

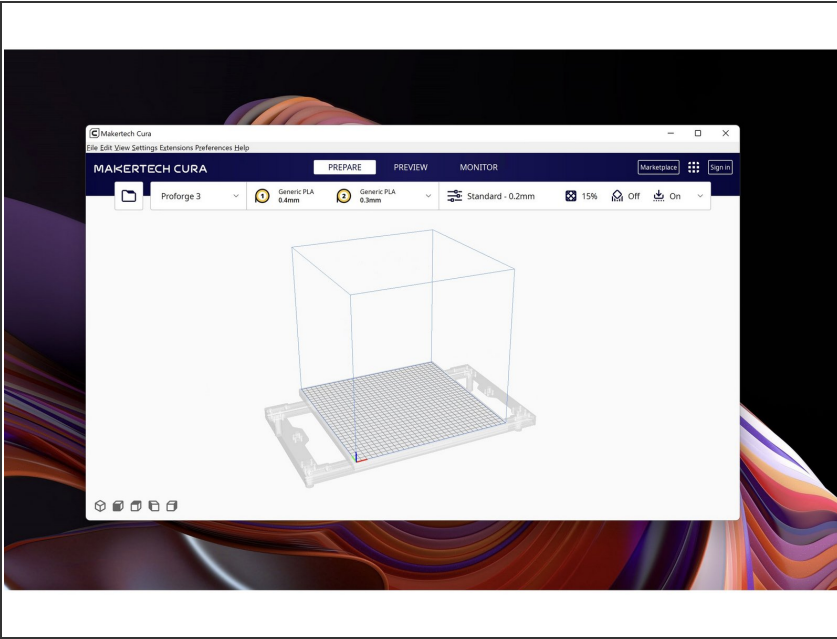
# Makertech

## Stage 09: First Print

Written By: Makertech 3D



## Step 1 — CURA - Windows



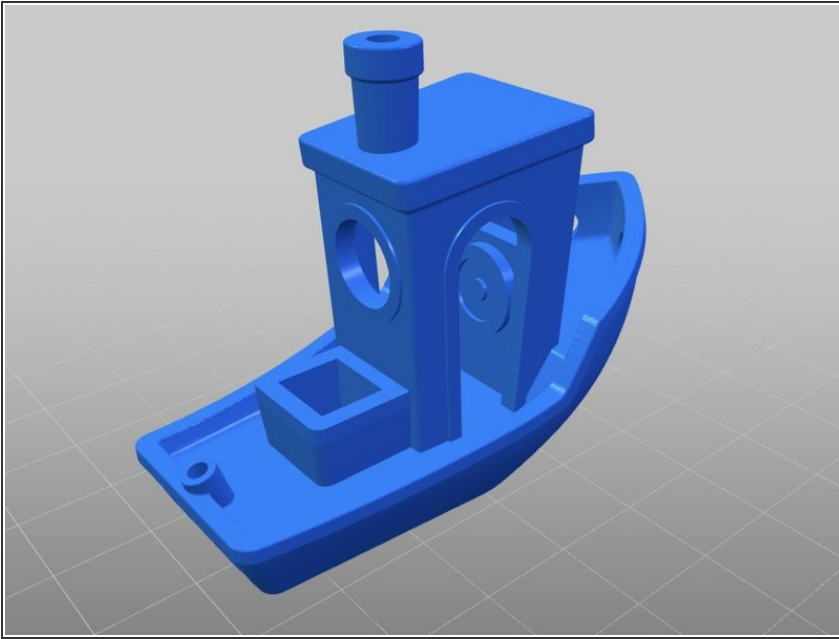
- ❗ Download the new v5.2.1 of Makertech Cura for windows [here](#).
- ❗ Run through the install and setup wizards.
- ⚠ If you have an earlier version of Makertech Cura installed, delete it from your programme folder before installing the new one.

## Step 2 — CURA - MacOS/Linux



- ❗ If you are running Mac OS or Linux the 3.5 profiles will be made available in the next release of the [official Ultimaker Cura](#).
- 📌 Follow the progress of this request [here](#).

