Printrbot

Getting Started Guide

For models:
Simple

Contents

4 **Chapter 1: Welcome**
4 Overview
4 Support

5 **Chapter 2: Your Printrbot has arrived... now what?**
5 Printrbot Kits
5 Pre-Assembled Printrbots

6 **Chapter 3: Building your Printrbot**
6 Required and optional tools
6 Assembly instructions
6 Help during assembly

7 **Chapter 4: Install Software**

8 **Chapter 5: Connect your Printrbot**

9 **Chapter 6: Configure Software**
9 Configure Repetier printer settings
9 Connection
10 Behavior
10 Dimension
11 Configure Slic3r settings
11 Layers and perimeters
12 Infill
12 Skirt and brim
13 Filament
13 Cooling
14 General
14 Extruder 1

15 **Chapter 7: Functional Pre-Print Test**
15 Manually Set Home Position
16 Connect to the printer
16 Test the Z axis
17 Test the X axis
17 Test the Y axis
18 Test the extruder with initial burn-in temperature
18 Troubleshooting

19 **Chapter 8: Printing your First 3D Model**
19 Download file
19 Add .STL file

20 Slice
20 Connect
21 Set temperatures
21 Temperature curve
21 Run
21 After the print completes
21 Congratulations

22 Chapter 9: Advanced Printing
22 Technical Forums
22 Community

23 Chapter 10: Common Issues and Troubleshooting
23 Filament is not extruding
23 Print not sticking to bed
23 Head is dragging across the print
23 Print not correct size (G-Code)
24 Other problems

Welcome

1

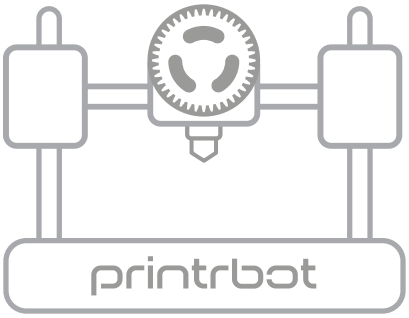
Read this guide to learn about your Printrbot, how to make your first 3D print, and more.

Overview

The purpose of this guide is to get you printing your 3D models as soon as possible! Whether you have a pre-assembled Printrbot or a build it yourself kit, this guide will walk you through the necessary steps to a successful first print. So let's get going!

Support

If at any time you need help or get stuck, please check our FAQ page and then open a support ticket if necessary. www.printrbot.com/support/



Your Printrbot has Arrived... now what?

2

The arrival of your bot can not only be exciting, but we realize it can be overwhelming at first. If you have received an assembled bot, your work is already nearly done. However, those with a kit have a little bit more work to do. But fear not, our support team is here to help and we will make sure you are printing in no time!

Printrbot Kits

If you have a Printrbot kit, proceed to **Chapter 3: Building your Printrbot**

Pre-Assembled Printrbots

If you have a pre-assembled Printrbot, please skip to **Chapter 4: Install Software**

Building your Printrbot

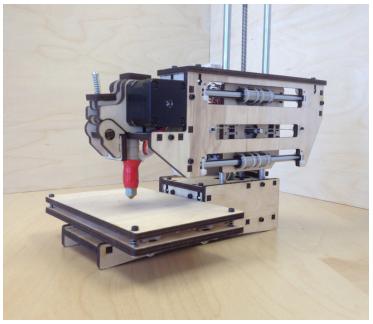
3

Required and optional tools

Before you begin to assemble your bot, please note that you are going to need several tools. Below is a listing of the typical tools used for assembling a Printrbot Plus. This list should cover you for both the LC and jr as well, however there may be a few other items you will need to purchase depending on which version Printrbot you have.

Required Tools	Optional Tools
Phillips Screwdriver 8mm or 5/16 inch Wrench 1/2 inch Wrench 2.5mm Allen Wrench 1.5mm Allen Wrench Hammer or Rubber Mallet Wood or Super Glue	utility knife flat hand file cordless drill 7/64" drill bit 9/64" drill bit 4" C Clamp micro cutters slim needle nose pliers

Assembly Instructions



Download the appropriate instructions for your model from www.printrbot.com/support/. We recommend using a combination of both the Step by Step Installation Guides as well as the Build Videos. Our designs have changed slightly since the time these instructions were originally created. However, using these in tandem will help to clarify the process during assembly.

Note: During the build process, some sub assemblies (such as the extruder) are detailed in their own build instructions.

Help During Assembly

If you get stuck during assembly, we recommend browsing our technical forum at www.printrbottalk.com as we have several power users and moderators that can give you quick assistance and troubleshooting help. However, if you are unable to resolve your issues please open a support ticket at www.printrbot.com/support/.

Install Software

4

Now that your bot is fully assembled, the next step is to install the printing software on your computer so that you can actually print a 3D model. While there are various free printing software on the market, we currently use and recommend Repetier http://www.repetier.com/ when first getting started. It's easy to use and supports Windows, Mac, and Linux. Once installed, the typical workflow is as follows:

1. Create a 3d model and export it in STL format or get it from the internet.
2. Arrange one or more models on a virtual print plate.
3. Slice the the models into thin slices and compute a path for printer head. This is done by a slicing software, which converts the model into g-code, the language your printer speaks.
4. Check the created g-code for errors and printability.
5. Send the g-code to your printer or copy the code to a SD card, which you can insert into your printer.
6. Monitor your printer.

Select your platform and download Repetier from www.repetier.com/download/ and install the software using the default configurations.

Windows users only: Download the required USB driver from http://pjrc.com/teensy/serial_install.exe, right click on it and "Run as Administrator" to install the INF file. It
should create a new COM port that you can then use in Repetier to connect to the
printer. For more information please see http://printrbot.com/wp-content/uploads/2012/04/Printrbot-Getting-Started-Guide2.pdf

Windows 8 users: If you are having trouble installing the driver, please try the steps on http://www.nextofwindows.com/how-to-install-an-un-signed-3rd-party-driver-in-windows-8/

Connect your Printrbot

5

1. Find a secure table or flat surface on which to print.
2. Place your printer close to your computer and plug in the power source for your bot.
3. Plug in the mini USB cable to the back of your Printrbot and connect the other end to a USB port on your computer.
4. You should now have the Printrbot powered on and connected to your computer.

