



3D printing guide

Nolop Makerspace
Tufts University

Revision 2.1RC1

Step 1: pick a printer

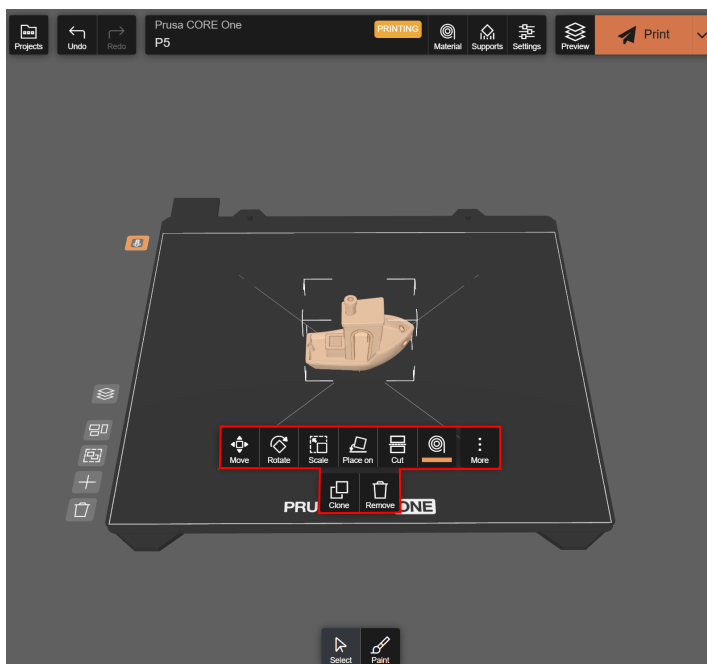
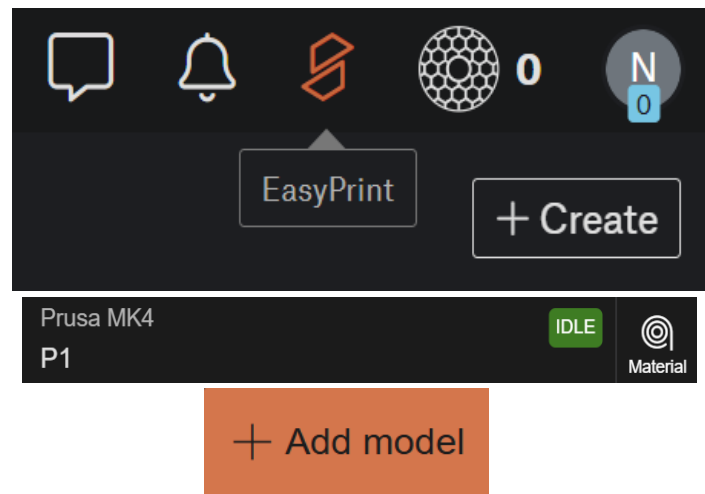
1. Pick a 3D printer that nobody else is using and make sure the bed is clear.
2. Check that there is filament loaded into the extruder.

Step 2: slice and connect to printer (if you've already sliced skip to step 7)

1. On your laptop, open printables.com
2. Log in with:
email: brandon.stafford@tufts.edu
password: nolop3dprint

Step 3: upload your 3D model

1. Click EasyPrint to open the slicer
2. Select your printer at the top
3. Click add model in the bottom right and upload your .stl file.