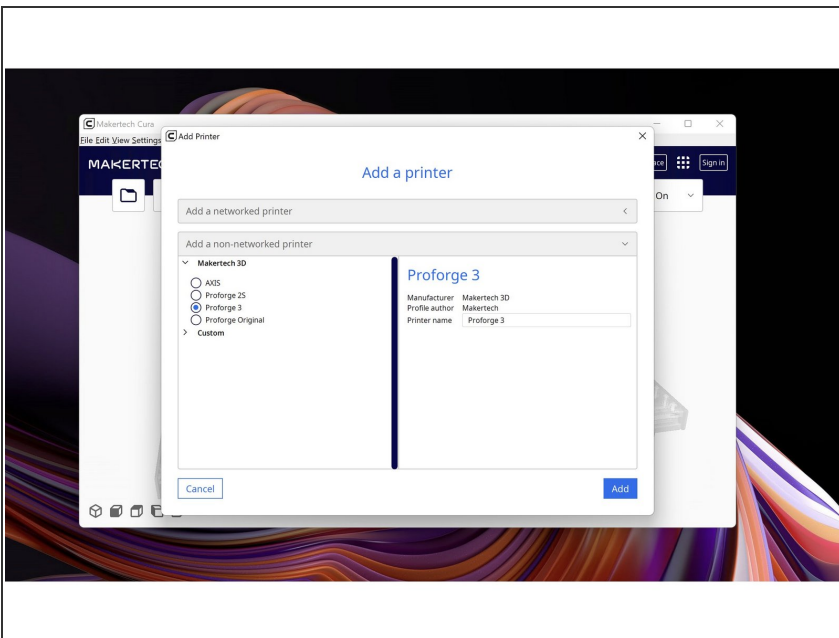
## Step 3 — 3D Benchy




 Our first print will be the 3D Benchy.

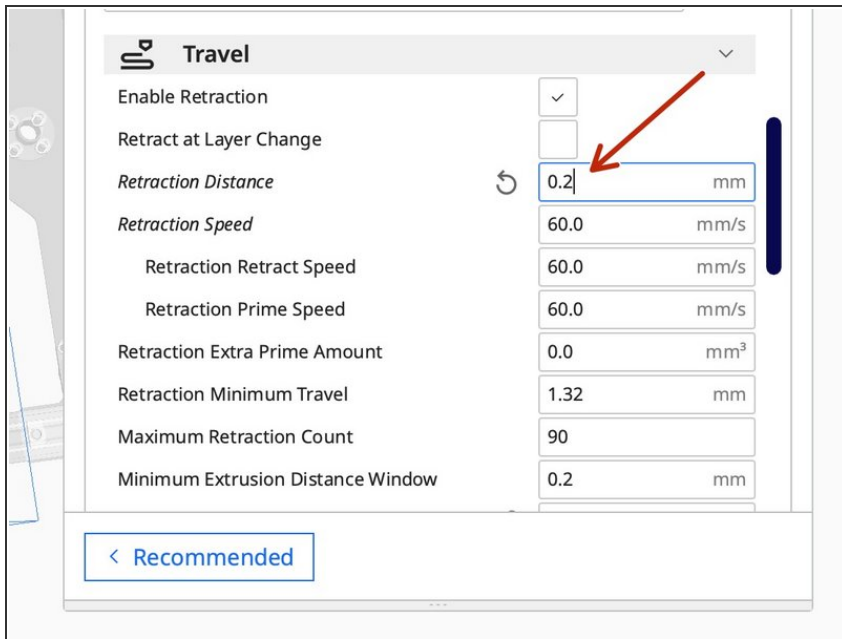
 Click [here](#) to download the STL file.

