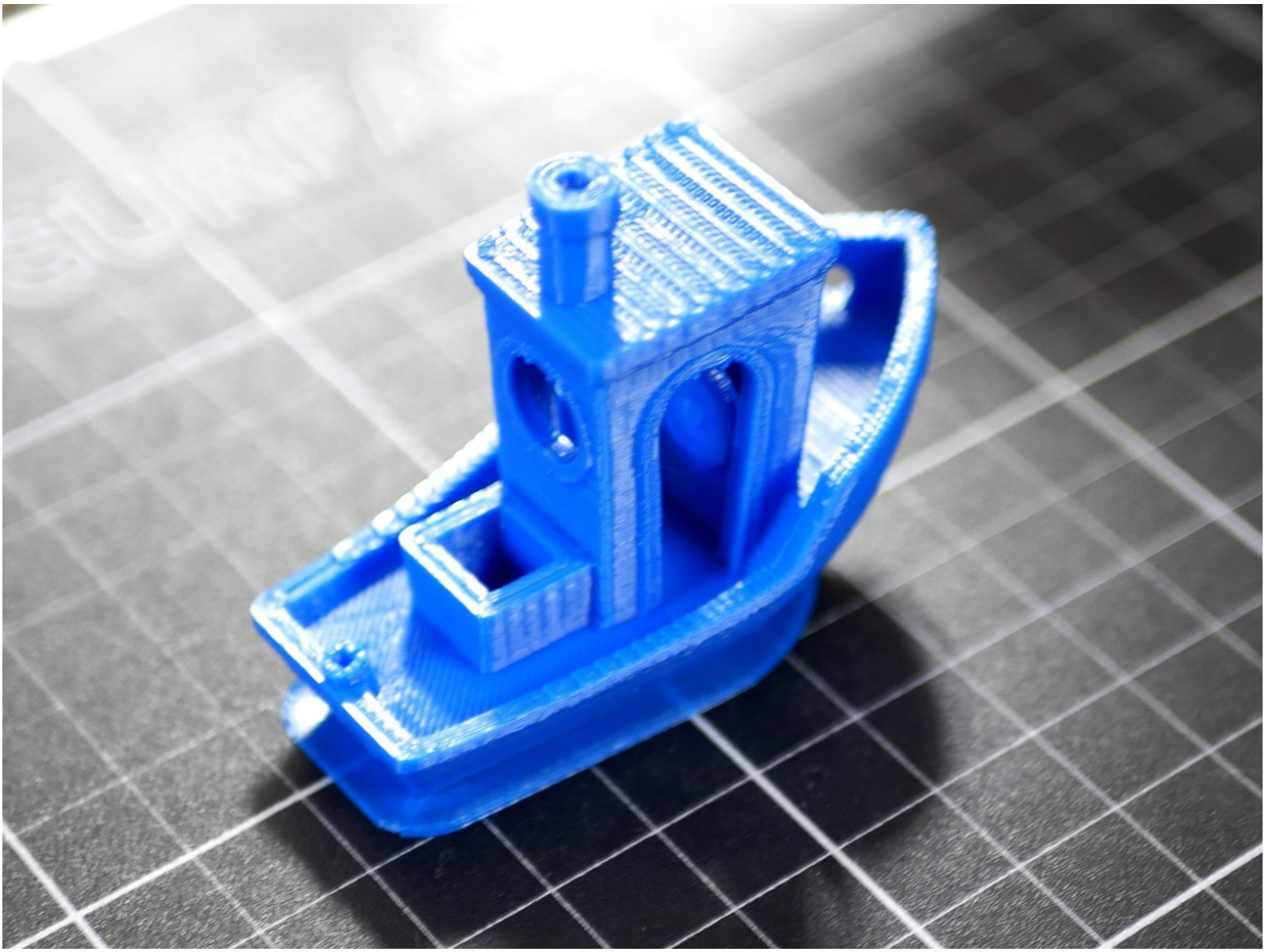
Makertech

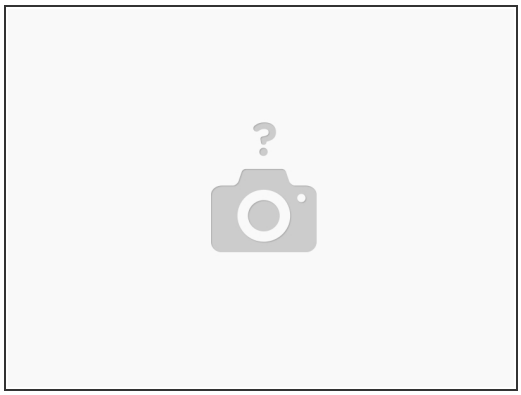
Stage 10: First Print

Printing your first object on the Axis 3d printer.

