# 3D printing guide

Nolop Makerspace
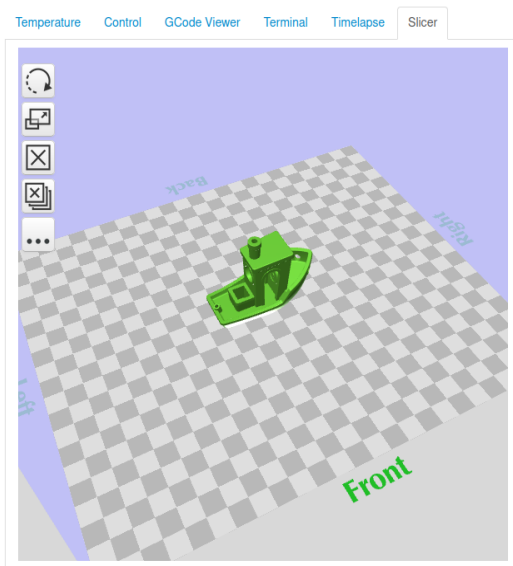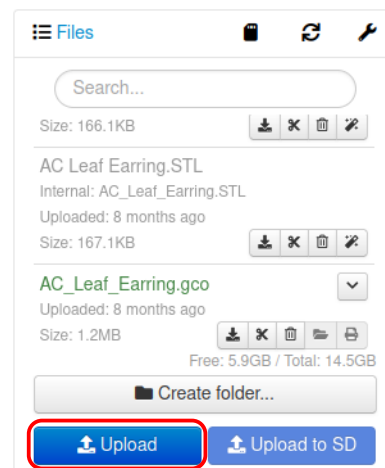Tufts University

## Step 1: pick a printer

1. Pick a 3D printer that nobody else is using and make sure the bed is clear.
2. Check that there is filament loaded into the extruder.

## Step 2: connect to the web interface

1. On your laptop, open **https://px.nolop.org**
where the **x** is the number of the printer you want to use.
2. Log in with **username: nolop**
              **password: 3dprint**

## Step 3: upload your 3D model

In the Files pane of the sidebar, click Upload. You can only upload .STL files. You can make an STL file with CAD software like Onshape or Solidworks.
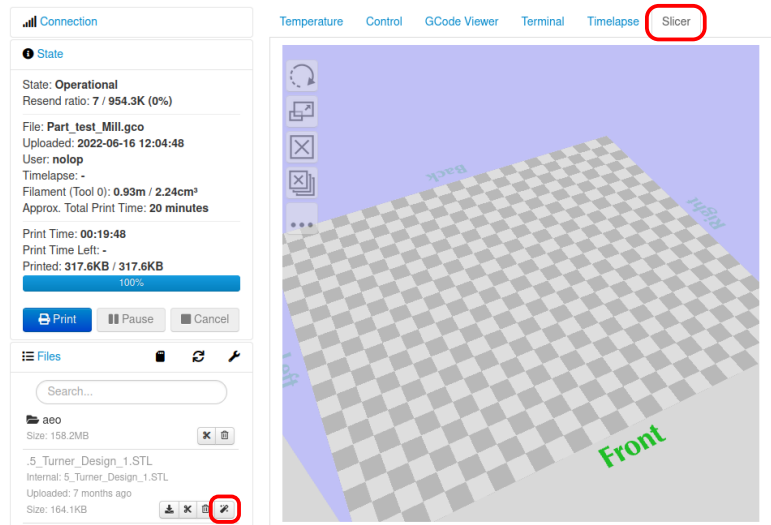
## Step 4: position your model

Use the 5 buttons in the slicer window to rotate, scale, delete, or duplicate your model. You can print two different models at once, but if one of them fails, both are ruined.