## Step 4 — CURA Setup



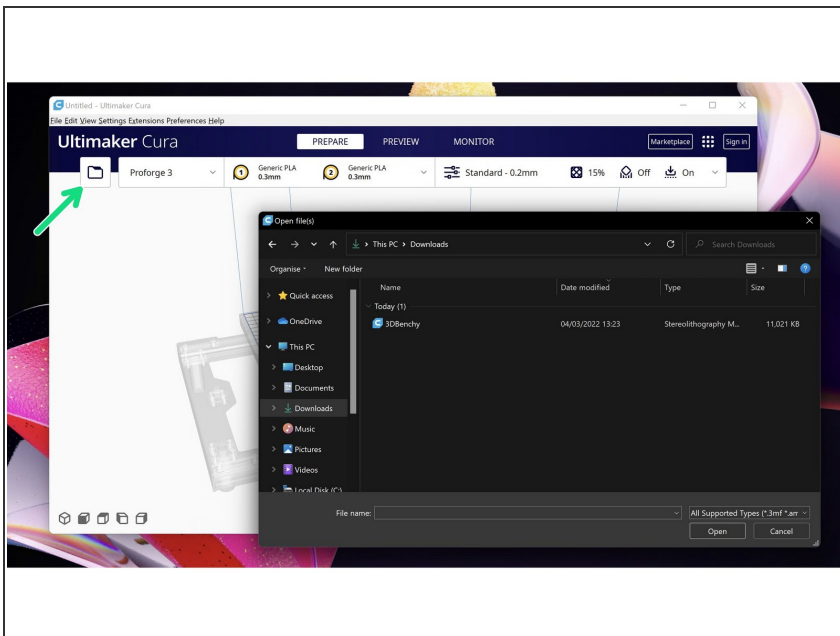
 In Cura add the Proforge 3 3D printer.

## Step 5 — IMPORTANT - Retraction Setting



- Under the *Travel* dropdown adjust the retraction setting to **0.2mm** for the direct drive extruder.

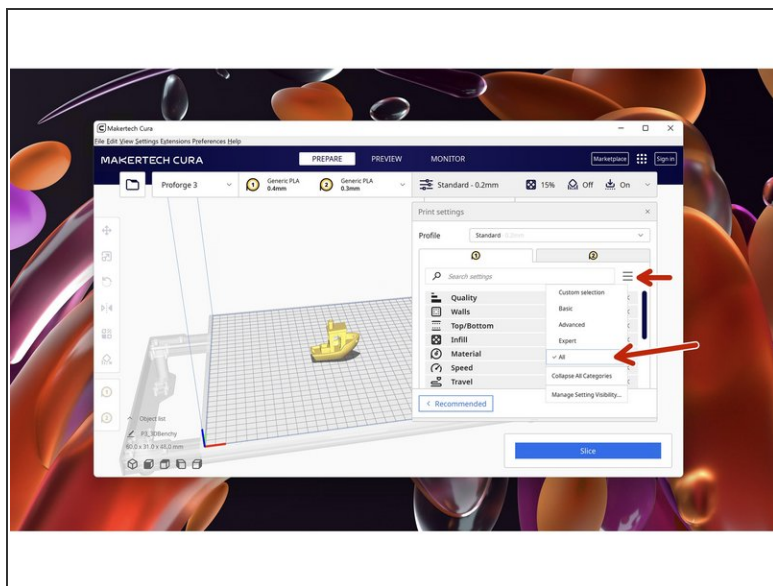
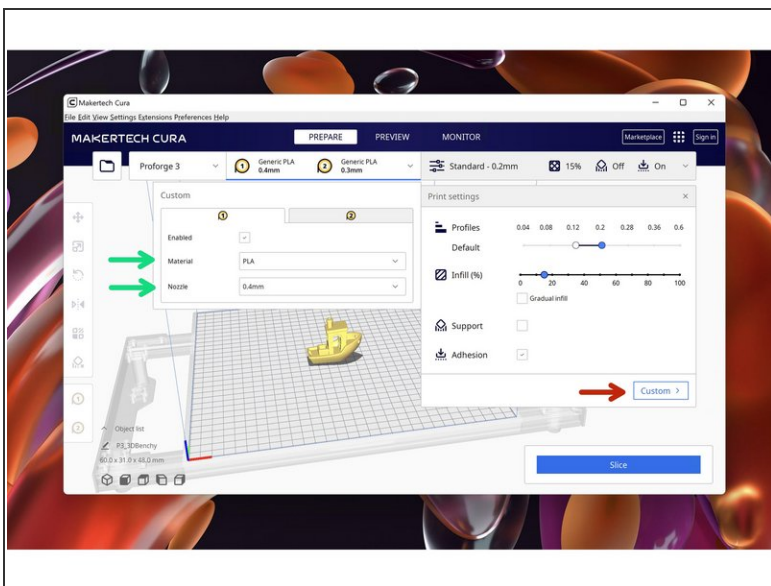
## Step 6 — Loading STL's



