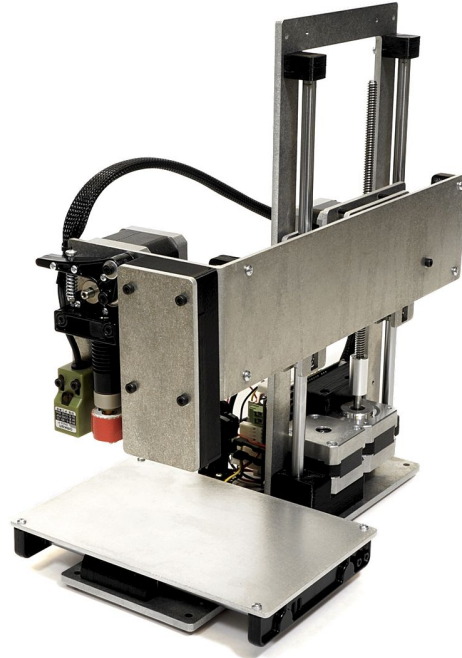


printrbot

Printrbot **Smalls**

model 1704



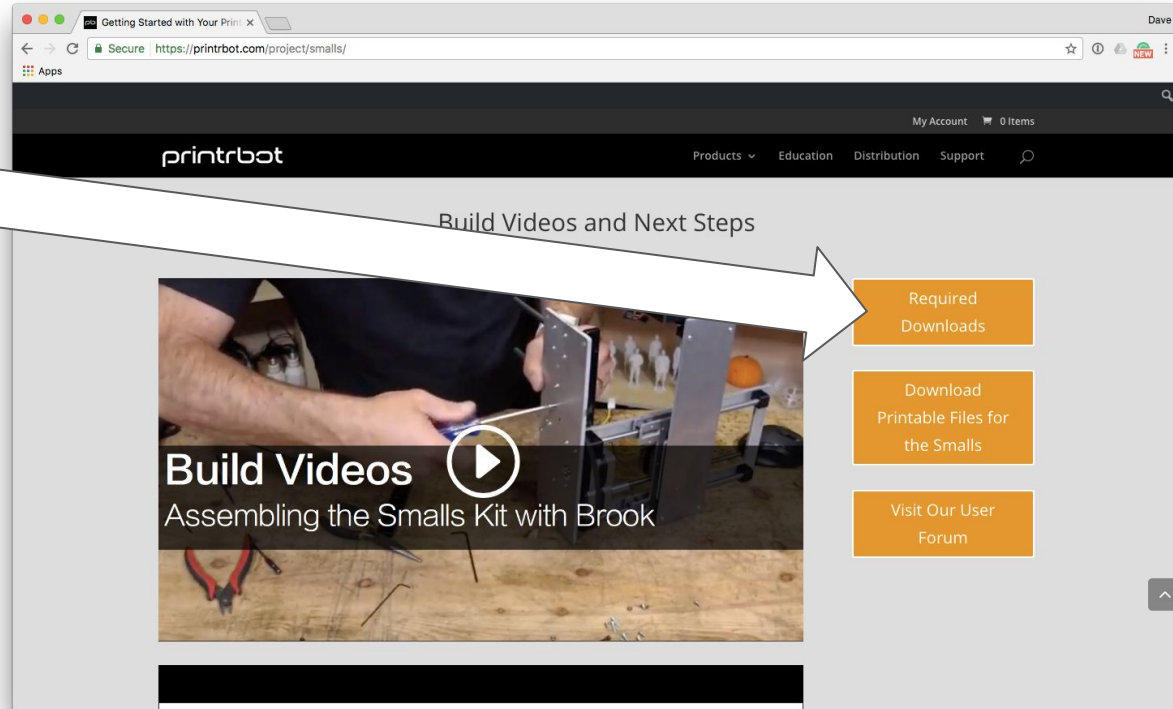
Software, Calibration, & Your First Print

Before You Begin

You will need the following items to complete this guide.

- **REQUIRED DOWNLOADS** folder available at printrbot.com/smalls
 - Cura software
 - Printrbot_testprint.stl
 - Smalls.GettingStarted.ini
 - USB driver for Windows users
- Sample of filament
- Print Bed Tape (optional)

**TIP: Turn OFF any “sleep” settings on your computer. If the screen goes dark while you are printing, the print will cancel.*

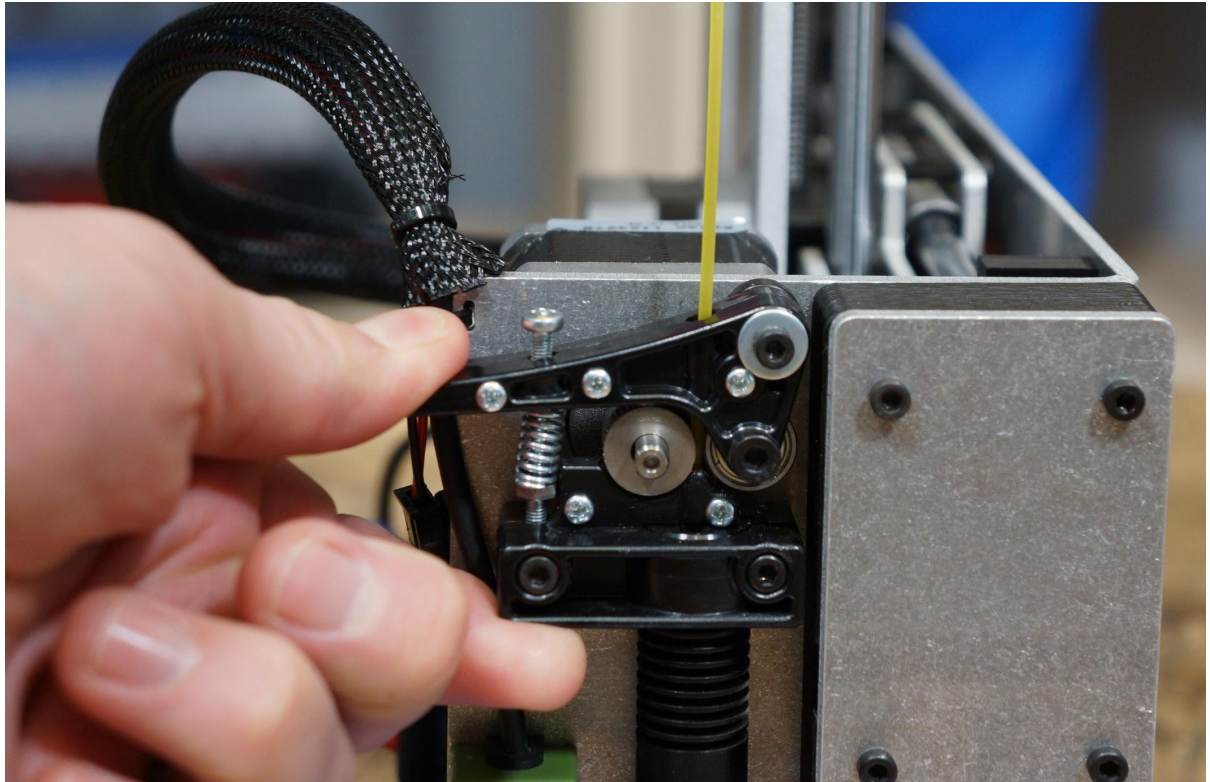


The screenshot shows a web browser window with the URL <https://printrbot.com/project/smalls/>. The page features the Printrbot logo and navigation links for Products, Education, Distribution, and Support. The main content area is titled "Build Videos and Next Steps" and includes a video player with the text "Build Videos" and "Assembling the Smalls Kit with Brook". To the right of the video are three orange buttons: "Required Downloads", "Download Printable Files for the Smalls", and "Visit Our User Forum". A white arrow points from the "REQUIRED DOWNLOADS" text in the left column to the "Required Downloads" button.