If possible, shrink your print down. Print time and filament usage scale roughly with the cube of the size of the print, so a print twice the size takes around 8 times longer.
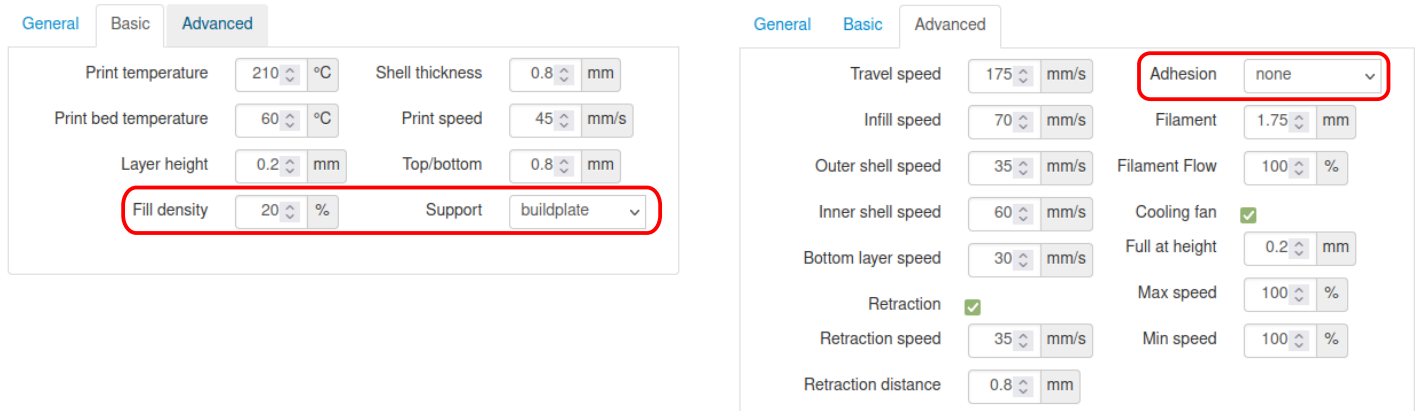
# No model showing?

1. Click on the Slicer tab along the top.
2. Select your uploaded STL file in the Files pane.
3. Click the small Slice button next to the filename.
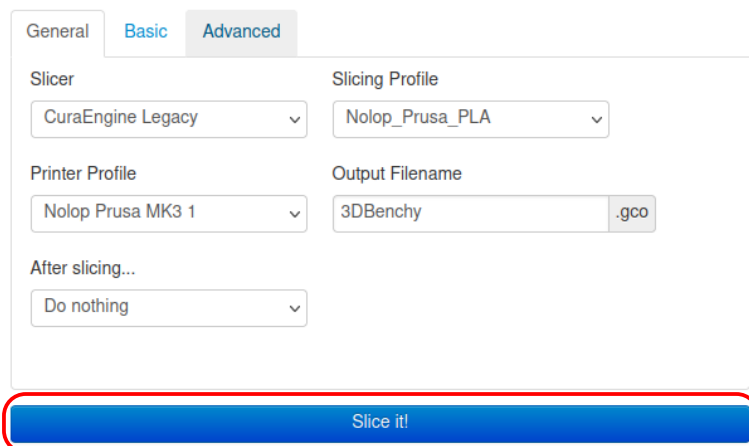Your model should then appear on the platform.

# Step 5: tweak slicer settings

You can print with the default settings, but you might want to tweak the fill density, support material, or adhesion method, boxed in red. Please don't change any temperature settings.
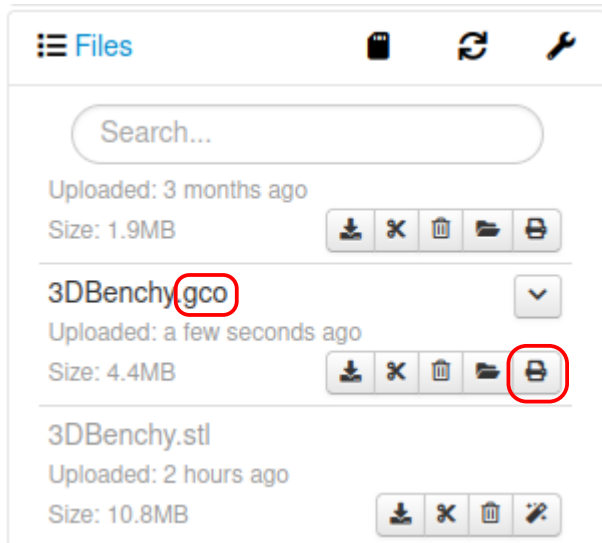
# Step 6: slice it!

Press the Slice it! button. This will generate a .gco file, which contains G-code, which are instructions that tell the printer how to move. Slicing might take a minute or two.

## Step 7: start the print

Find your .gco file in the Files pane and click the Load and Print button.



## Step 8: monitor your print

Check that your print is starting properly. The heated printbed takes 2-3 minutes to warm up. You should watch the first couple of layers print to make sure that the filament sticks to the bed, but after that, you can leave it printing unattended.

## FAQ

**What is the hairspray for?**

Every 10 prints or so, the bed needs a quick spray to make it sticky. You can spray a light coat if your print has trouble sticking to the bed.

**Can I print with my own filament?**

Sure, as long you're willing to unclog the extruder if it clogs up. Most PLA tends to print fine.

**How do I change the filament to another color?**

Use the load and unload filament functions with the LCD display knob. A Nolop staff member can show you; it's not that hard.