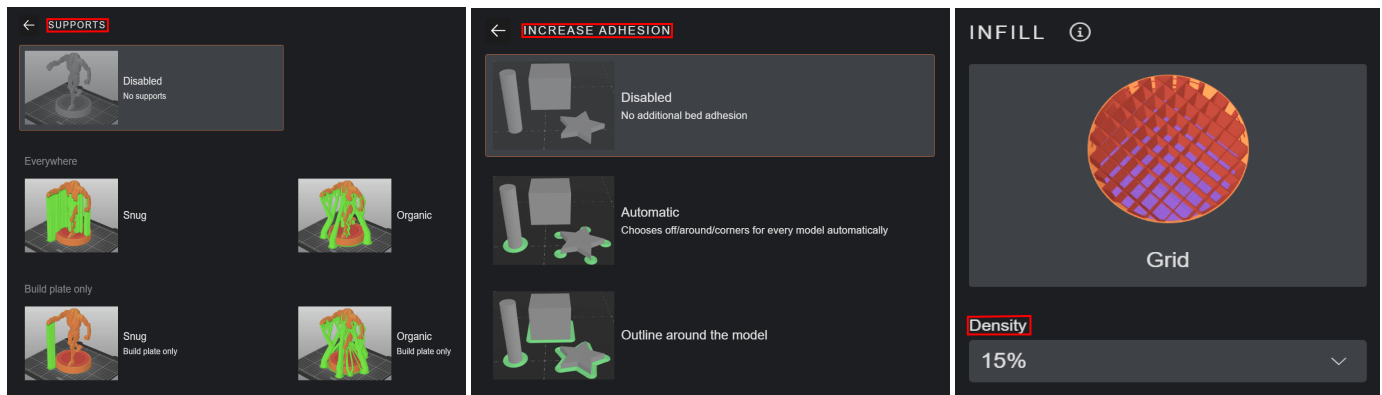
Step 4: position your model

Use the buttons, boxed in red, 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.

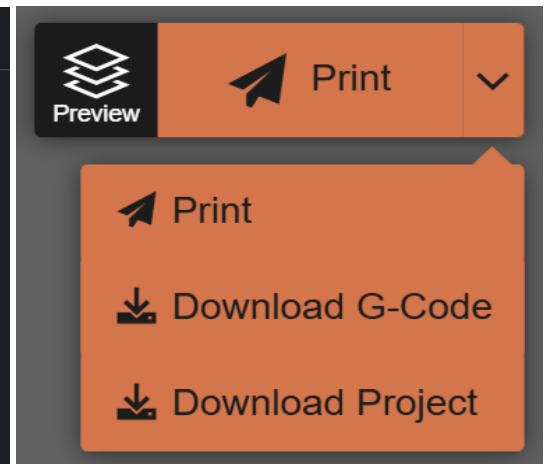
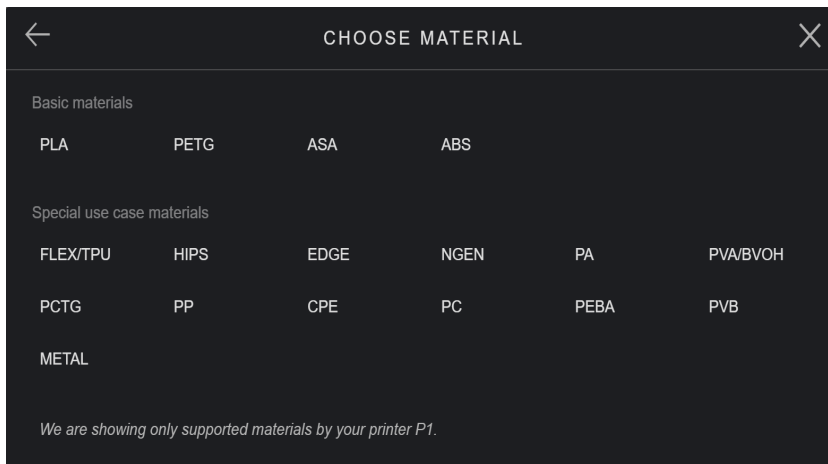
Step 5: tweak slicer settings

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



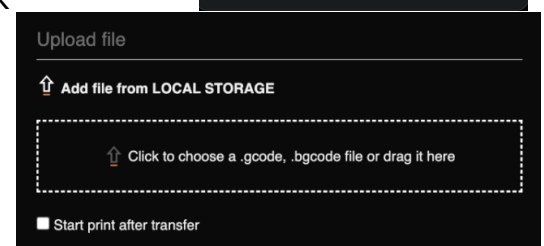
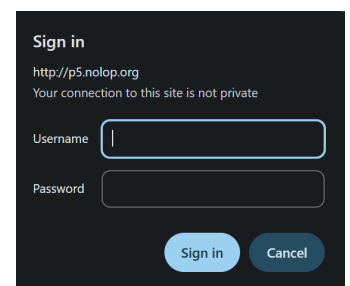
Step 6: Print It!

Press the **Print** button. It may first ask you to select a material if one is not chosen, please select Generic PLA. This will generate a .bgcode file, which contains G-code, which are instructions that tell the printer how to move. This file is sent directly to the chosen printer. Please make sure the printer is open. If you do not want to print and only want the G-code, press the arrow next to the **Print** button and select the **Download G-Code** option.



Step 7: Already Sliced?

1. If you already have a sliced model go to: **p#.nolop.org** (replace # with the number of the printer you chose)
2. Login with the username and pw of your printer (on printer go to: SETTINGS > NETWORK > PRUSALINK)
3. Upload your sliced model and print!



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 Nolo staff member can show you; it's not that hard.