Load Your Filament

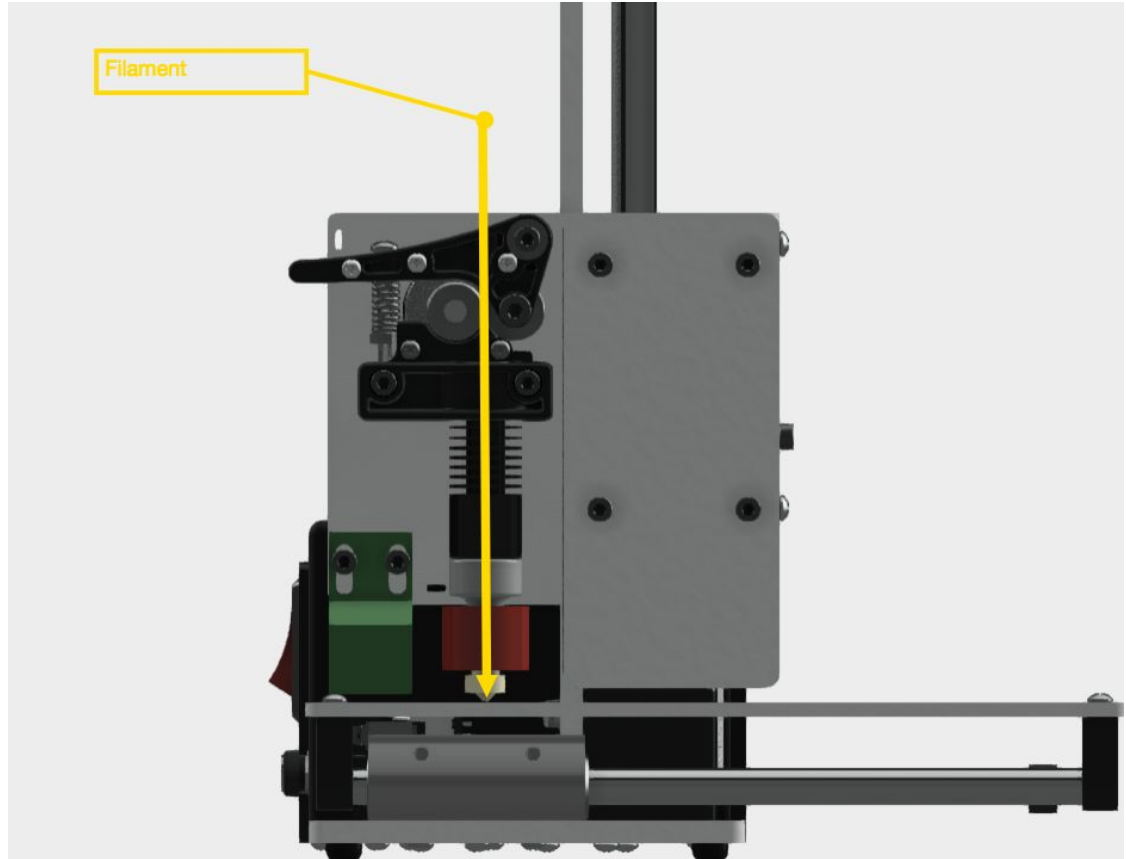
- Insert your filament into the top of the fan shroud.
- Direct the filament down into hole in the extruder arm.
- Pinch the extruder arm and continue to push the filament down into the hot end.
- Take a moment to add print bed tape to your bed.

See a more detailed explanation of how to load filament on the next slide.



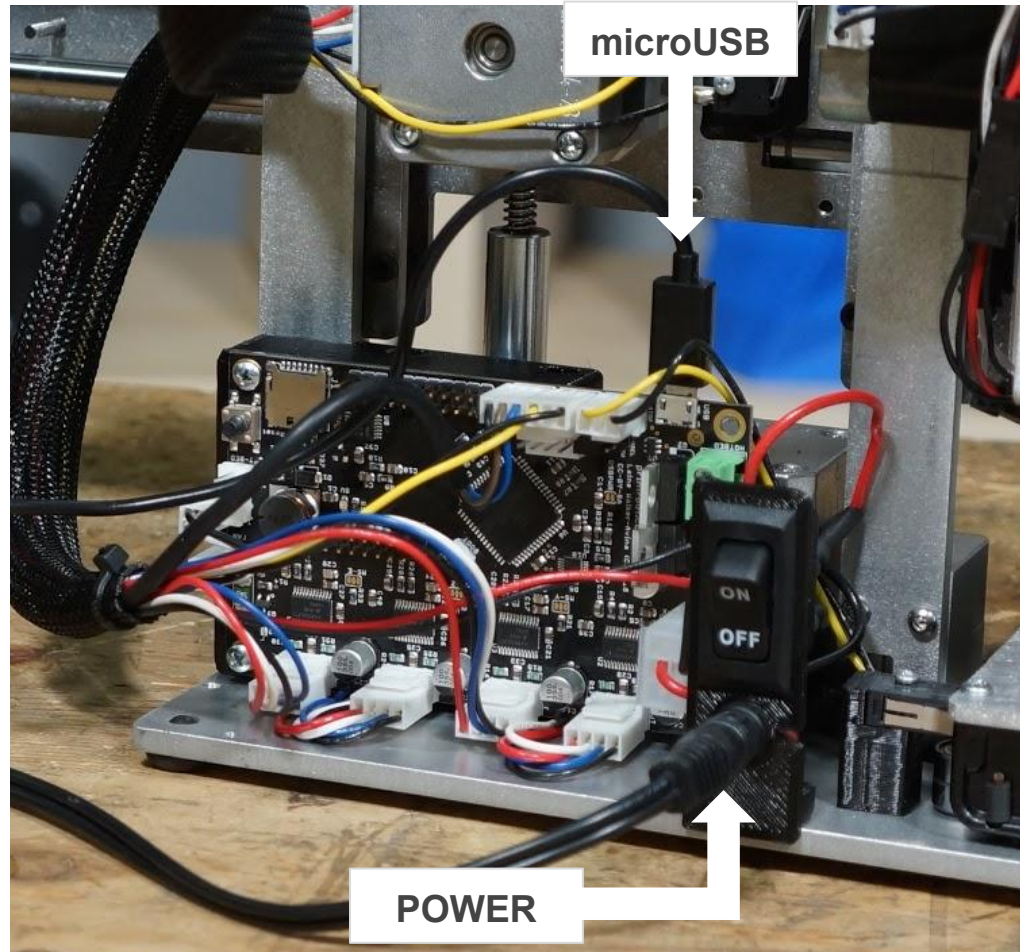
Loading Your Filament ...a closer look

- Insert your filament into the top of the extruder arm,
 - ...between the drive gear and pulley
 - ...through the extruder base
 - ...and down into the hot end.