- Click here to load the Benchy STL that you downloaded earlier.

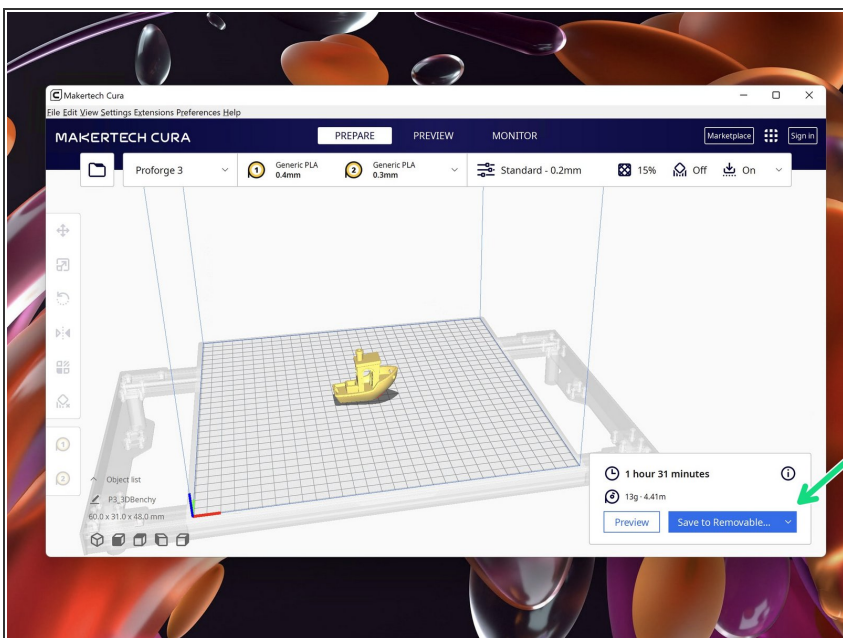


## Step 7 — Slicing



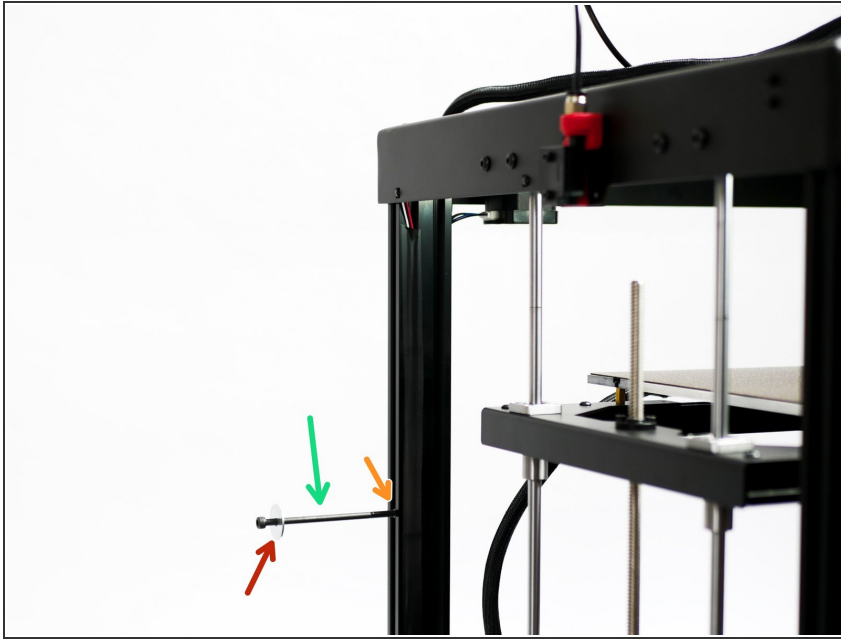
- Set the correct nozzle size and material for your set-up.
- The default profiles give you a good starting point, these will need to be edited to get the best results out of your material, this can be done by going into the custom settings.