Written By: Makertech



Step 1 — Cura and Benchy

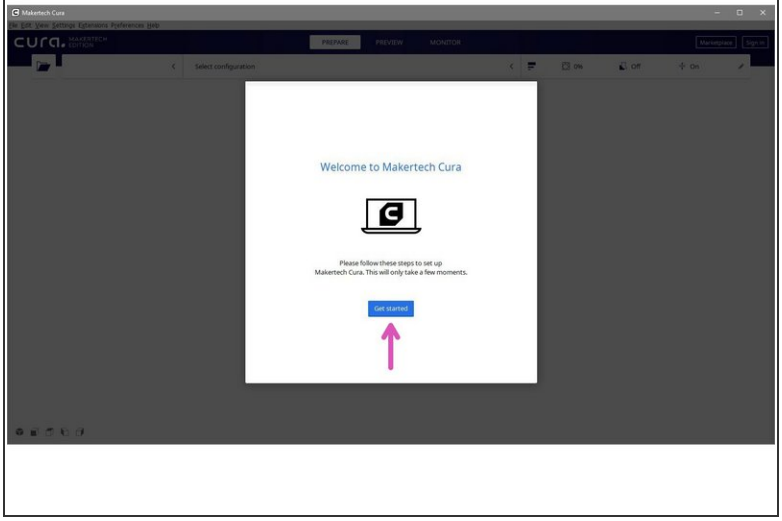


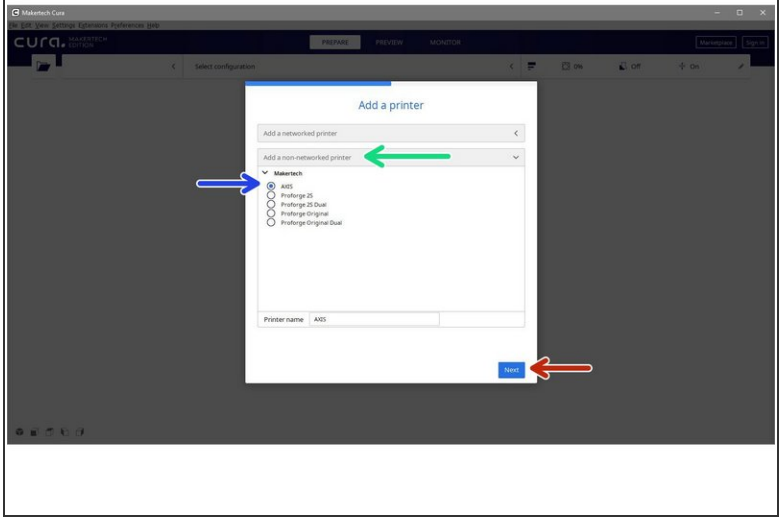
Download the latest version of Cura here.

Run through the setup wizard and install it onto your computer.

Also download the benchy test print, this will be your first print and at the same time will test that the printer can safely print all of its geometries.