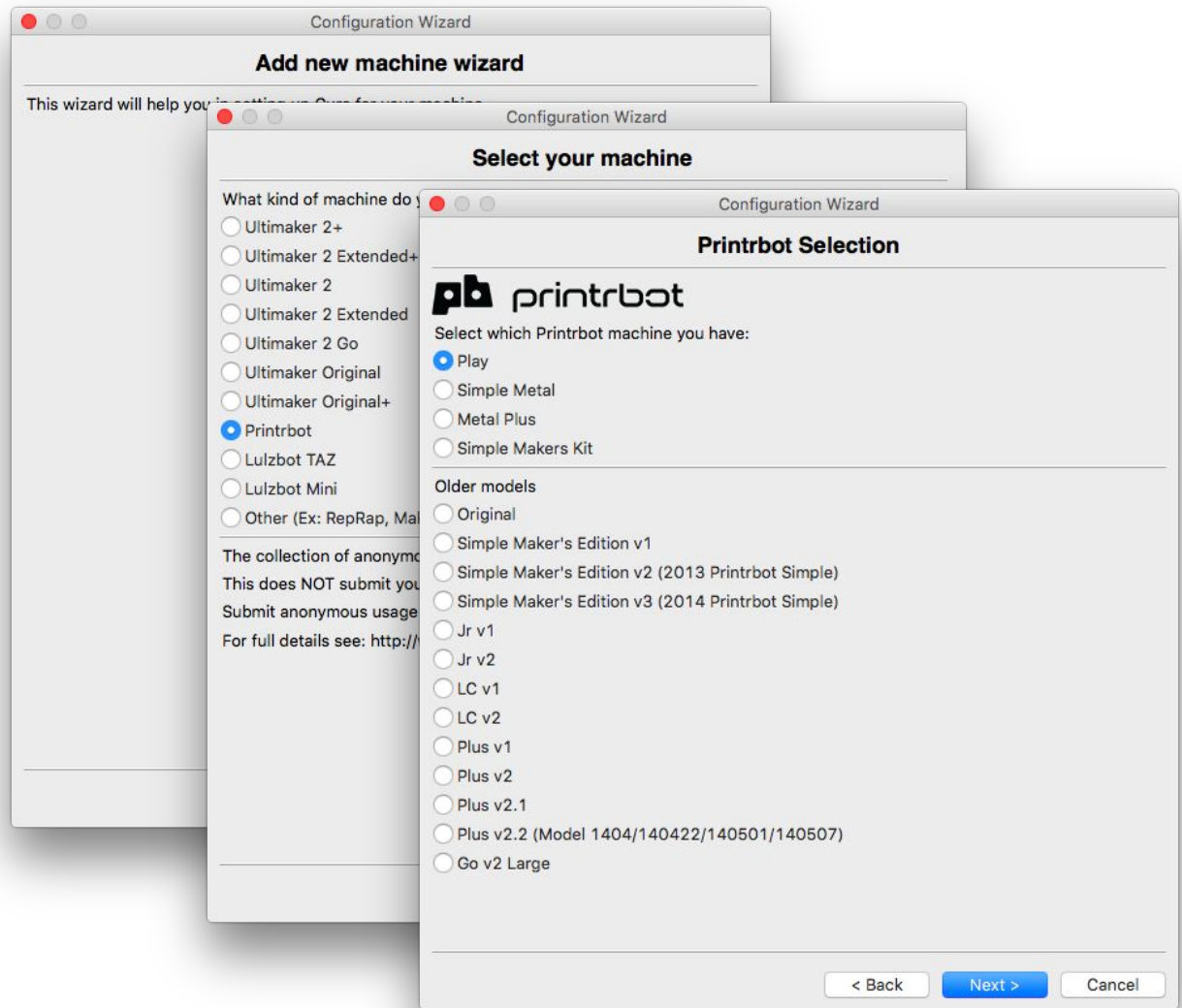
Power & USB

- Plug the microUSB cable into the USB port.
- **NOTE: DO NOT PLUG THE MICRO USB CONNECTOR IN IMPROPERLY. THIS WILL DAMAGE YOUR PRINTRBOT.**
- Connect the other end of the USB cable to your computer.
- Connect the power cord to the laptop power supply.
- Plug the laptop power supply into the power port on the side of the bot.
- Flip the power switch to the ON position.



Cura: Configuration Wizard

- After installing Cura on your computer, open the software and walk through the Configuration Wizard.
 - Select “Printrbot”
 - Select “Play”
- Continue through the Configuration Wizard by clicking “Next, Next...”

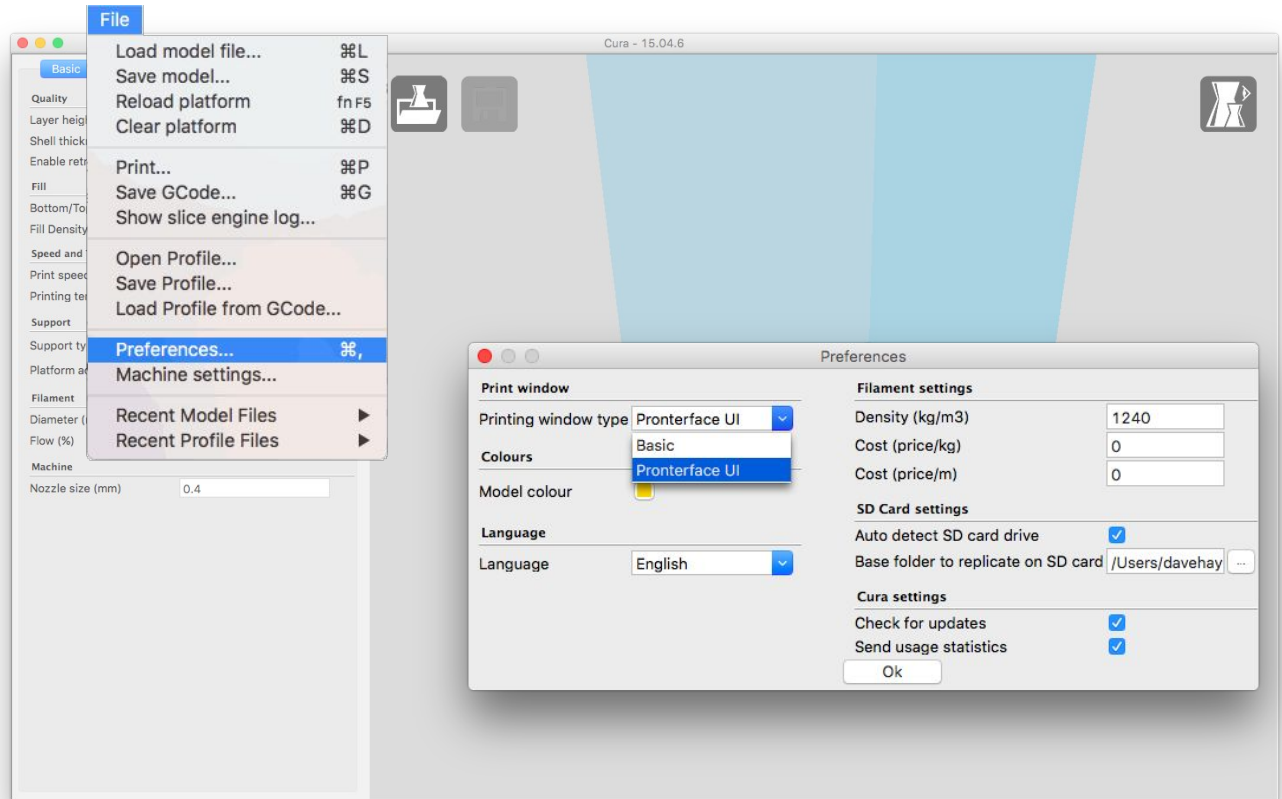


Cura: Pronterface User Interface - **IMPORTANT**

After completing the Configuration Wizard, the Cura control panel will open. Enable the Pronterface User Interface (UI).

- **File**
- **Preferences...**
- **Pronterface UI**

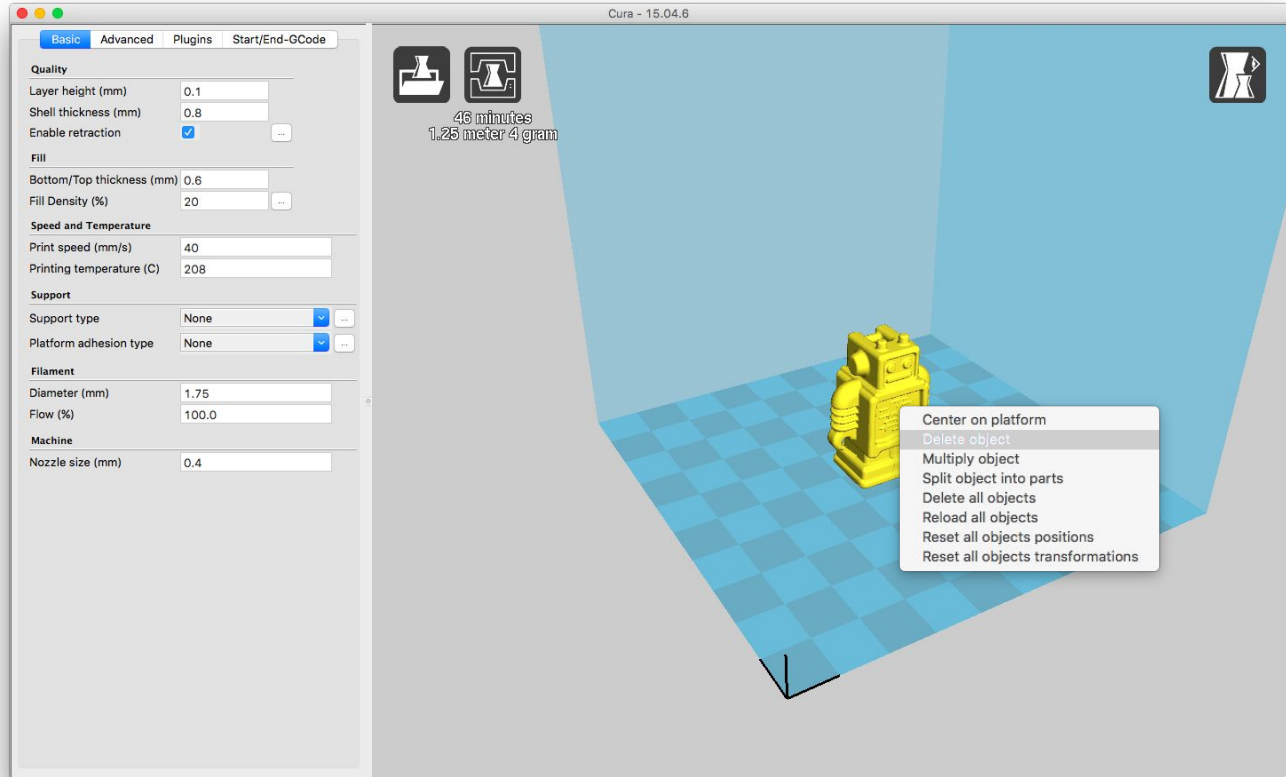
Click "Ok" to save.