Configure Software

6

The following settings are recommendations for your first prints. Over time you can change and modify them as you get more familiar with your printer as different 3D models often require different settings for a more accurate and precise print. The reason for this is that your 3D printer is more like a tool than an appliance. Every 3D model is unique and often requires a different "tool" to get the job done. So you will find over time that certain settings work better than others. It's also worthy to note that the printers themselves often have their own personalities and quirks as they have been assembled by hand. So expect to have small issues here and there as time goes on. With that said, let's get your software configured!

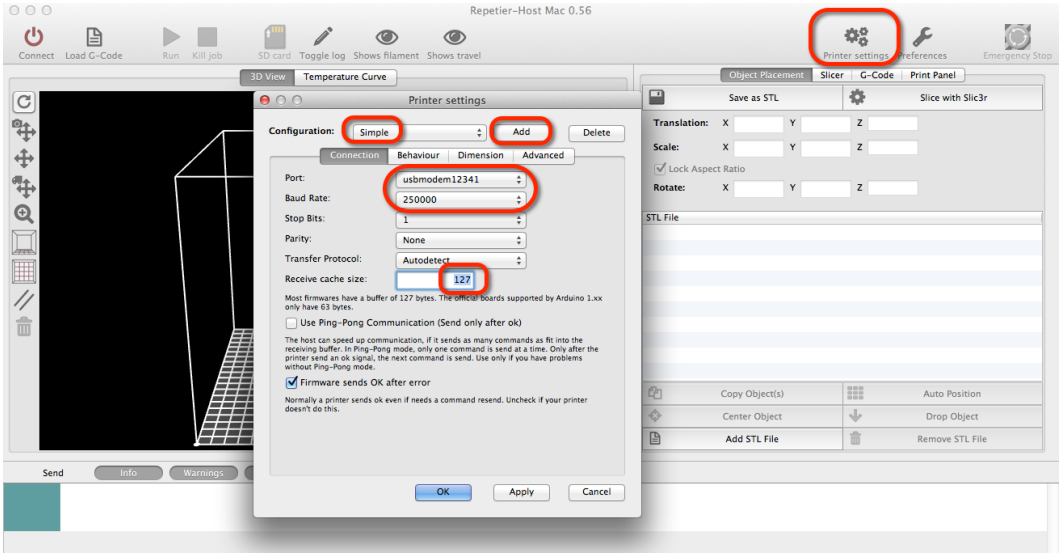
Note: The following screenshots and settings were taken from the Mac version. There may be slight variances when using the Windows and Linux versions.

Configure Repetier Printer Settings

Open Repetier and click on **Printer Settings** at the top right.

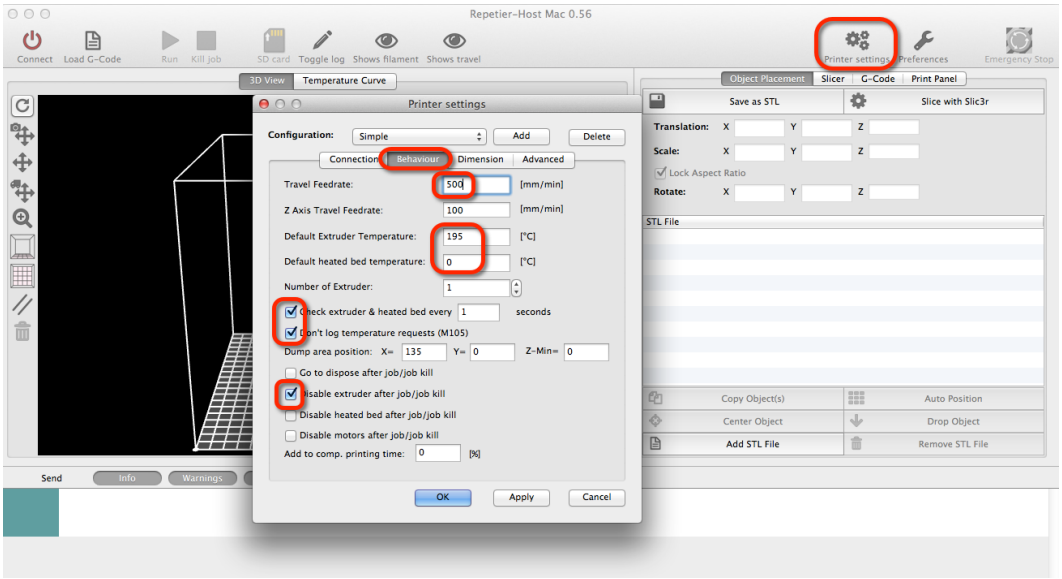
Connection

Select the port that matches your USB connection and set the Baud Rate to 250000. Click the Add button and type "Simple" into the text field that pops up. Change the cache size to 127 and click Apply and then OK.



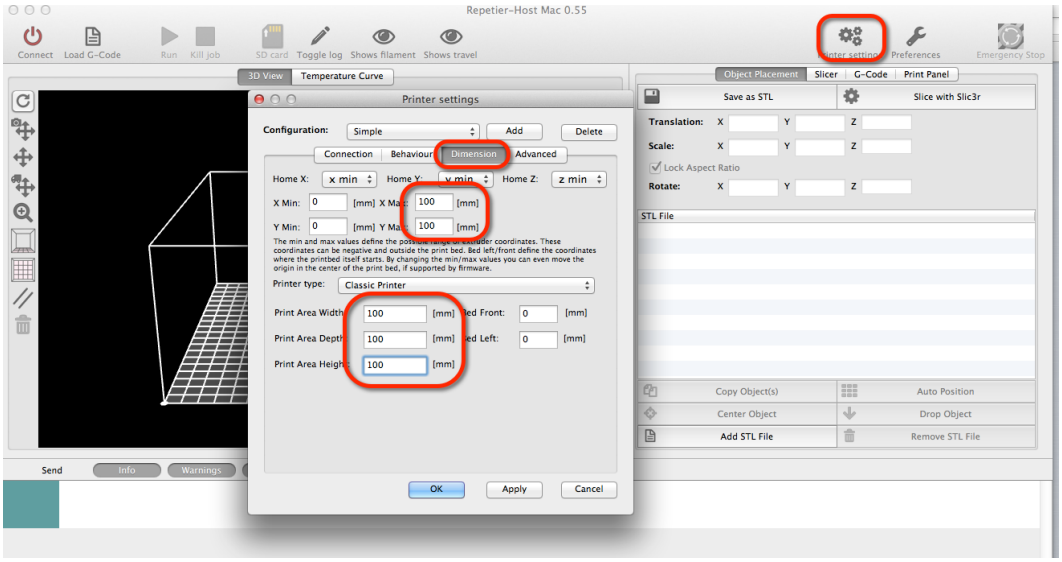
Behavior

Set the extruder temperature to 195 and the heated bed to 0 degrees. Travel Feedrate should be 500 and check the following boxes.