Step 2 — Launching Cura

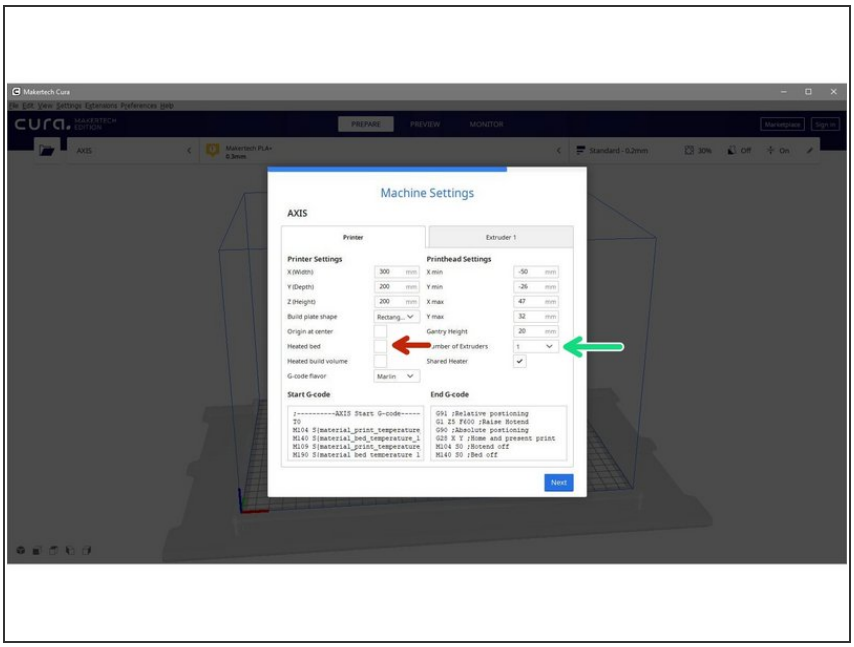




Open up the Cura application.

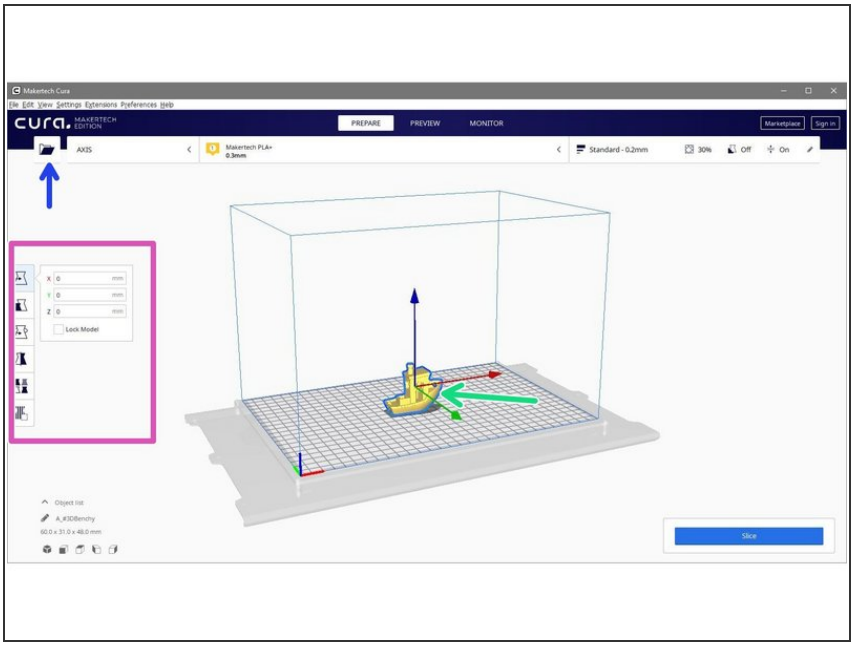
- Run through the startup wizard.
- Click on "add a non-networked printer"
- Select the AXIS 3d printer.
- Then hit next.

Step 3 — Machine Settings



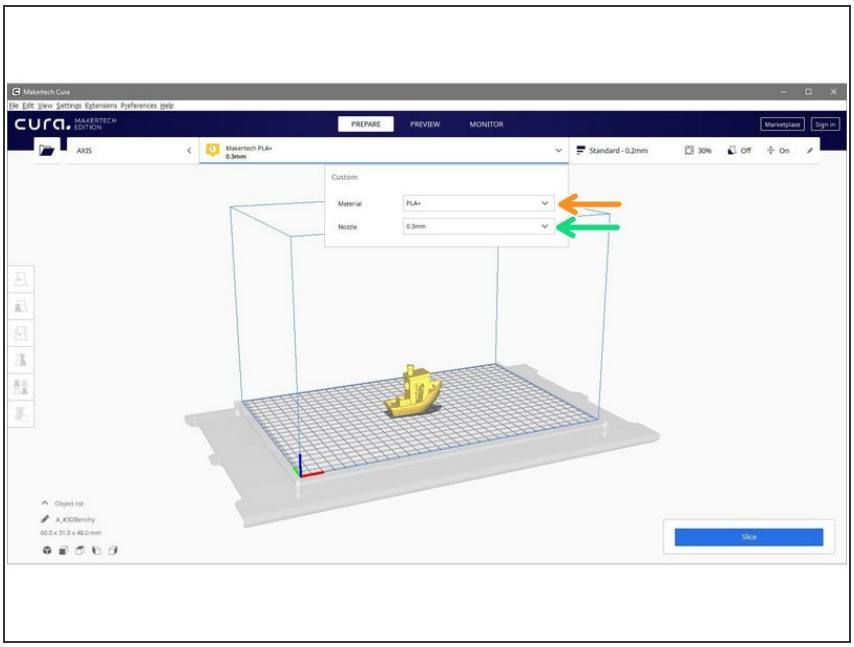
- Here these setting will vary depending on your setup.
- If you have a heated bed installed, check this box.
- If you have dual extruders set-up, set number of extruders to 2.
- Finally hit next and then finish.

