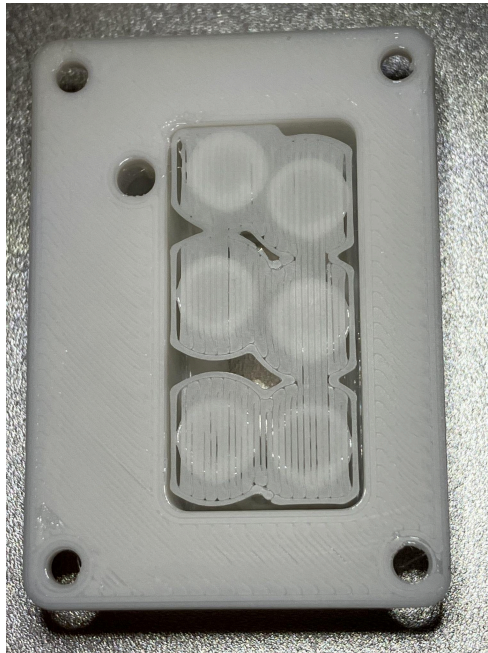
ASSEMBLY INSTRUCTIONS

Post Processing

1. Remove Supports off off the parts containing supports

- ELT Face Plate
- ELT Switch

Face Plate



Example of the supports found on the Face Plate - Easiest method to remove these is via the back, push down until the interface material releases, once a bit has released you should be able to pull the rest. Use a tool to remove the supports if they are jammed in hard. If your supports are impossible to remove I suggest **moving your z top distance for supports down to 0.2 or lower.**

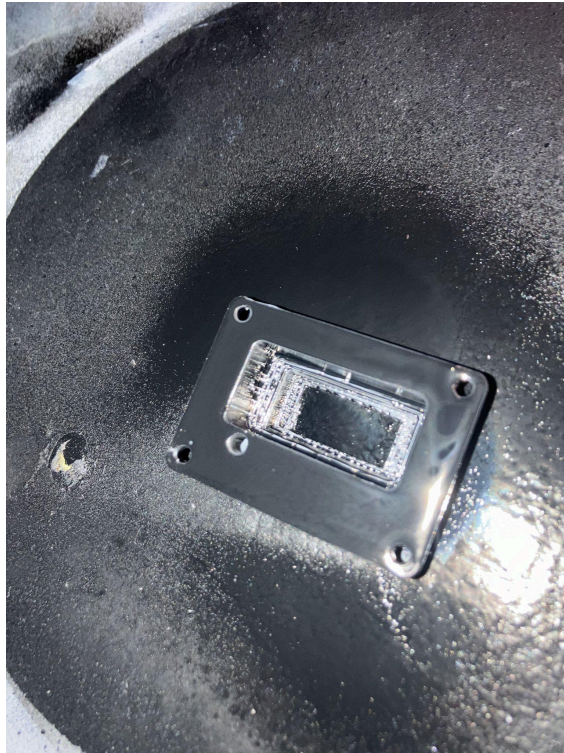
ELT Switch

No image

The supports should auto release off of the print from the build platform if you're using a brim, if not, they should be super simple to rip off. The area may be pretty messed up but the looks will not matter a large amount, some sanding can help a bit and will help a bit with less friction but I didn't need to-do this for any of my various prototypes.

Painting (optional)

If you decided you wanted to paint your parts then great! There is a strong likelihood that you have already previously painted 3d printed parts before, the method I choose to always paint parts is spray paint. It is cheap, easy, fast, and gives a great result. There is also a strong likelihood that you have previously used spray paint before, spray painting 3d printed parts is exactly the same as spray painting wood or any other material. Go light on the coats if you intend to engrave text as 3d printed parts tend to bleed paint in. However, do as many coats of paint as you need (I needed 2 plus a clear coat on top to help with the paint's longevity.)



This was my result after 15 minutes of drying and just after applying my clear coat. As you can see, using the smooth surface means that the layer lines are hidden quite well. Good enough for me for now. The interior isn't fully covered but that isn't seen very well and that is just a rough surface reflecting on the phone's flashlight mainly.

[Image of finished painting](#)