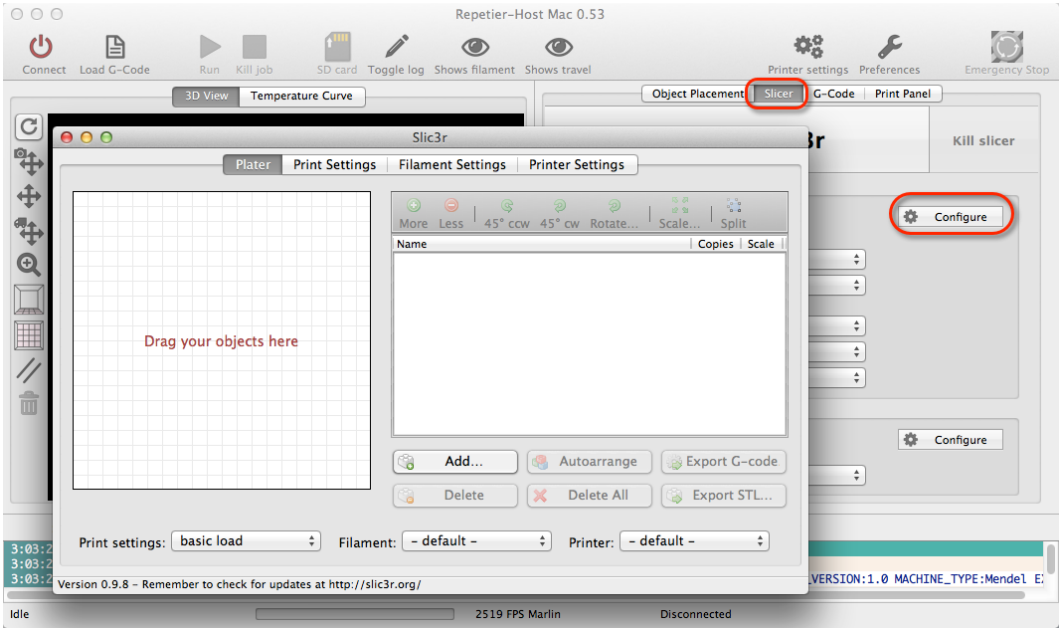
Dimension

Set all print area settings to 100mm. Click Apply and then OK.