## Step 8 — Save to SD



- Once happy, hit slice and save the gcode to your SD card.

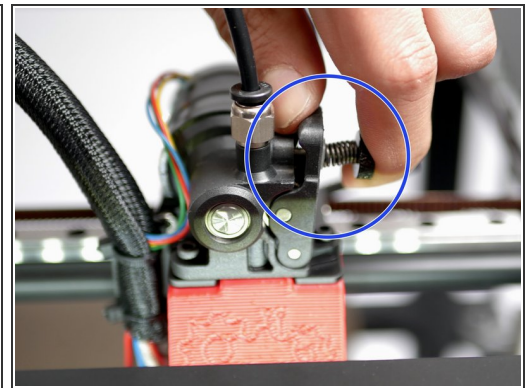
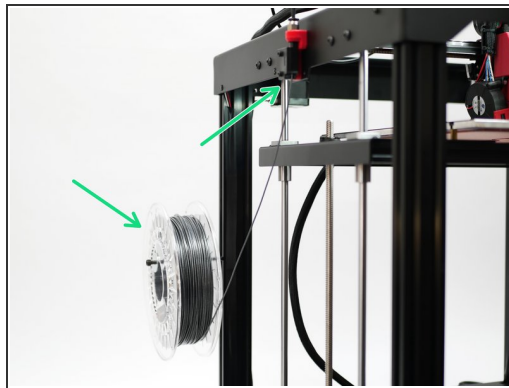
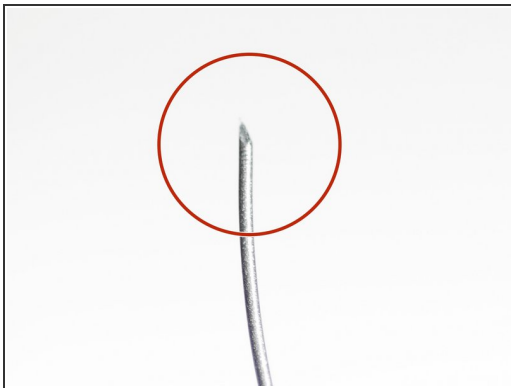
## Step 9 — Spool Holder



❗ Fix the spool holder assembly to the side of the printer:

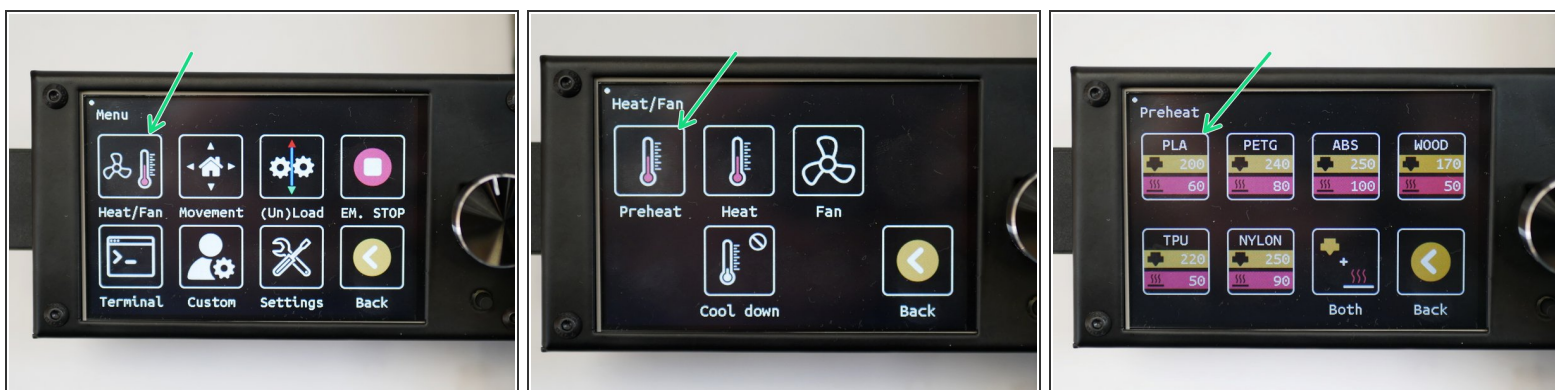
- M5 x 100mm Bolt
- M5 Penny Washer
- M5 T-Nut