Step 4 — Opening Benchy



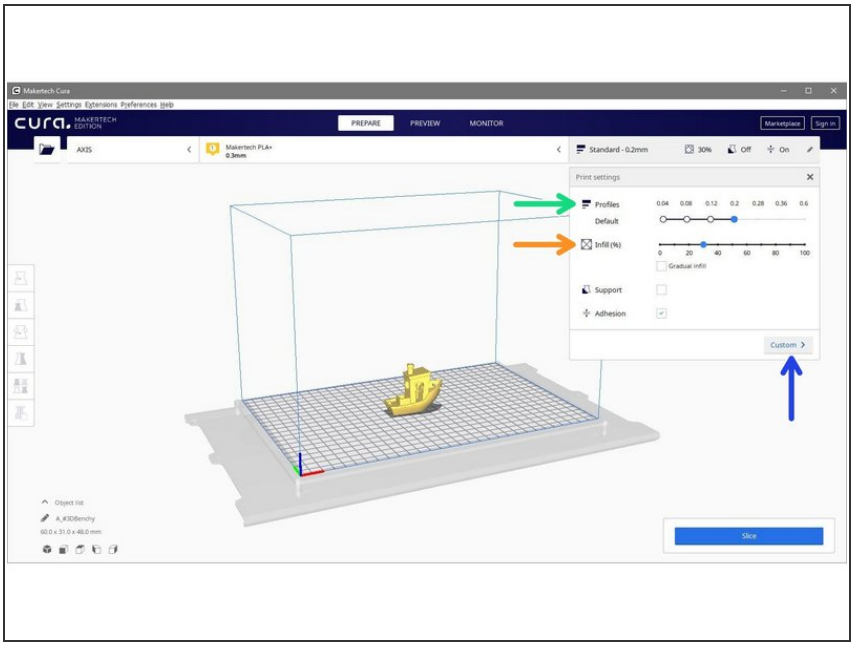
- Click on the open file icon.
- Navigate to and open the Benchy.stl file.
- You can also use the tools on the side of the screen to move, scale and rotate your model.

Step 5 — Nozzle and Material



- Select the nozzle size you have installed.
  - ⓘ We recommend to have installed 0.4mm or 0.3mm.
- Select the material.
  - ⓘ we recommend picking from the Makertech list and choosing PLA+ for your first print.

Step 6 — Print Settings



Leave setting as their defaults for the first print.

- Profiles: Print profiles are default settings at different layer heights. The lower the number the more layers, hence more detail, but at the cost of greater print time.

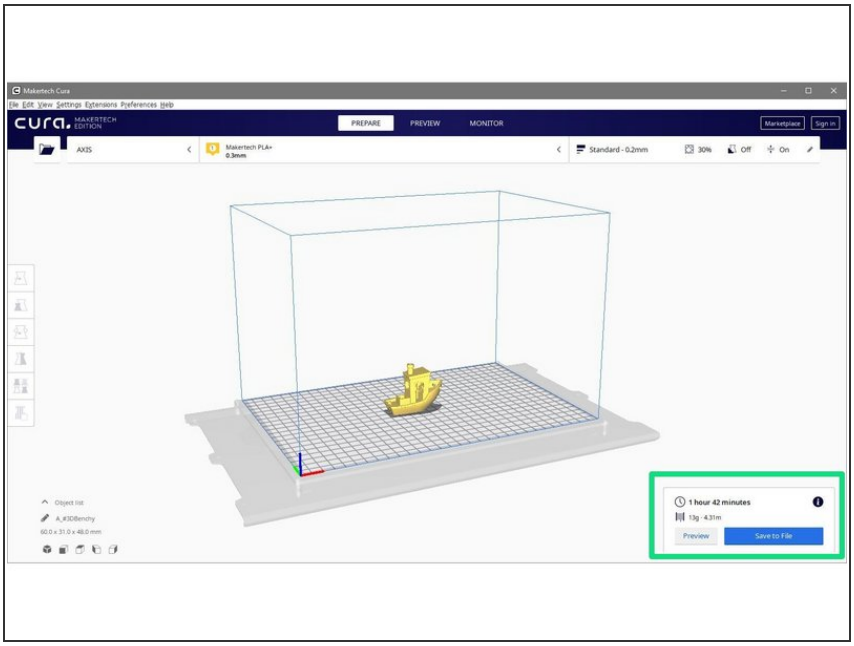
  Note, different nozzle sizes will have different layer height ranges.