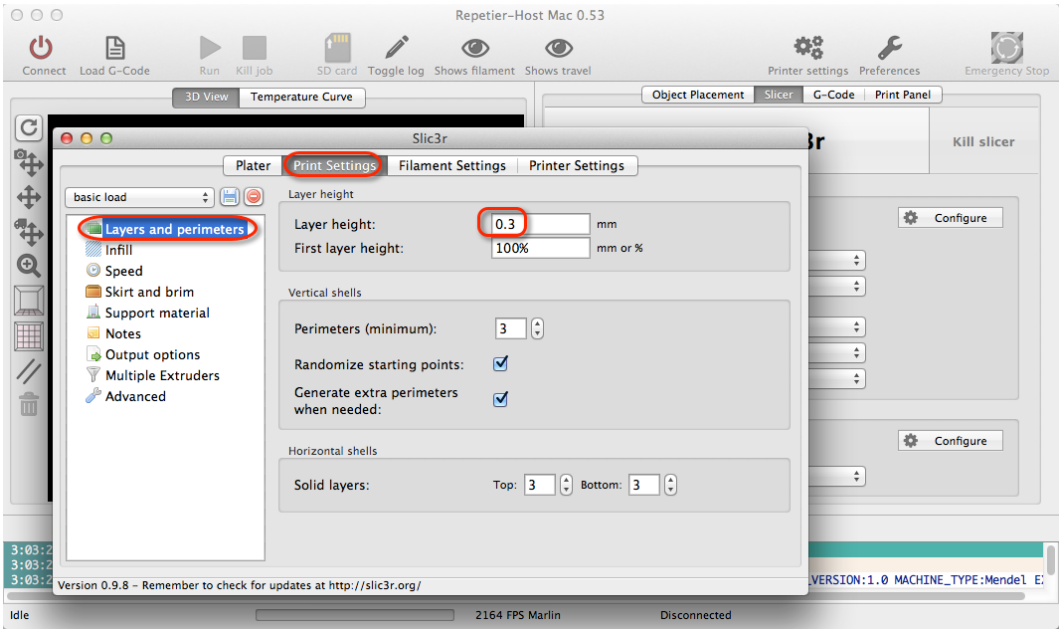
Configure Slic3r Settings

Select Slicer within Repetier and click on the Configure button



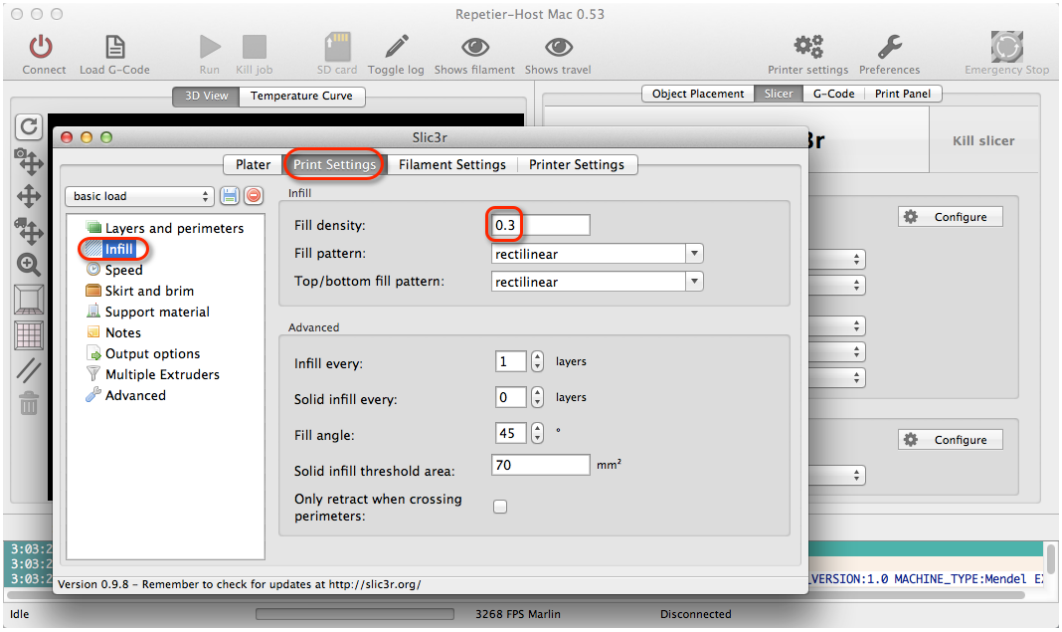
Layers and perimeters

0.3 mm is a good place to start for layer height



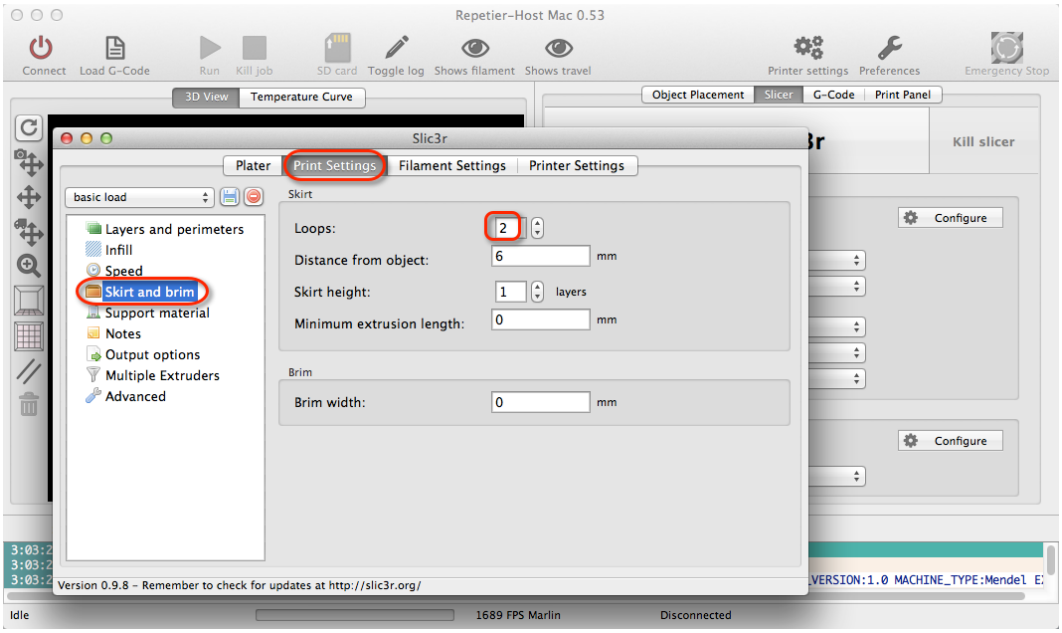
Infill

Rectilinear is a good place to start for fill patterns



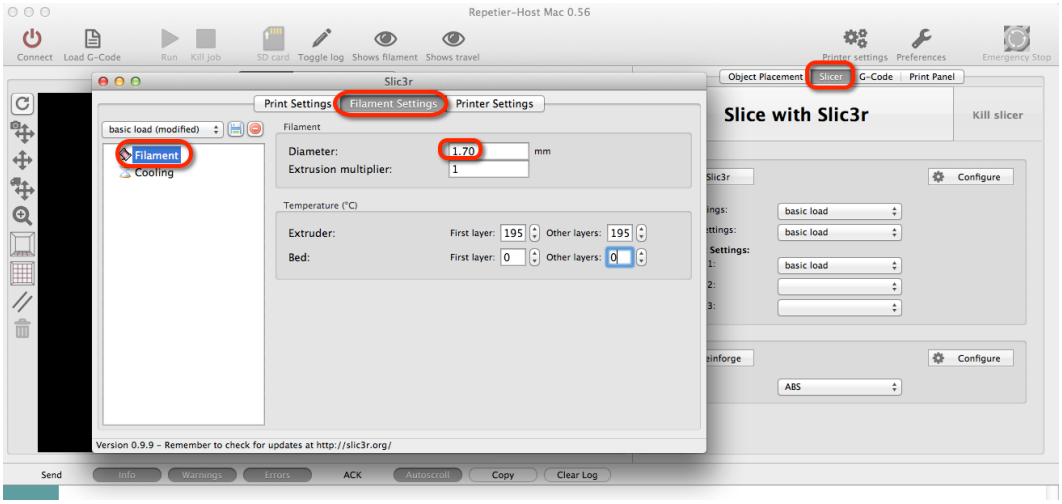
Skirt and brim

Set Loops to 2



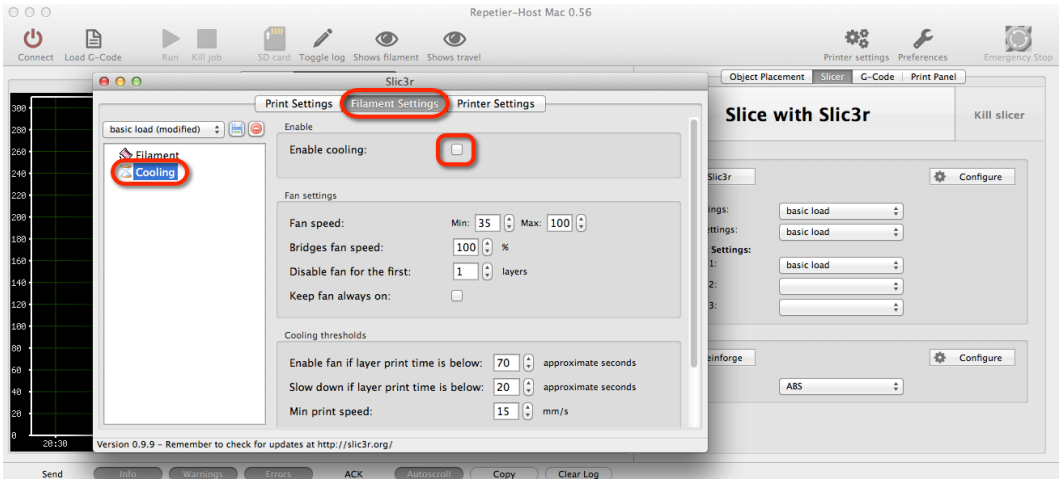
Filament

Set your diameter to 1.70



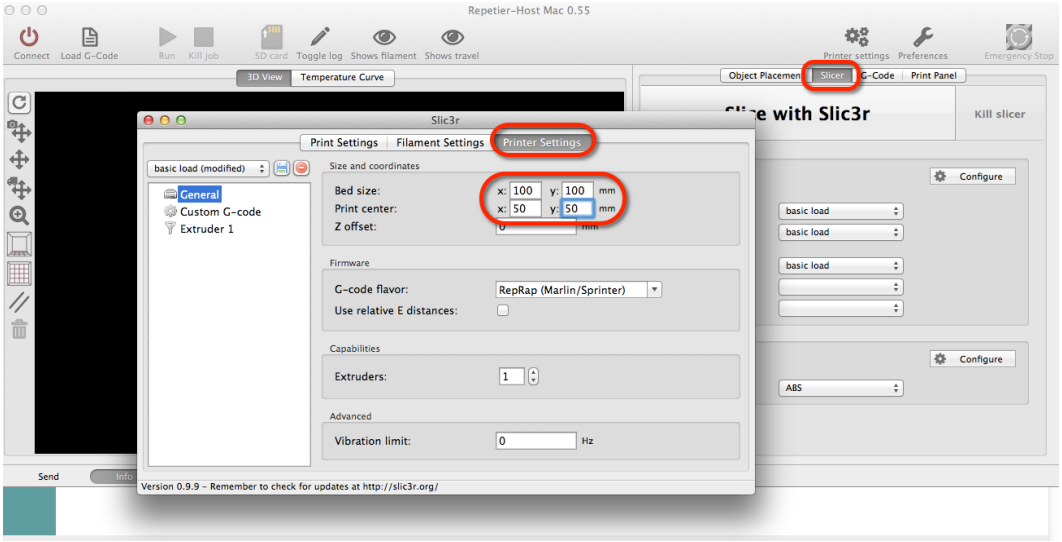
Cooling