Cura Control Panel

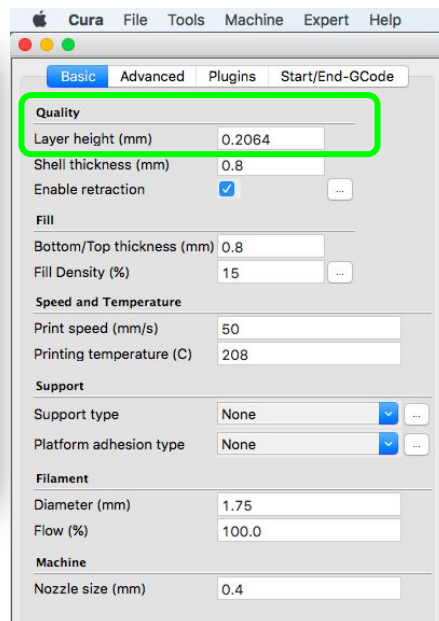
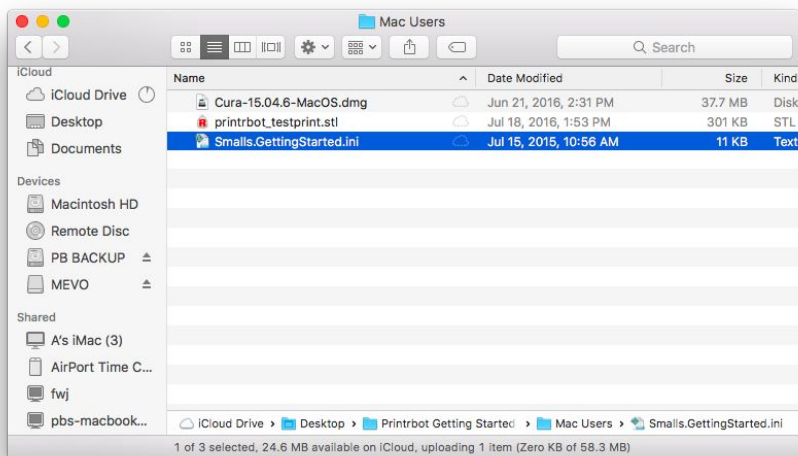
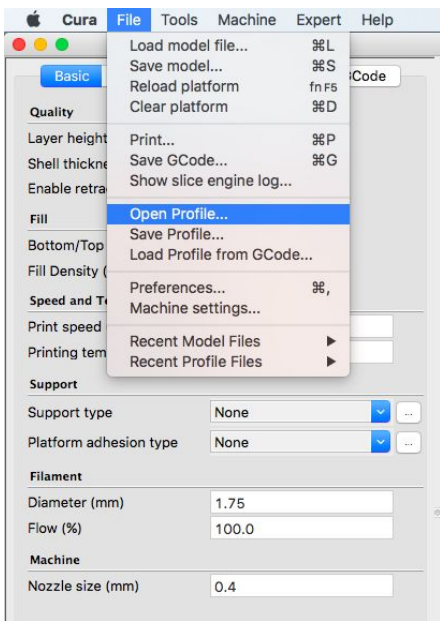
- The first time that you open Cura, you will see the Ultimaker Robot on the print panel.
- To remove this model, right click on the robot and select “Delete object”.

You can use this model as your first print if you really want to. No hard feelings :).

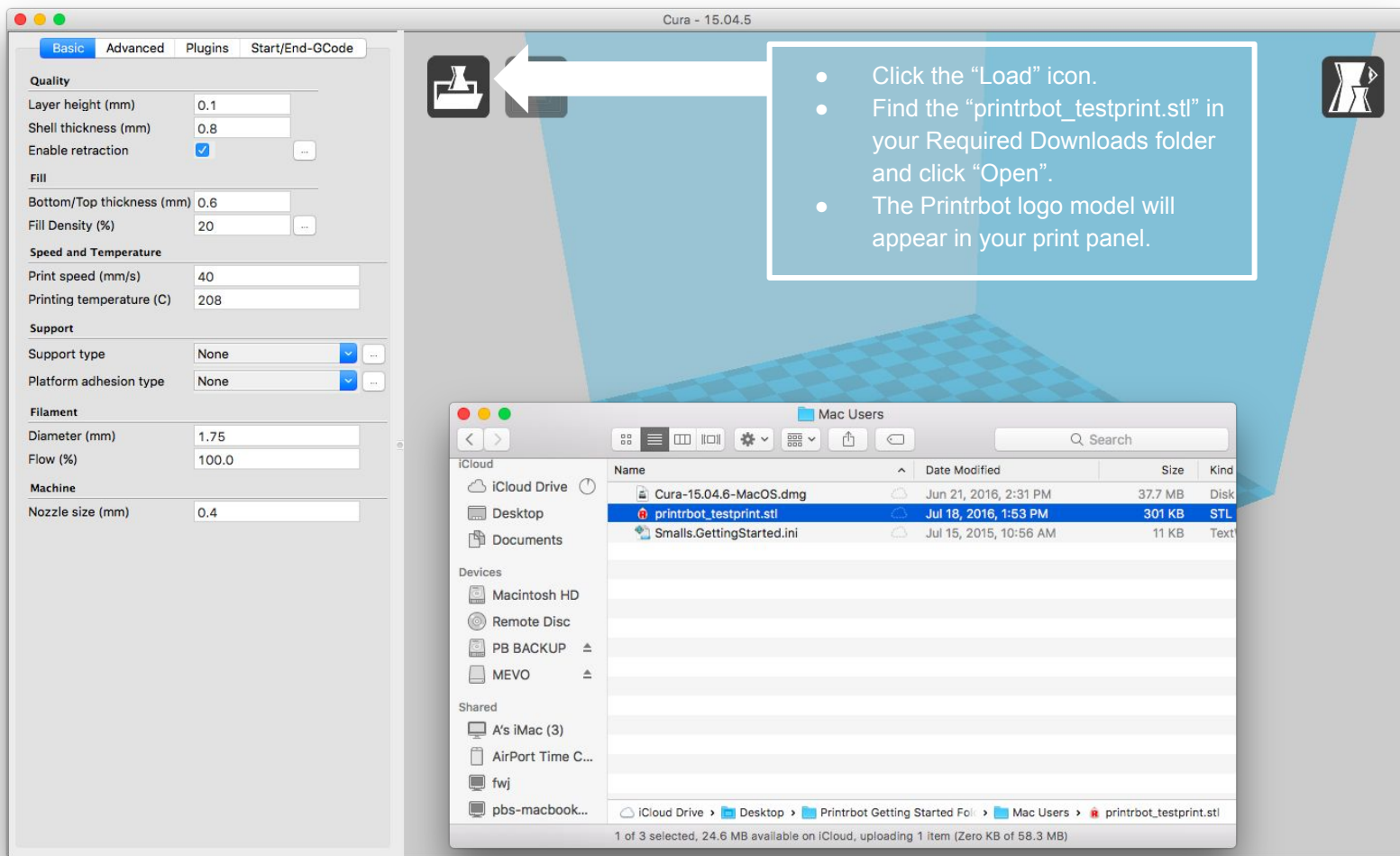


Load Configuration Profile

- Go to File/Open profile
- Select the “Smalls.GettingStarted.ini” from your Required Downloads folder.
- *TIP: You can verify that the new settings are in place by checking your Layer height (mm). It should read “0.2064”.*



Load Your Model



The image shows the Cura software interface with a 'Load' icon highlighted by a white arrow. A text box on the right provides instructions for loading a model. Below the main interface, a file browser window shows the file 'printbot_testprint.stl' selected in the 'Mac Users' directory.

