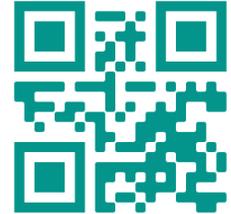


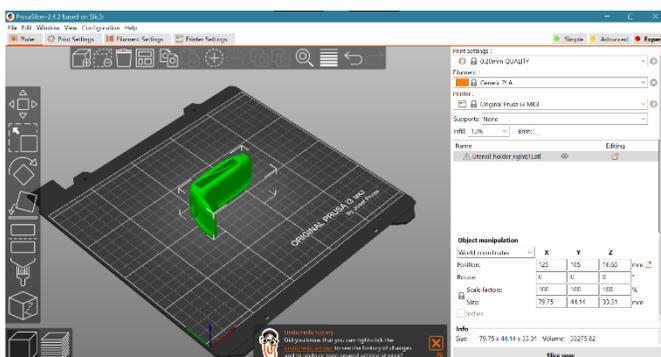
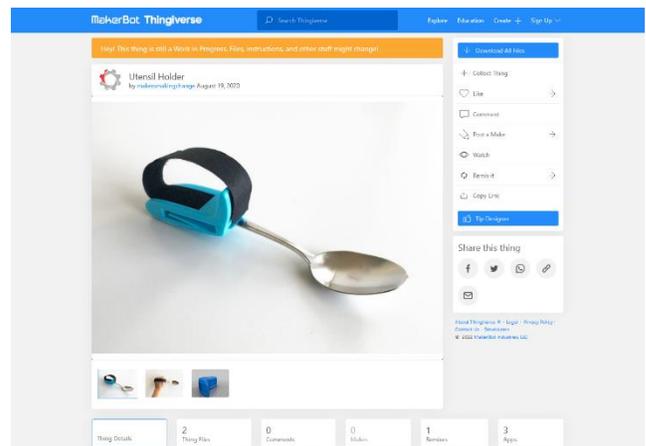


This document is designed to serve as a general introduction to 3d printing. Other useful guides and videos related to more specific topics can be found at OT3d.org



Just to get started, here is a picture of a Prusa Mini, a smaller FDM printer that is at my local makerspace. Let's go through its parts to learn about how a 3d printer works. 'FDM' stands for *fused deposition modeling*, a term that is associated with printers using a thermoplastic filament.

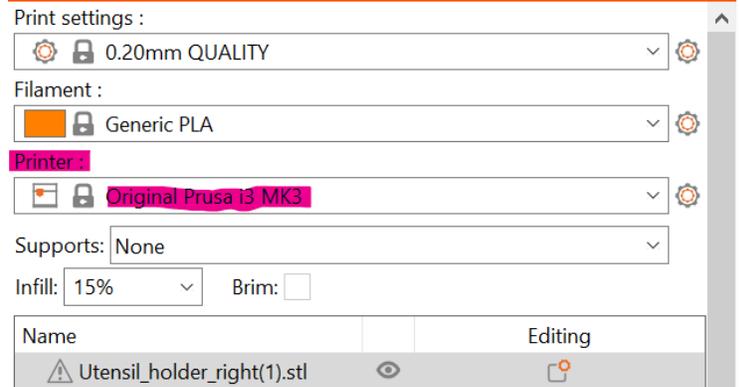
When we want to make a 3d Print, we will start with a design. You can use websites like thingiverse.com to find existing designs, or you can visit websites like tinkercad.com if you would like to make your own. You will download a file ending with .stl or .obj



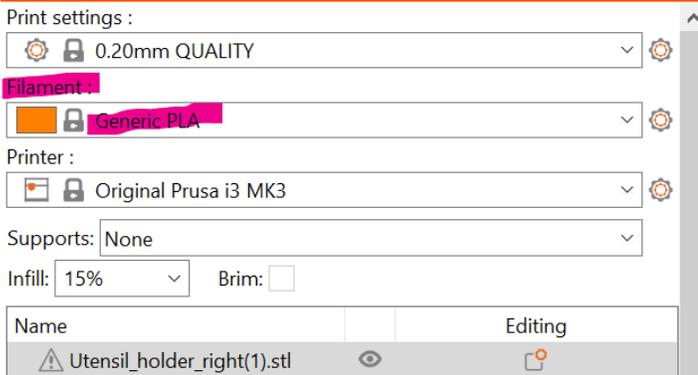
Once you have your file, you will want to upload it into a 'slicer' program. In this demonstration, we will use the Prusa slicer, although other options like Cura will work as well. Both are available to download for free from the internet on their respective websites.