Make sure to uncheck the cooling option as there is no fan on the Simple.



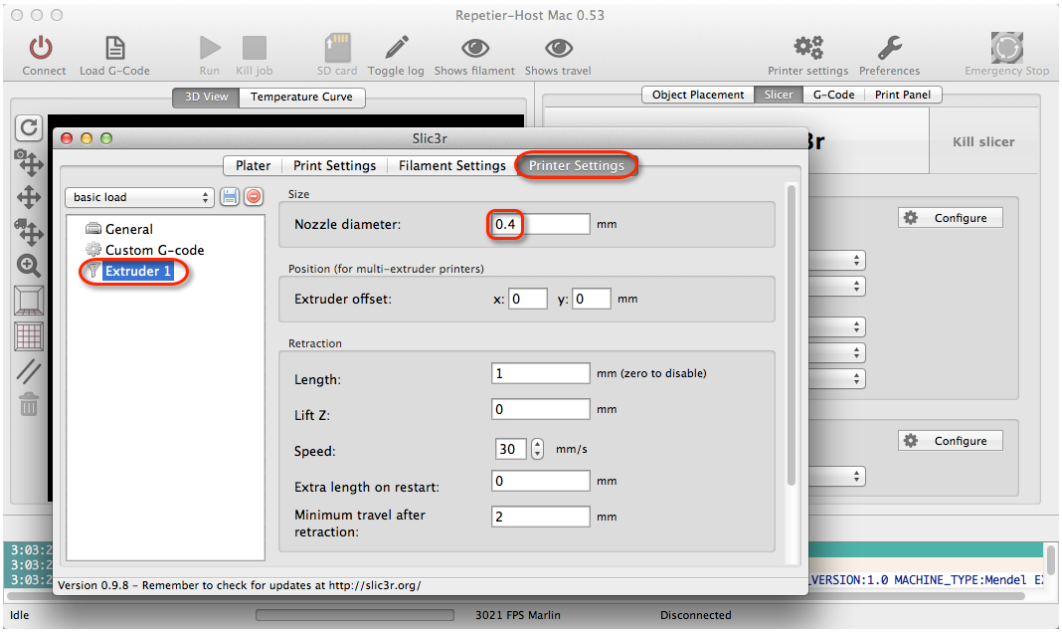
General

Make sure your bed size is set to 100mm and 50mm center points.



Extruder 1

Standard tip size with all bots is currently 0.4 mm. Otherwise enter the appropriate diameter to match the tip you are using.



Functional Pre-Print Test

7

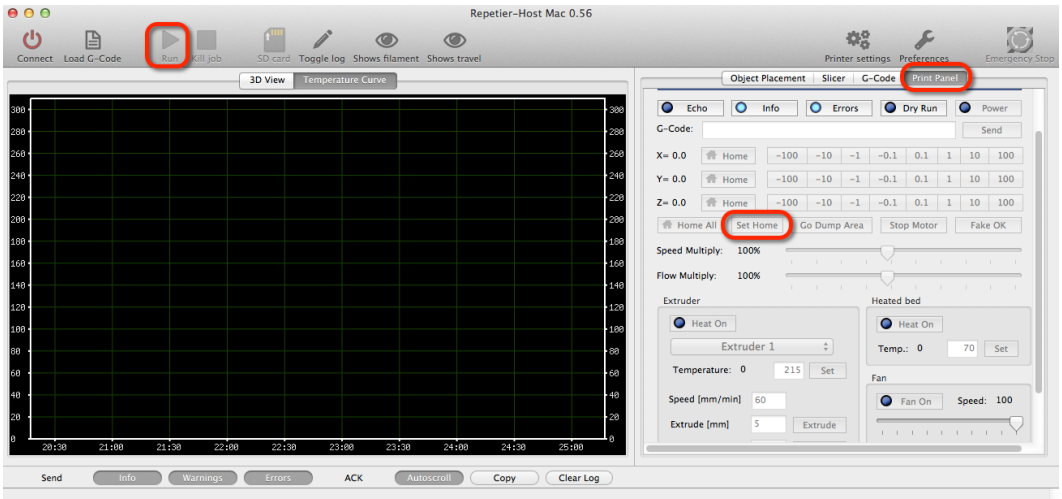
Before testing your first print, you need to make sure all of your axis are working as expected on your bot. Connect your printer to your computer and make sure both of them are powered on.