- Infill: This is the amount of material the printer will deposit inside the print. Between 20-30% is ideal in most cases.

- Other options can also be tweaked by clicking custom. Hovering over an option will give you a description of its purpose.

You may find you need to adjust certain settings to get your printer working perfectly, we recommend posting in the forum for the best advise on this.

Step 7 — Slicing to SD Card

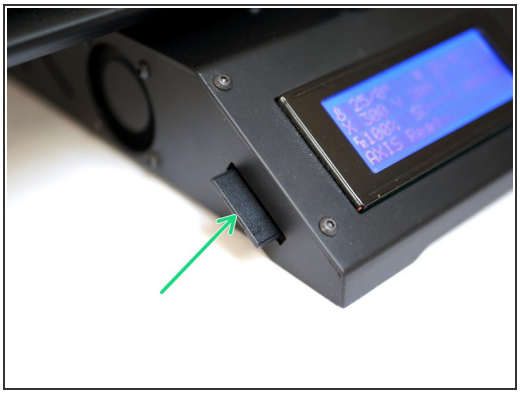


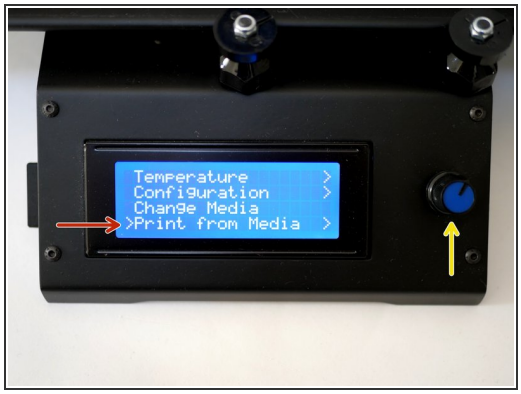
- When happy with the settings, hit slice.

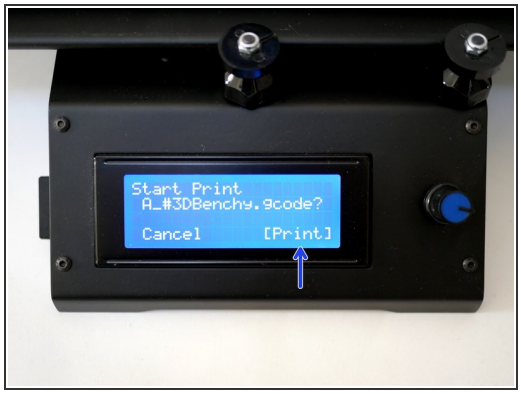
Slicing will convert your 3d model (an .stl file) into and .gcode file which in turn can be read by the printer to print your model.

- If you have the LCD screen installed (from the highly recommended power pack upgrade) you can save your sliced file to an SD card and print directly from your printer.