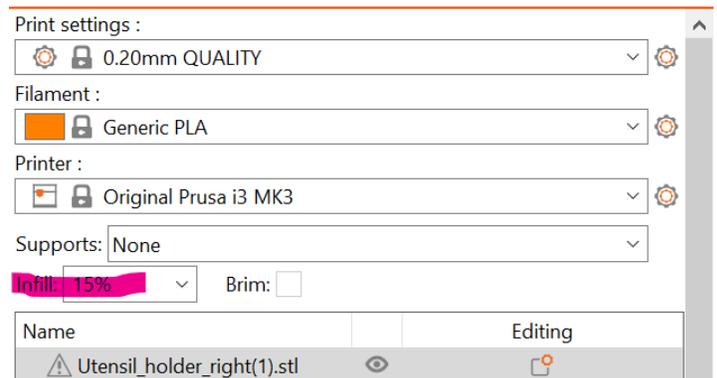
Inside the slicer program, there are a few settings that we will need to select. The first is our printer. Because I have a Prusa mini, I will select it from the list of options. This tells the slicer the dimensions of the printer we will be using. If you were to use a different printer, you would select it here



Next, we can select our material. For this print, I have a roll of PLA filament hooked up to the printer, so I will choose PLA. Different materials have different temperature requirements and selecting the correct one tells the printer how hot it needs its components to be to use the material.

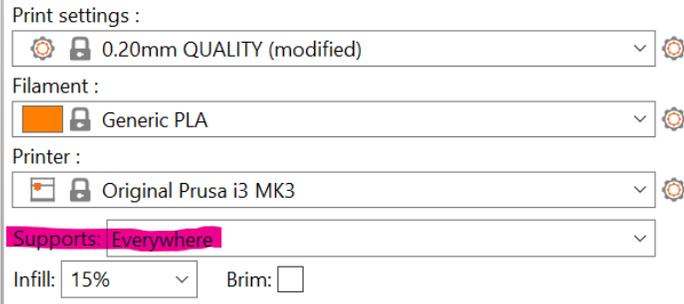


Next, we can select our infill percentage. Infill is the percentage of the print that will be comprised of plastic. 20% infill will be 20% plastic and 80% air. More infill is typically stronger, although there are diminishing returns around 50% infill.



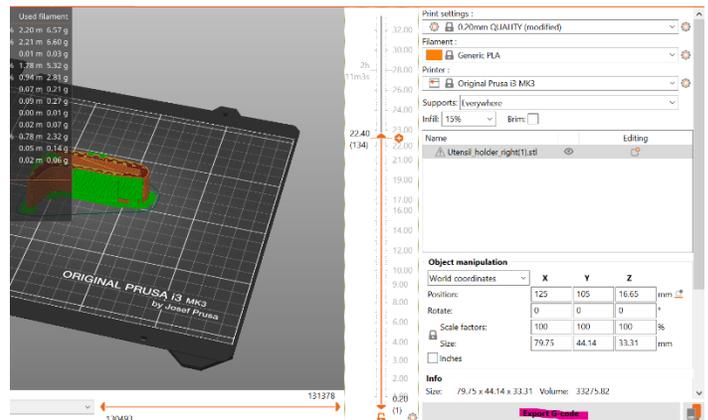
Helpful Hint:

If you have a local makerspace, consider making an appointment to learn about local instruction and resources from pros.



Up next, we will choose if we want supports. If you have areas that are suspended in the air or a design with holes or hollow sections, you should consider using supports to create a buildable surface underneath the hovering area.

Next, we will select the option that says 'slice.' This will transform our print from a 3d object into layers, which the 3d printer will be able to understand. This will be a G-code file. Save it to a storage device that is compatible with your printer, and we are ready to start our print!



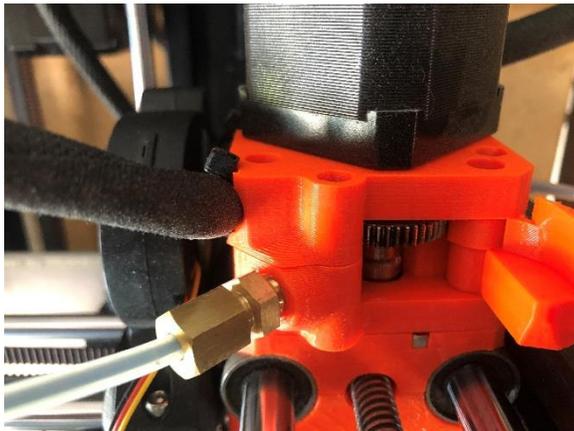
After inserting the storage device and turning your printer on, you can navigate the menu system to the file that you would like to print and start the print. Up next, I will show you the parts of a 3d printer and how they are used.