Manually Set Home Position

In order to begin print on the Simple you will need to manually home the tip of the hot end to the front, left side of the print bed. Then in Repetier you will click Set Home in the print panel. Then press Run once the STL file is loaded.

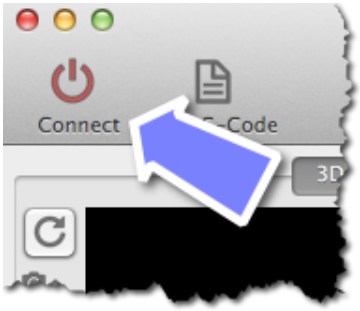
Note: If you are not using Repetier, type G-Code command "G92" after you have manually homed the hot end. Do NOT click the Home All button on Print Panel. You must manually set this position as stated above. You have to manually move the print head and then press Set Home before every print.

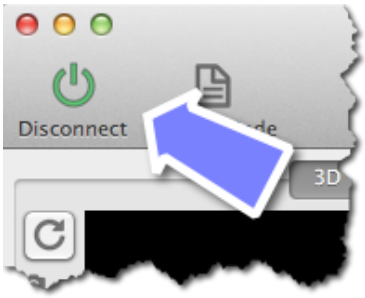
This step must happen before every print: Once you've physically placed the hot end in the front left position on the bed, type command, 'G92 X0 Y0 Z0'. This will set the printer to home position. Then type 'M114' and verify that all values for X,Y,Z are 0.



Connect to the printer

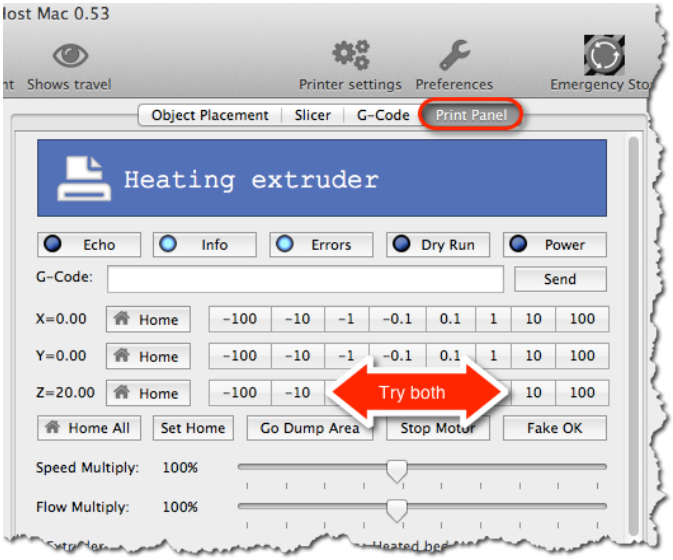
Click on the Connect button and it should turn green to indicate your are properly connected

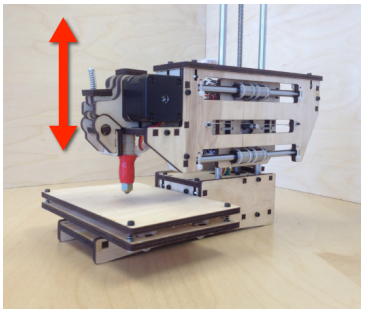




Test the Z axis

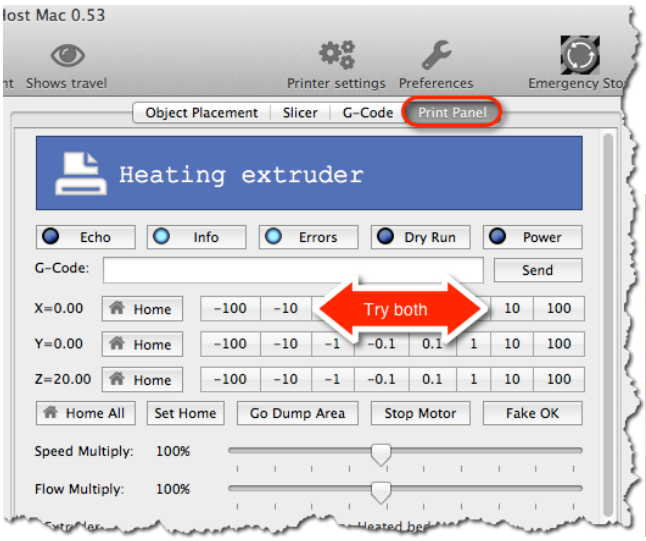
Click on both the 10 and -10 buttons for the Z axis. Your extruder should be able to move up and down freely.