- Click the “Load” icon.
- Find the “printbot_testprint.stl” in your Required Downloads folder and click “Open”.
- The Printrbot logo model will appear in your print panel.

File Browser Details:

Name	Date Modified	Size	Kind
Cura-15.04.6-MacOS.dmg	Jun 21, 2016, 2:31 PM	37.7 MB	Disk
printbot_testprint.stl	Jul 18, 2016, 1:53 PM	301 KB	STL
Smalls.GettingStarted.ini	Jul 15, 2015, 10:56 AM	11 KB	Text

Basic Advanced Plugins Start/End-GCode

Quality

Layer height (mm) 0.2064
Shell thickness (mm) 0.8
Enable retraction

Fill

Bottom/Top thickness (mm) 0.8
Fill Density (%) 15

Speed and Temperature

Print speed (mm/s) 50
Printing temperature (C) 208

Support

Support type None
Platform adhesion type None

Filament

Diameter (mm) 1.75
Flow (%) 100.0

Machine

Nozzle size (mm) 0.4



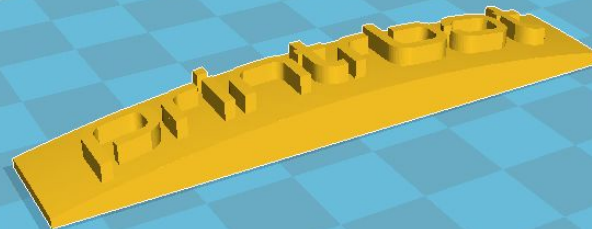
15 minutes
1.05 meter 3 gram

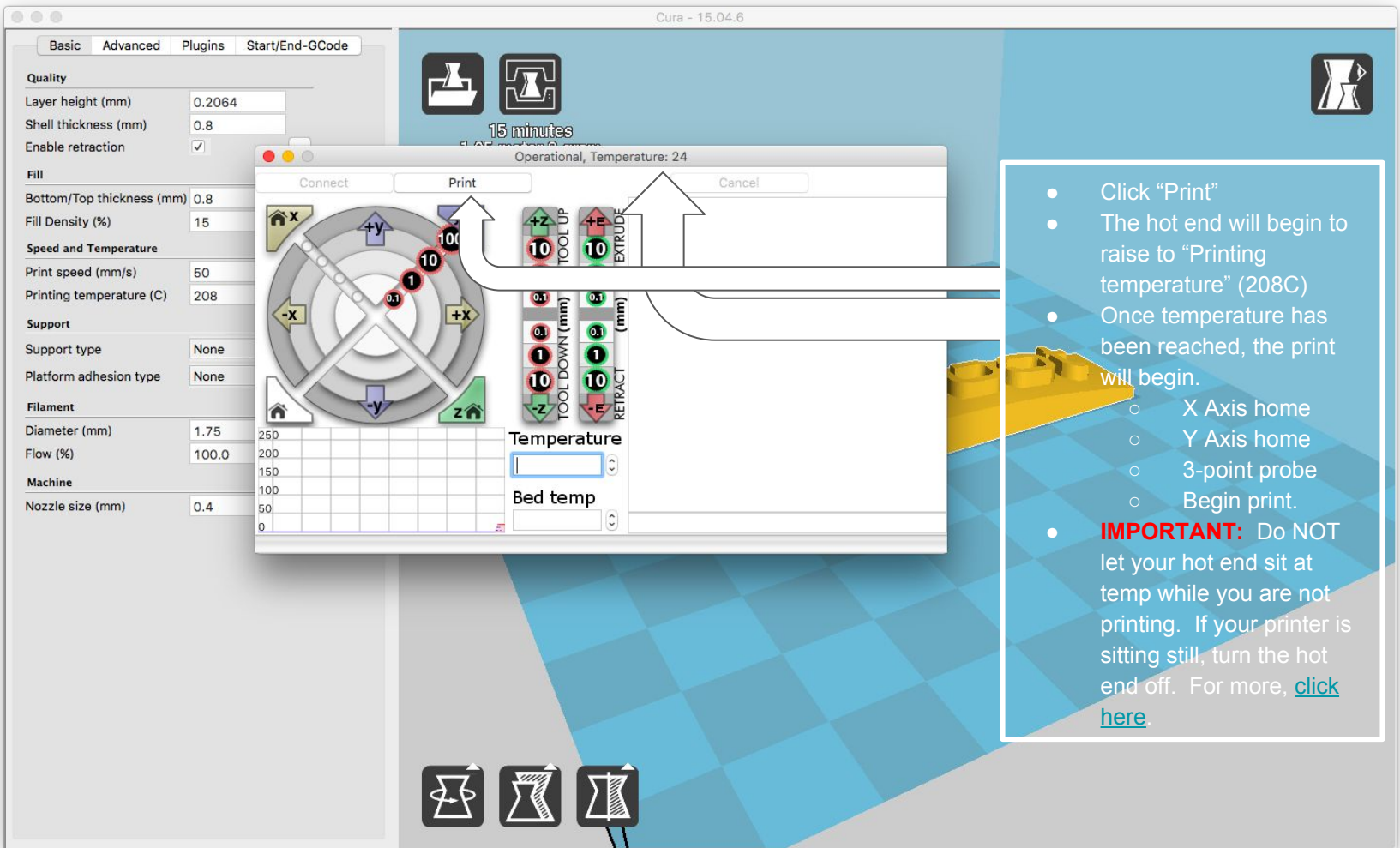


- Click the "Print with USB" icon.



**Not seeing the "Print with USB" icon? Check out our guide on [Troubleshooting Connection](#).*

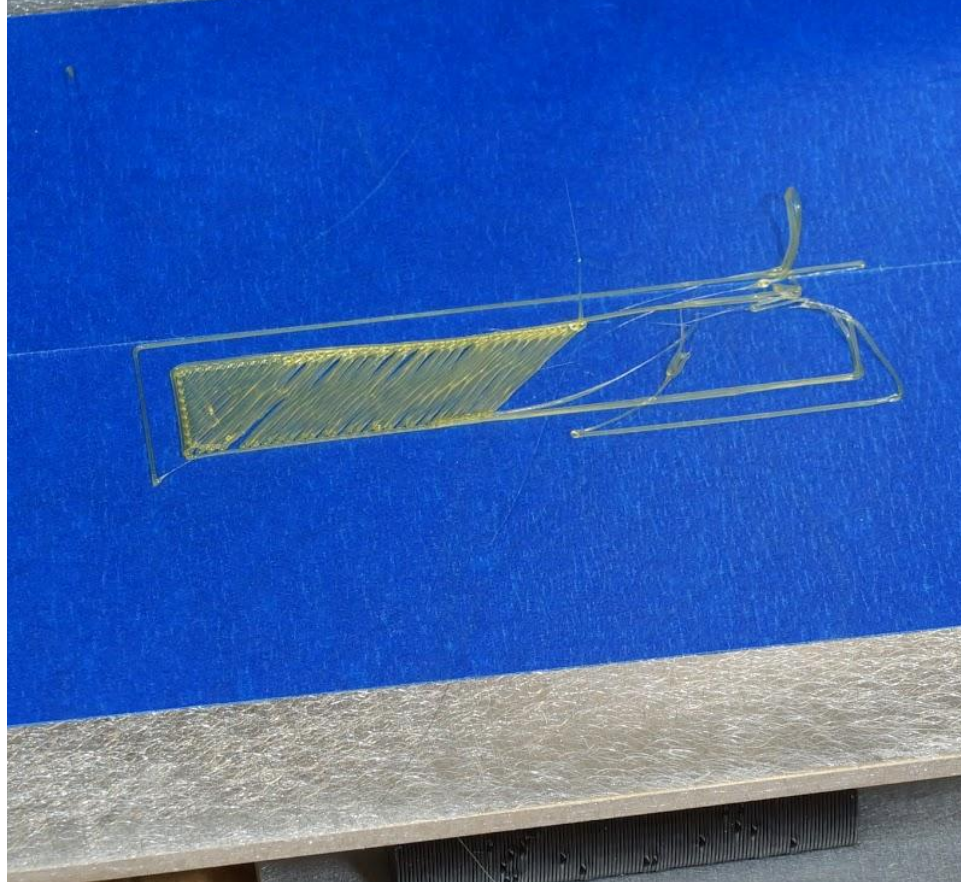




- Click “Print”
- The hot end will begin to raise to “Printing temperature” (208C)
- Once temperature has been reached, the print will begin.
 - X Axis home
 - Y Axis home
 - 3-point probe
 - Begin print.
- **IMPORTANT:** Do NOT let your hot end sit at temp while you are not printing. If your printer is sitting still, turn the hot end off. For more, [click here](#).