Step 8 — Printing via SD Card



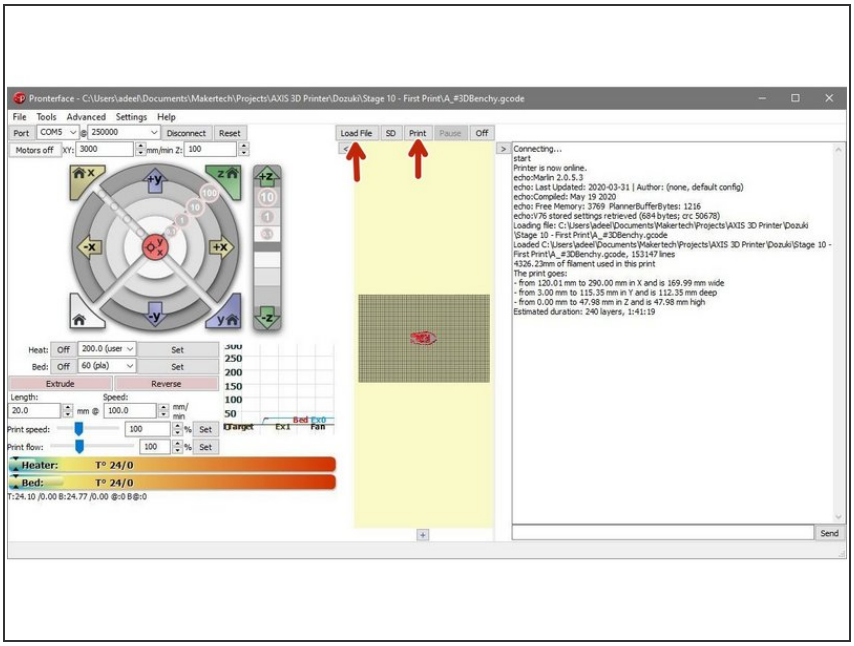




The most efficient way to print is via SD card, its self contained and doesn't rely on a computer connection.

- After saving your .gcode to the SD card, slot it into the side of the Axis like shown.
- Click the turn wheel once and turn it clock wise to scroll to the bottom of the menu.
- Click on "Print from Media"
- Print the .gcode file.

Step 9 — Printing via USB

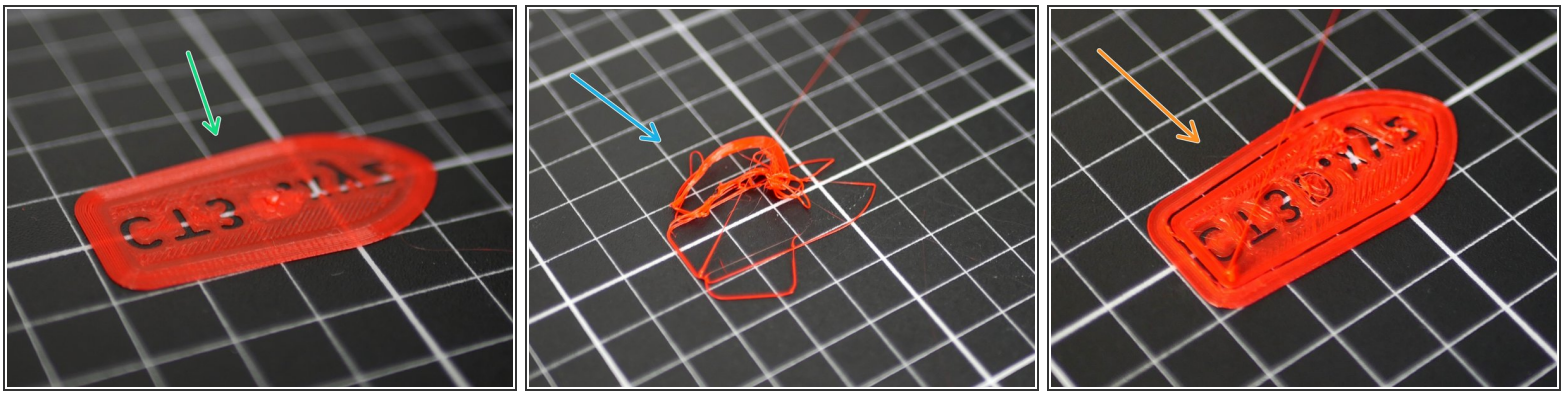


To print via USB you will first need to connect the Axis to your printer via USB.

- Open your saved .gcode in pronterface and hit print once it's loaded.

In case of an error pronterface should receive a message in it's terminal to help diagnose.