## Step 10 — Load Filament



- Use a pair of scissors or side cutters to cut a sharp point to your filament.
- Place the spool onto the spool holder and feed the filament through the filament sensor and up to the extruder.
- Put pressure on the extruder idler arm and continue to feed the filament into the hotend.

## Step 11 — Pre-Heating



- Pre-heat the hotend and heated bed.

- ☑ Remember that the bed is measuring the temperature of the heater, the platform still requires a few minutes to uniformly reach that temperature.

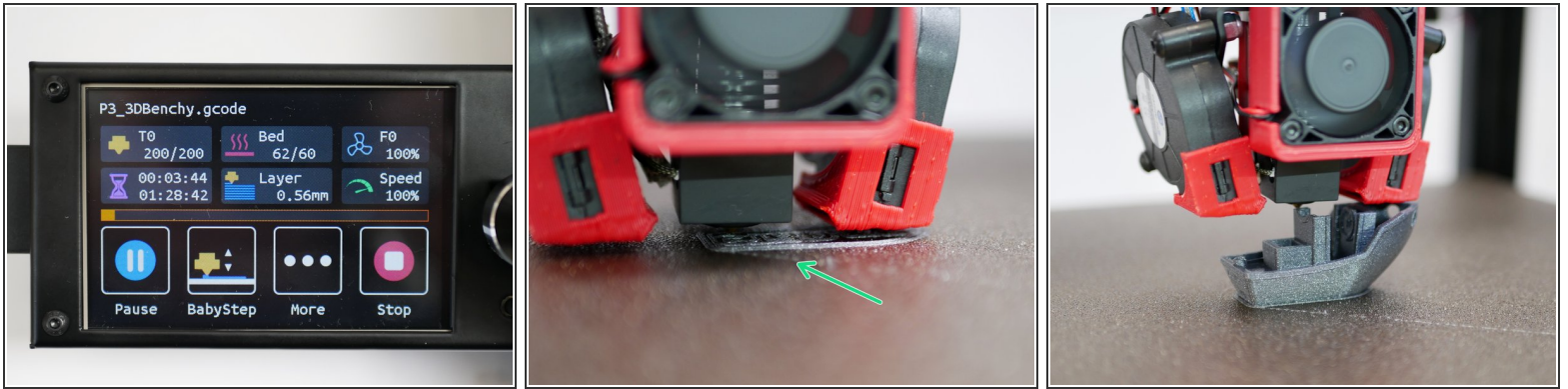
## Step 12 — Printing From SD Card



- From the home screen go to *Print* -> *TFT SD* -> *P3\_3DBenchy.gcode*



## Step 13 — Print Begins



- ① Once the print begins, you can monitor/adjust it from the print menu.
  - Make sure that you have a good first layer, this is key to a good print.
  - ☑ If it's too close (the nozzle is scraping the bed) or too high (the first layer is peeling away) adjust your P offset. See the previous guide for a reminder on how to do that.
- ① Keep monitoring your print.

## Step 14 — Congratulations!



- ① Your Proforge 3 is successfully up and running!