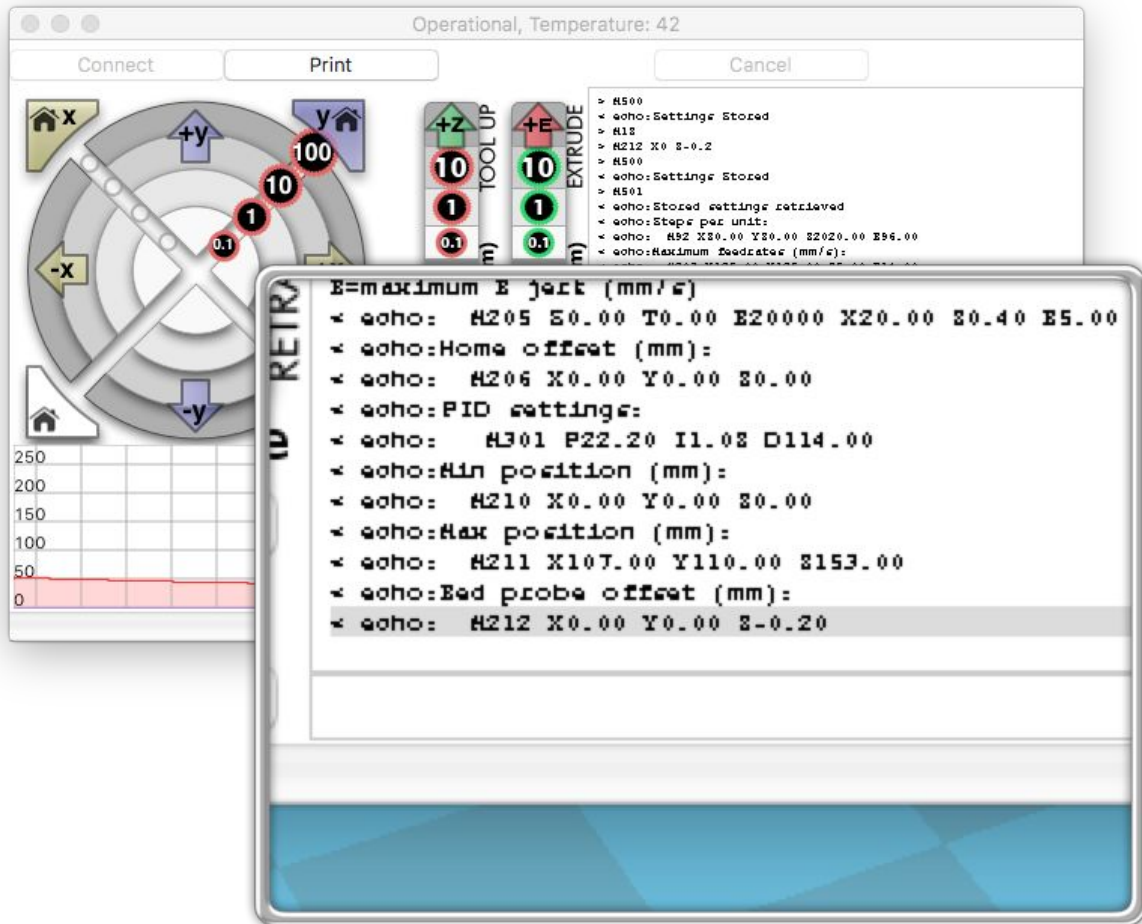
Z Offset Too High

- Your first print will likely look “stringy”. This is because the Z offset (M212) needs to be adjusted.
- Cancel the print job and remove the filament from the print bed.
- The next slide will explain how to adjust the Z offset in order for the first layer to lay down better.



Adjust M212 Z (and X) Values

- Enter the following code in the command line of your Pronterface UI. ...Click “Send” after each of these lines.
 - M212 X0 Z-0.2
 - M500
 - M501
- After entering M501, you will see the settings that you just entered in the readout.
- **Pro Tip:** Send an “M106” command between prints to power ON your fan and keep the heat sink on your hot end cool.



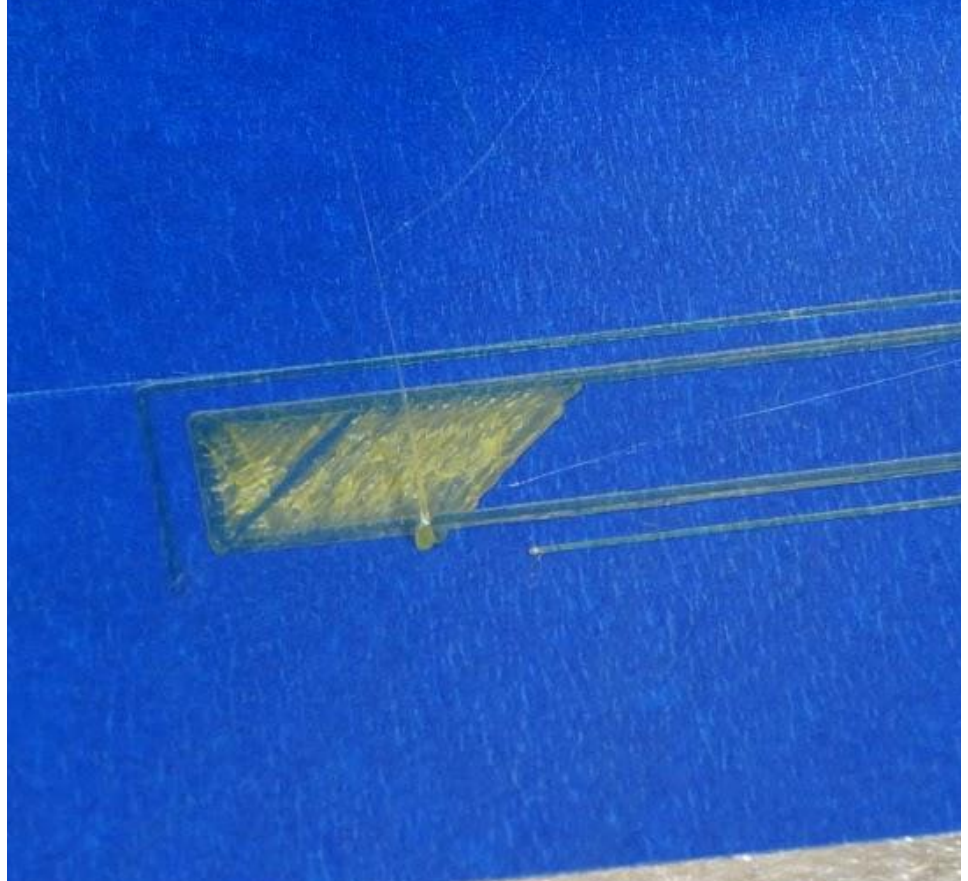
The screenshot shows the Pronterface software interface. At the top, it says "Operational, Temperature: 42". Below that are buttons for "Connect", "Print", and "Cancel". The main control area features a circular dial with directional arrows for X, Y, Z, and E, and a grid for setting values. A terminal window is open, displaying the following G-code commands and their responses:

```
> M500
< echo: Settings Stored
> M12
< echo: M212 X0 Z-0.2
> M500
< echo: Settings Stored
> M501
< echo: Stored settings retrieved
< echo: Steps per unit:
< echo: M52 X30.00 Y20.00 Z2020.00 E96.00
< echo: Maximum feedrate (mm/s):

E=maximum E jerk (mm/s)
< echo: M205 E0.00 T0.00 E20000 X20.00 Z0.40 E5.00
< echo: Home offset (mm):
< echo: M206 X0.00 Y0.00 Z0.00
< echo: PID settings:
< echo: M301 P22.20 I1.08 D114.00
< echo: Min position (mm):
< echo: M210 X0.00 Y0.00 Z0.00
< echo: Max position (mm):
< echo: M211 X107.00 Y110.00 Z153.00
< echo: Bed probe offset (mm):
< echo: M212 X0.00 Y0.00 Z-0.20
```