Step 10 — First Layer



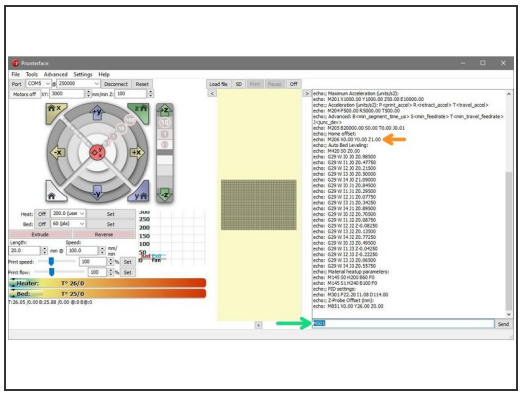
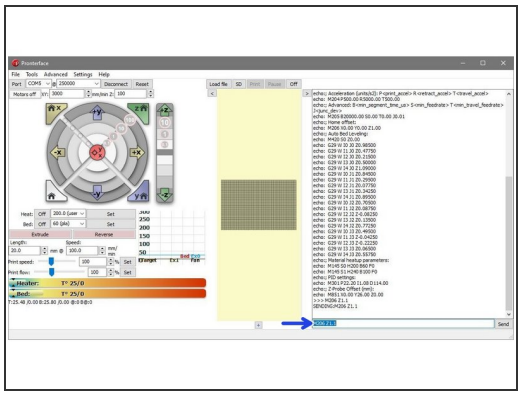
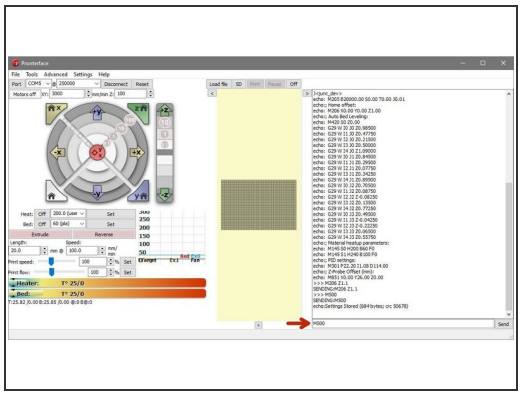
The most critical part of any print is its first layer, this is the contact point between the rest of the print and the print platform.

If printed too high then the model will detach mid-print, too low and will be a pain to remove form the platform.

- What we want is a happy medium like this.
- Too high - the first layer isn't sticking or the model has peeled off.
- Too low - the nozzles is pressing the plastic down too much, or at worst digging into the bed.

If you find your first layer to be too low or high, stop the print and adjust the z-offset, as shown in next step.

Step 11 — Z-Offset

Connect the Axis to Pronterface.

- Send M501 to check what your previous z-offset is set to.
  - In our case this was 1.0.

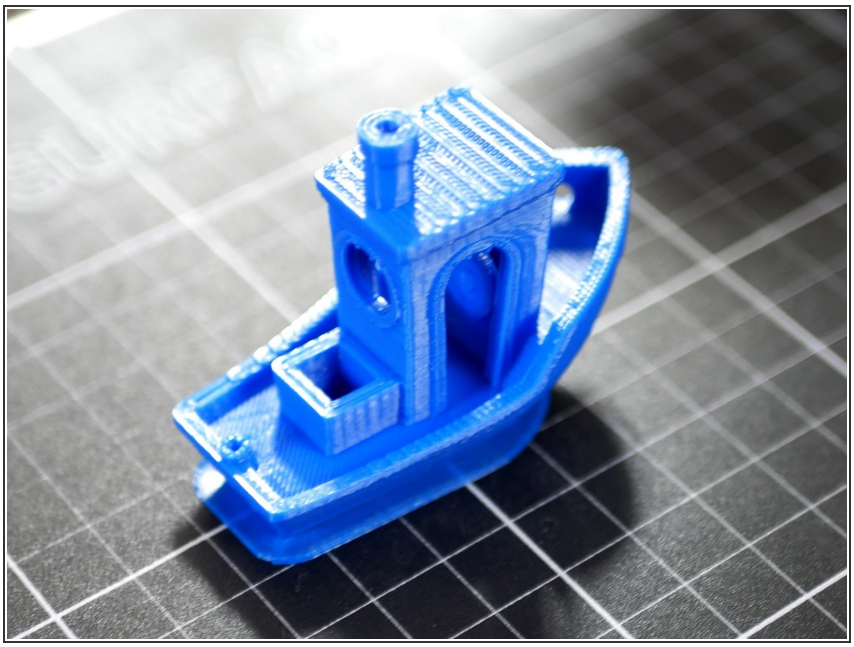
First layer too high, increase the Z-offset by 0.1mm.

First layer too low, decrease Z-offset by 0.1mm.

E.g. The first layer isn't sticking (it is too high) we will increase the Z-offset by 0.1mm to 1.1mm.

- Enter M206 Z1.1 to set the new Z-offset.
- Enter M500 to save to printer memory.

Step 12 — Congratulations!



With the Z-axis offset dialled in you should have a successful first print!

If you're having issues with getting your first print done, post your issue in the forum here and we'll do our best to help!