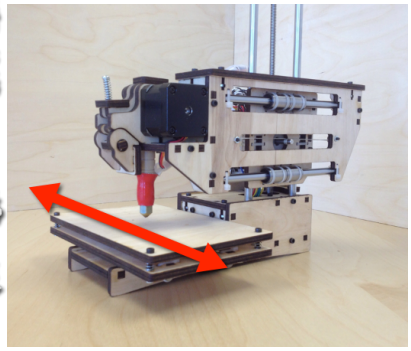




Test the X axis

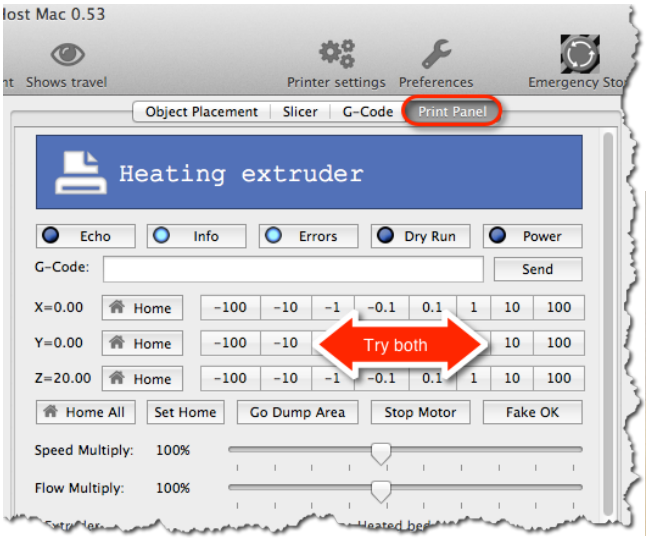
Click on both the 10 and -10 buttons for the X axis. Your extruder should be able to move left and right freely.

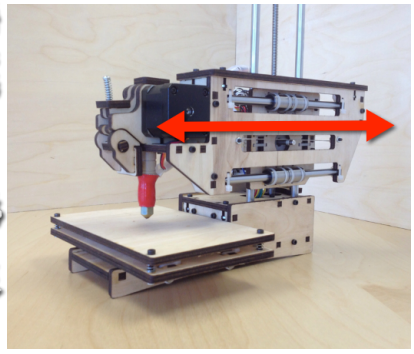




Test the Y axis

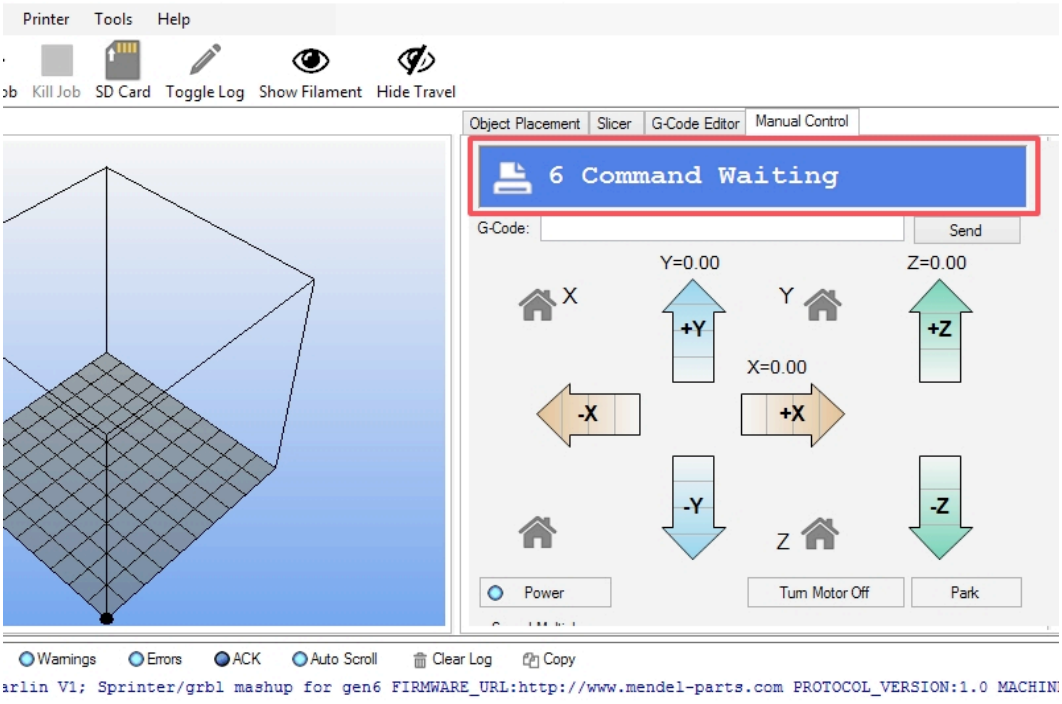
Click on both the 10 and -10 buttons for the z axis. Your printer's bed should be able to move forwards and backwards freely.