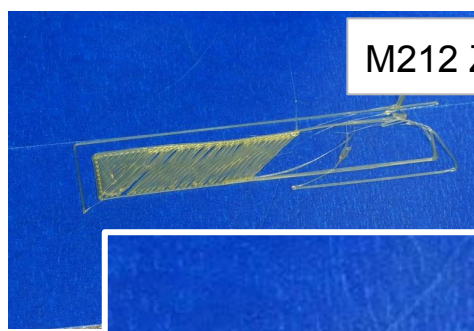

Z Offset Too Low

- If you adjust the M212 too dramatically, the first layer will appear “choppy”. You may also notice the filament almost looks transparent because such a small amount was allowed out of the nozzle.
- If your extruder is clicking, that is another sign that the Z offset is too low.

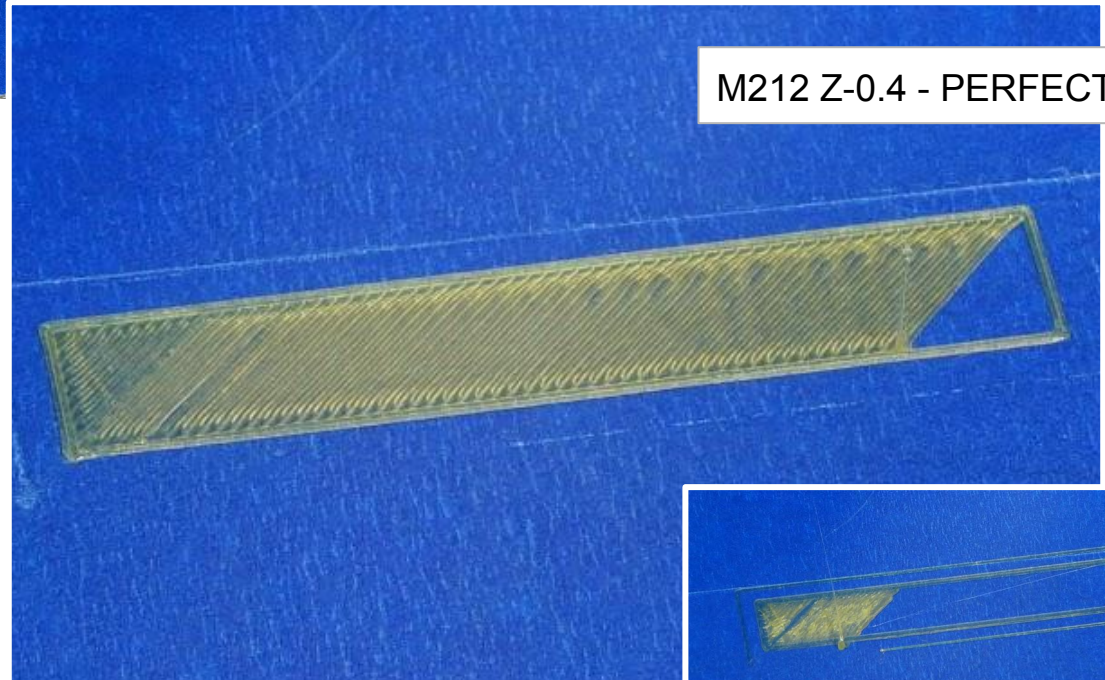


Test Again ...and Again ...and Again.

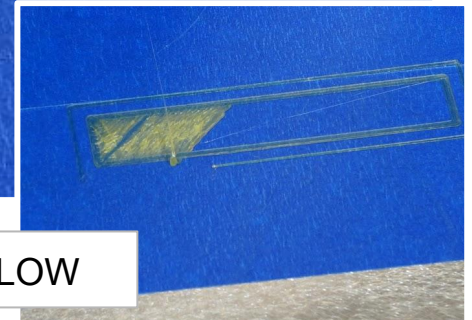
- Send the following commands. Press “Send” after each one.
 - M212 Z-0.2
 - M500
 - M501
- After adjusting your M212 value, click “Print” again.
- You may not see much difference between this print and the first one. Keep testing: Start the print ...cancel ...adjust M212 ...start the print, etc. Continue to adjust the M212 Z offset into the negative direction in small increments. No more than 0.2 each time. So for example ...M212 Z-0.2 ...M212 Z-0.4 ...M212 Z-0.6, and so on.
- **NOTE: Your “perfect” M212 value may not be Z-0.4. You will have to test and find the proper value for your printer.**



M212 Z0.0 - TOO HIGH



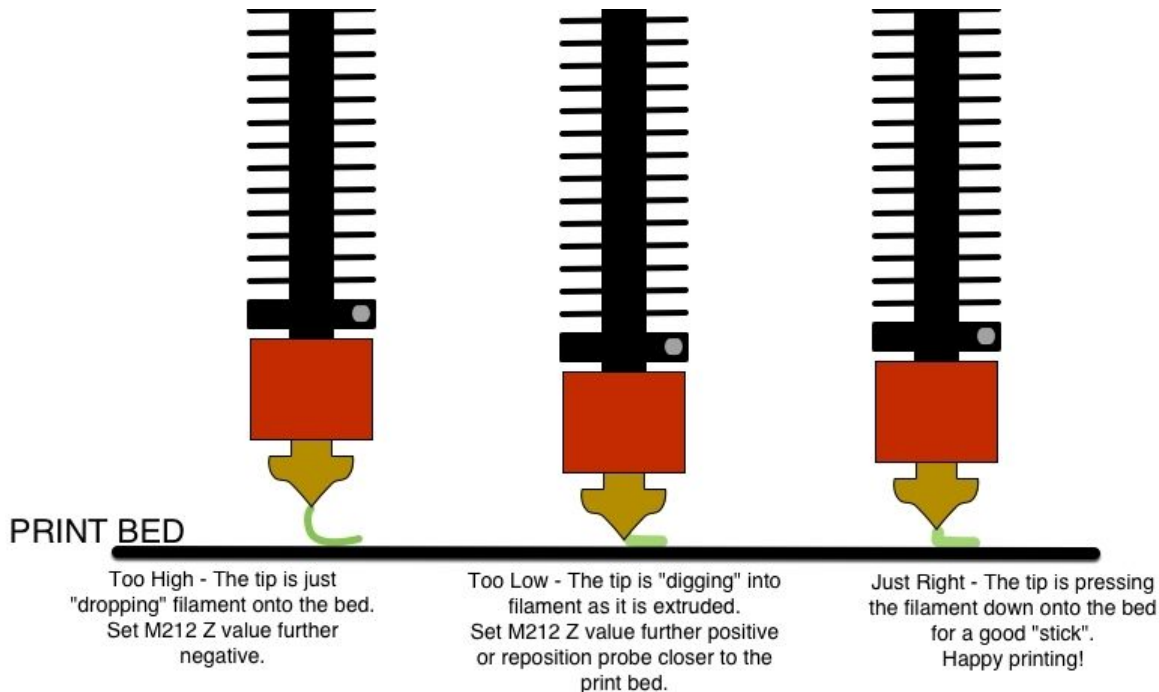
M212 Z-0.4 - PERFECT



M212 Z-0.6 - TOO LOW

The Perfect "Bead"

- You're looking for the perfect bead. If the value is too high, you will not achieve first layer adhesion. If the value is too low, you will hear the extruder click as the filament backs up in the hot end.
- When the M212 value is set properly, the first layer sticks well, without jamming the hot end.



