Beginners' 3D Printing Protocol

Introduction

This is a beginner's protocol for how to 3D print. The intended audience is UW students, but the protocol can easily be modified to fit other situations. This protocol is not intended as a troubleshooting guide or a step-by-step process for an advanced 3D printing enthusiast- rather, it is meant to provide initial steps to start printing.

Things you need

1. Your model in a .STL file
    a. There are other file types to store modelling data- however, .STL types are the best for 3D printing as it has a small file size and is widely compatible with nearly all CAD programs and 3D printers.
        i. For example, .OBJ files (another common 3D modelling file type)will also store texture and color data where an .STL file will not- however, this level of precision/detail isn't as necessary for 3D printing.
2. A 3D printer
    a. There are many different printers on the market- Prusa, Creality, and Bambu are some brands generally favored by enthusiasts. The Mill at UW uses primarily Prusa MK4 printers.
3. A slicer software
    a. PrusaSlicer or Ultimaker Cura are popular ones. These can be downloaded to your personal computer or be used on computers at the Mill.
4. Filament
    a. This is what your printed object will be made of. There are many different kinds of filament on the market, varying in size, color, material type, and price range. While each serves its own purpose, we generally recommend the below stats for a first-time printer
        i. Type: PLA
        ii. Size: This is the diameter- check your printer for compatibility
            1. The Mill printers take 1.75mm
        iii. Price: A kilogram of this filament should range between 15 to 25 dollars.
            1. Amazon has many good listings for filament. You can also purchase directly from suppliers such as Hatchbox, Tangled, or Overture, just to name a few.

Summary

To print a model, you will need to upload your .STL file to a slicer software, which will take your model and the various settings you set and export a gcode file. Take this gcode file (on a USB) and plug it into the printer. After preparing the printer (which includes selecting some settings, heating up the filament, and feeding it through), select the file to print.

Protocol

1. After creating your 3D model (in whatever software), export it as a .STL file
2. Upload the .STL file to the slicer software
3. Once uploaded, there will be some options for your print. Go with the defaults and make sure it is set to PLA.
    a. Set infill to 15% or 20%. Infill is the internal structure of the print and affects the weight, density, and print time of the object. 15% or 20% is a standard- pretty durable while also being light and not wasting too much filament
    b. Depending on the model, you may need to add supports (namely if it is complex such as not having any wide flat faces or very tall with overhanging parts). Supports are printed along with the actual object to make sure your print doesn't fall over or move around while printing is in place, which can ruin your project. They are meant for you to snap them off after. The slicer software will identify the places that need supports and put them in for you. Review it before proceeding.
4. Make sure to check the following below before proceeding:
    a. Size of print
    b. Layout/orientation of print (if you are printing multiple parts or parts with complex shapes- are they on the surface in a way that makes it easy to print?)
    c. Infill
    d. Print time- most small (palm sized) objects will take at least one hour, but if the print time feels excessively long, recheck your settings
5. Export your print as a gcode file and save it to a USB flash drive
6. Take your USB over to the printer and insert it
7. You will now have to select some settings based on the filament and other preferences. Remember what kind of filament you are using!
8. Now preheat the printer using the menu options

9. While it is preheating, hang up your filament and feed it through any guides. Use the menu options to feed it into the extruder (which will then spit it out through the hot end)
10. The printer will spit out some filament before your print as a test
11. Once preheated, select your file in the menu to print
12. Watch the first layer print. Afterwards, you're good to leave your print to finish! The finish time will be displayed on the screen.

Citations

Adobe. "STL Files Explained | Learn about the STL File Format | Adobe." *Www.adobe.com*, www.adobe.com/creativecloud/file-types/image/vector/stl-file.html. Accessed 5 Mar. 2025.

hfsperks. "How to Use the Prusa MK4 3D Printers at the MILL." *YouTube*, 24 July 2024, www.youtube.com/watch?v=RS2ZAjKSuYc. Accessed 6 Mar. 2025.

McClements, Dean, and Aaron Lichtig. "STL vs. OBJ: What Are the Differences?" *Xometry.com*, Xometry, 24 May 2024, www.xometry.com/resources/3d-printing/stl-vs-obj/.

Seah, Sean. "8059Blank." *8059Blank.*, 31 May 2020, 8059blank.github.io/individual/3D-printers-102/. Accessed 6 Mar. 2025.

Pictures

Here is a printer for some reference