Helpful Hint:

Test out changing the orientation of your print to reduce the number or supports needed and save material

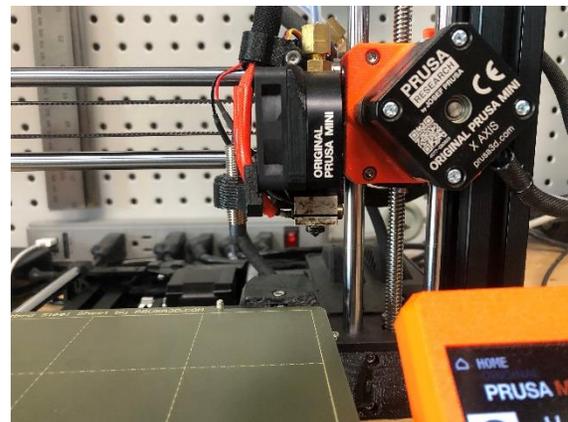


This roll of PLA is our raw material that will be melted down and shaped into the final product. There are many different materials that can be used in FDM printers for different purposes, but PLA is by far one of the most common, affordable, and available.



The extruder is the first step for our plastic. A set of gears move the plastic from the roll into the heated components of the printer at a pace that is set by the slicer G-code file that we made.

The plastic is then fed into a hot end, a metal tunnel that heats up the plastic until it melts. The pressure of the extruder adding more plastic forces the melted plastic out of a small metal nozzle.

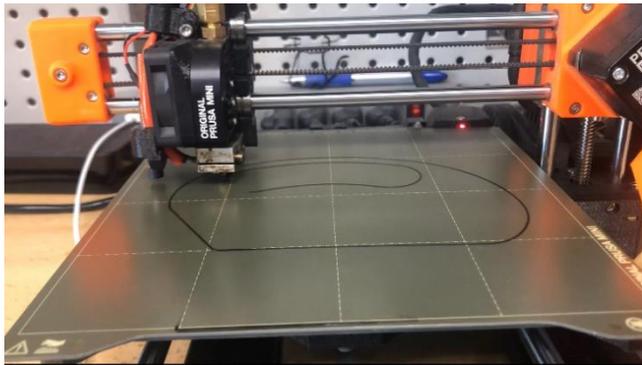


Helpful Hint:

If you run into any issues, there are many guides online that can help you to troubleshoot any issues

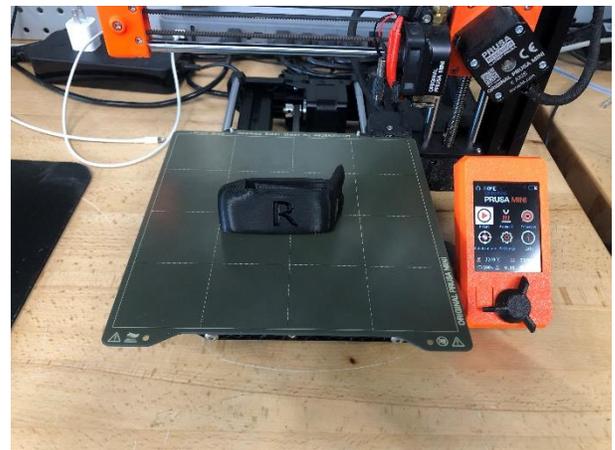


There are 3 motors that control the movement of the printer. The Z axis motor moves the hot end up and down, the X axis moves the hot end left and right, and the Y axis moves the heated bed forward and backward. Different printers may have these in different locations.



The motors work in unison to place the hot plastic onto the printer's heated bed. The heat of the bed allows the plastic to stay stuck in a fixed location on the bed while the bed and extruder move to stack lines of filament to build the print.

After some time has passed, the layers will continue stacking until the print is complete. It is now safe to remove the print from the printer bed. You may need to remove supports using a pair of snips. PLA can be sanded using standard sandpaper to achieve a smoother finish if desired.



Helpful Hint:

Check out OT3d.org for more instructional guides, videos and resources!