The Perfect "Bead"

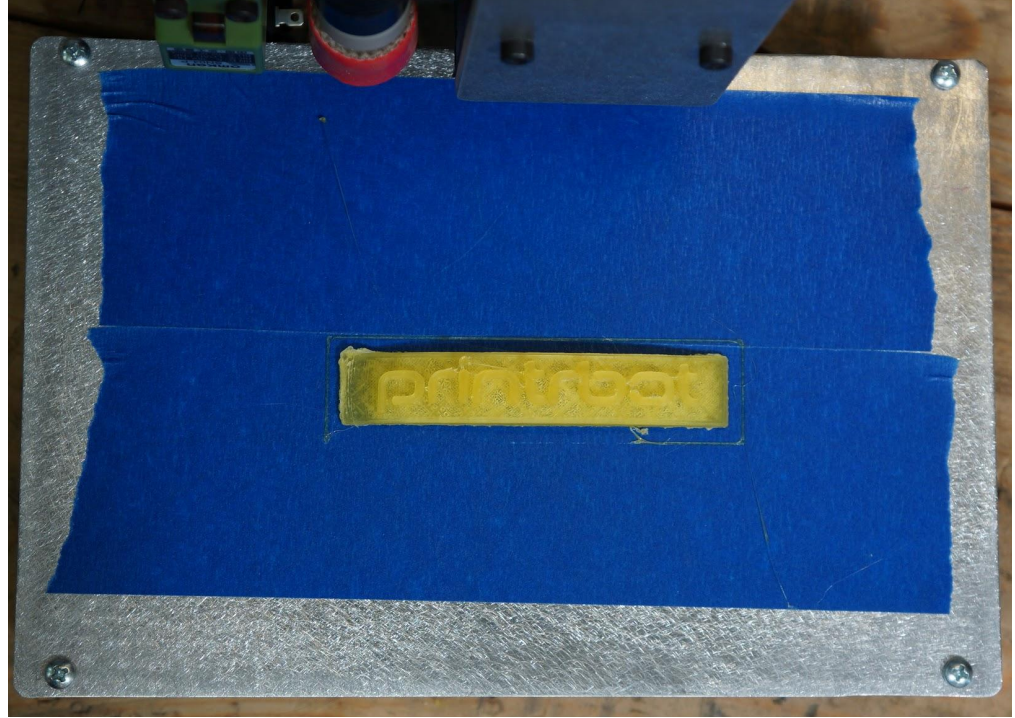
Nice Work! First Print Complete.

NEXT STEPS:

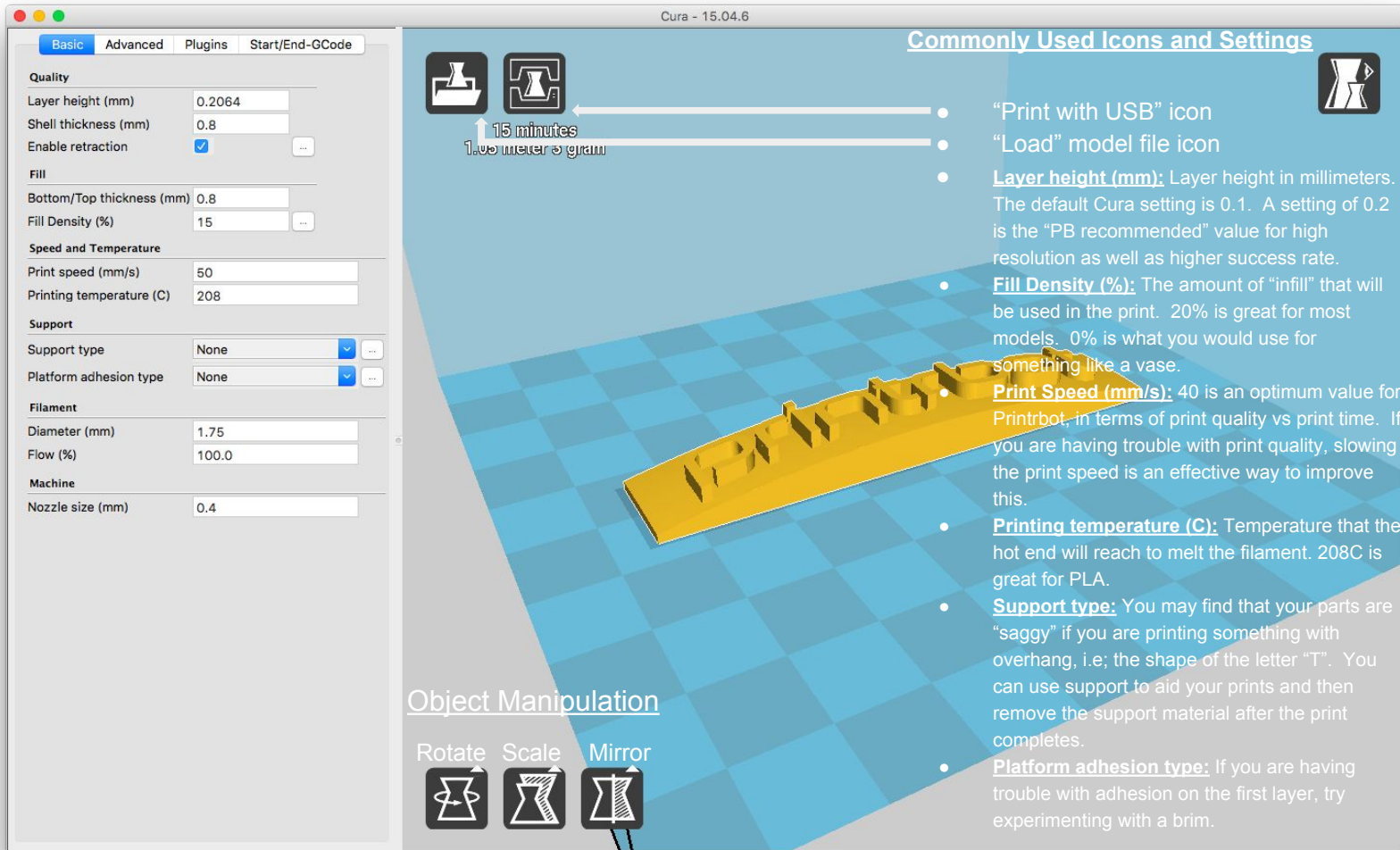
- Visit youmagine.com for loads of free printable files, including parts specifically designed for your Printrbot Smalls.
- See the next slide for more insight on Cura settings/terminology.
- [Download Fusion 360](#) and make your own designs!
 - *Note: The card lists an expiration of 3/31/17, but Autodesk has been generous to extend their offer!*

One year of Fusion 360 FREE with the offer code:
printroff

- No luck? Don't be discouraged. 3D printing is hard.
- Visit our forum at support.printrbot.com. We have section specifically for Smalls Kit builders!



Cura Settings Quick Reference for Printrbot Users



Cura - 15.04.6

Commonly Used Icons and Settings

- “Print with USB” icon
- “Load” model file icon
- **Layer height (mm):** Layer height in millimeters. The default Cura setting is 0.1. A setting of 0.2 is the “PB recommended” value for high resolution as well as higher success rate.
- **Fill Density (%):** The amount of “infill” that will be used in the print. 20% is great for most models. 0% is what you would use for something like a vase.
- **Print Speed (mm/s):** 40 is an optimum value for Printrbot, in terms of print quality vs print time. If you are having trouble with print quality, slowing the print speed is an effective way to improve this.
- **Printing temperature (C):** Temperature that the hot end will reach to melt the filament. 208C is great for PLA.
- **Support type:** You may find that your parts are “saggy” if you are printing something with overhang, i.e; the shape of the letter “T”. You can use support to aid your prints and then remove the support material after the print completes.
- **Platform adhesion type:** If you are having trouble with adhesion on the first layer, try experimenting with a brim.

Object Manipulation

- Rotate
- Scale
- Mirror

15 minutes
1.00 meter 5 gram

Category	Setting	Value
Quality	Layer height (mm)	0.2064
	Shell thickness (mm)	0.8
	Enable retraction	<input checked="" type="checkbox"/>
Fill	Bottom/Top thickness (mm)	0.8
	Fill Density (%)	15
Speed and Temperature	Print speed (mm/s)	50
	Printing temperature (C)	208
Support	Support type	None
	Platform adhesion type	None
Filament	Diameter (mm)	1.75
	Flow (%)	100.0
Machine	Nozzle size (mm)	0.4