**COFC 3D Printers**

There are three 3D printers available for students to use. Two are Ender 3 V2 filament printers and 1 resin printer. All three printers are available via a webgui anywhere on campus. The two filament printers have raspberry Pi’s with OctoPrint installed to remotely print, monitor, and view your 3D image. The resin printer uses FormLabs software.

**Current Features Setup**

Remote Print – Print/Monitor/Adjust all remotely

GCode Viewer – See each layer of slice

Emergency Stop – will stop the print instantly

Auto Heat Off – Will turn off heating when print is complete

Pre-set Temps (Make sure GCode has no temps set) – Sets temperatures appropriate for Filament

Live Video Feed – Watch the print live anywhere on campus

Timelapse Video – Rewatch the print

Auto Bed Leveling – Automatically adjusts bed level

**How to Connect**

<http://153.9.220.104> – Steve Jobs (Right Printer)

<http://153.9.220.102> – Bill Gates (Left Printer)

\*\*SPEAK TO SYSTEM ADMIN FOR ACCESS\*\* - Resin Printer

**Login Info**

Login: student

Password: Student1234

**How to Use 3D Printer**

3D printers have many features depending on the project. We have setup our 3D printers with parameters to prevent students from adjusting settings that would be outside the normal use. We also setup features to put safety in place for any user. 3D Printing is about a lot of trial and error depending on the print image.

Before printing remove or add the filament color desired for the print. The painters tape helps the print stick to the bed, it’s recommended to replace this before every print. To use the 3D printers you will need to create a gcode file. Its recommended to use ultimaker Cura to do this. Upload a gcode file in the files window of the webgui. Large code files can take a few minutes. To print the code file, click on the printer icon in the file list. If you want to reprint, use the print button in the state window. The printer icon button does work for reprints. The cancel button in the state window can be used to abort a print. When the print is done please clear the print bed for the next user.

<https://docs.octoprint.org/en/master/> - Master Guide For Using OctoPrint

**Ultimaker Cura Setup**

Download - <https://ultimaker.com/software/ultimaker-cura>

The easiest way to convert a 3d print file you found online to a gcode file is using Ultimaker Cura. Follow these steps to get Ultimaker setup so you can get to making 3d prints:

1. Download Ultimaker Cura
2. When you first open Ultimaker select the printer setup at the top left (to the right of the file icon)
3. Select add printer
4. Select add a non-networked printer
5. Scroll until you find Creaility3D and select the dropdown menu
6. Select Creality Ender-3 and then click add (if a menu comes up with default settings just go with the defaults and then select next)
7. Select print settings at the top right
8. Navigate to material and change the build plate temperature to 75
9. Navigate to material and change the printing temperature to 200 to 220 (depending on filament)

You will only have to do the steps above once. Once you have done all of that you can then add any 3D printing file to Ultimaker then change any other settings you may want (Infill, support, adhesion, etc.) and then hit file at the top left and select save as type gcode and then navigate to the 3D printer webgui and select the appropriate file.

**3D Printer Care**

Any filament not currently being used should be stored in a bag in the drawer

Dry packets keep the filament lasting longer while stored

Clean the print bed after each use