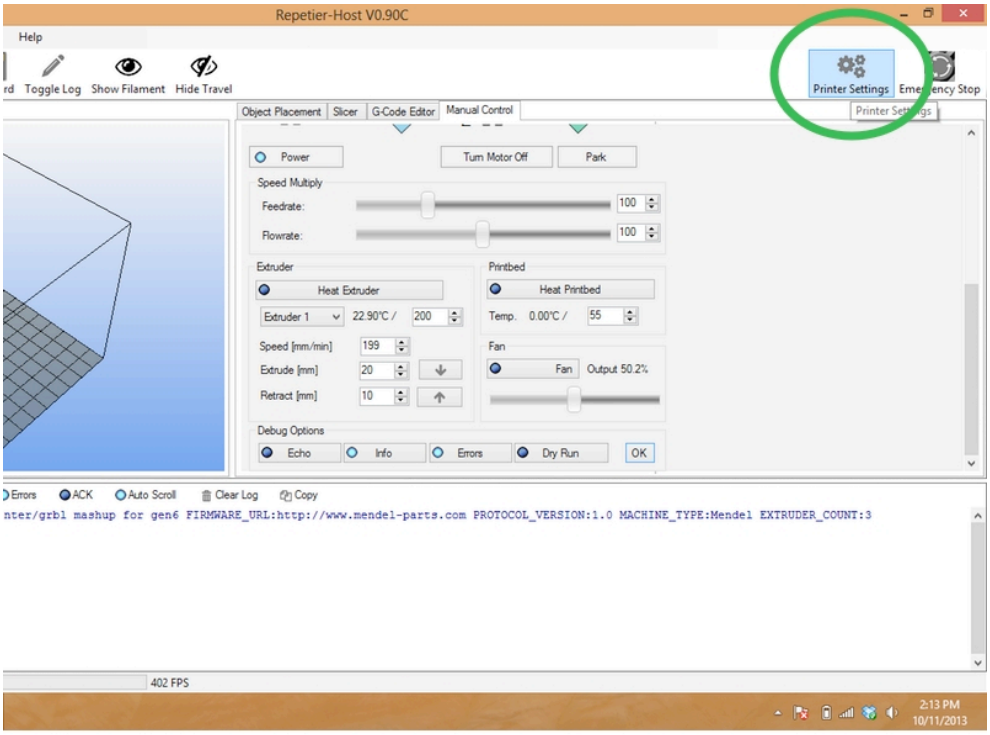


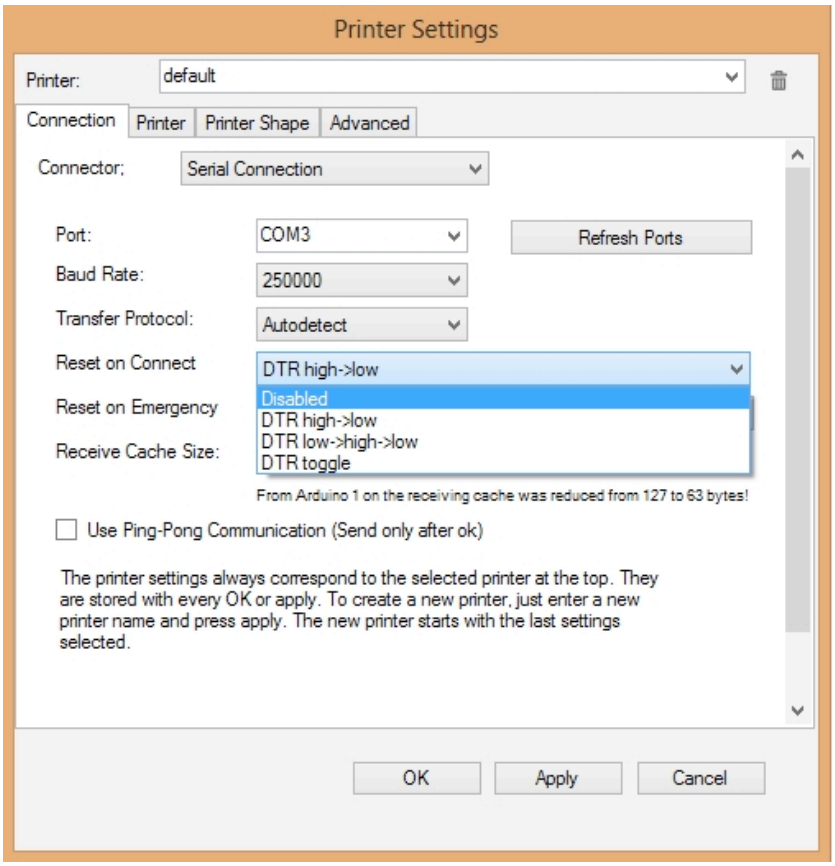


Troubleshooting tip: **Commands piling up?**

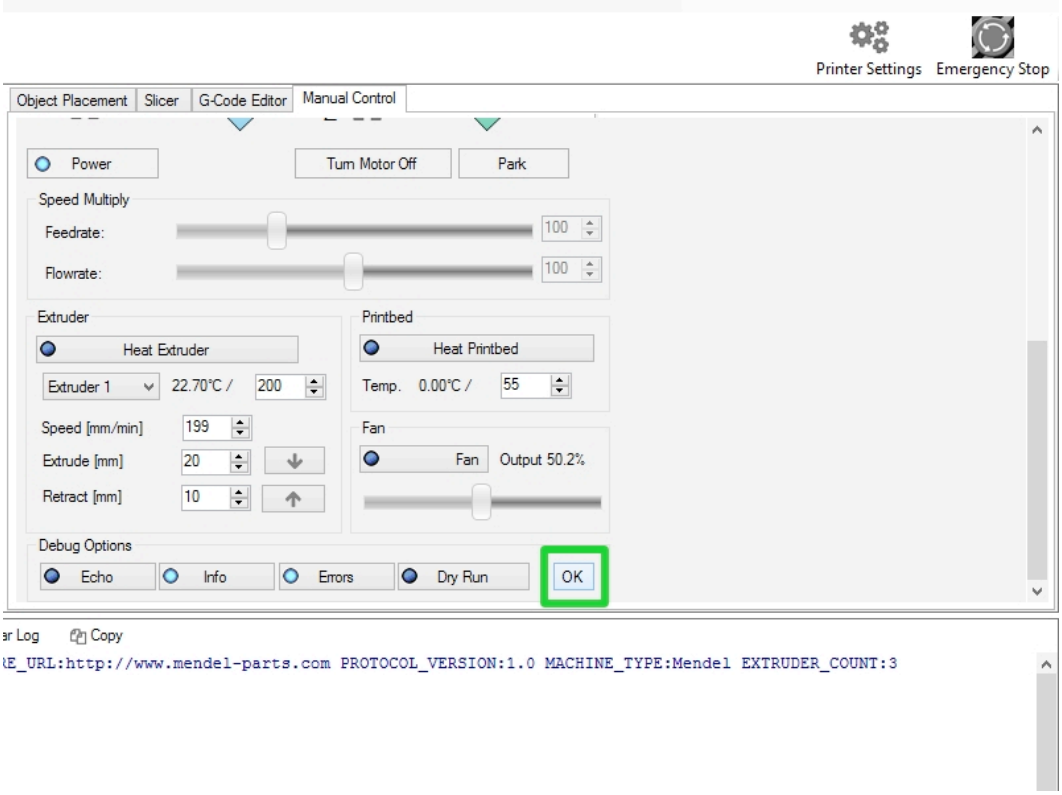
If you are using Repetier V0.90C, you will need to adjust disable “Reset on Connect” in your Printer Settings.




Repetier-Host V0.90C
Help
Toggle Log
Show Filament
Hide Travel
Printer Settings
Object Placement
Slicer
G-Code Editor
Manual Control
Power
Turn Motor Off
Park
Speed Multiply
Feedrate:
Flowrate:
Extruder
Heat Extruder
Printbed
Heat Printbed
Speed [mm/min]
Extrude [mm]
Retract [mm]
Fan
Output 50.2%
Debug Options
Echo
Info
Errors
Dry Run
OK
Errors
ACK
Auto Scroll
Clear Log
Copy
402 FPS
2:13 PM
10/11/2013

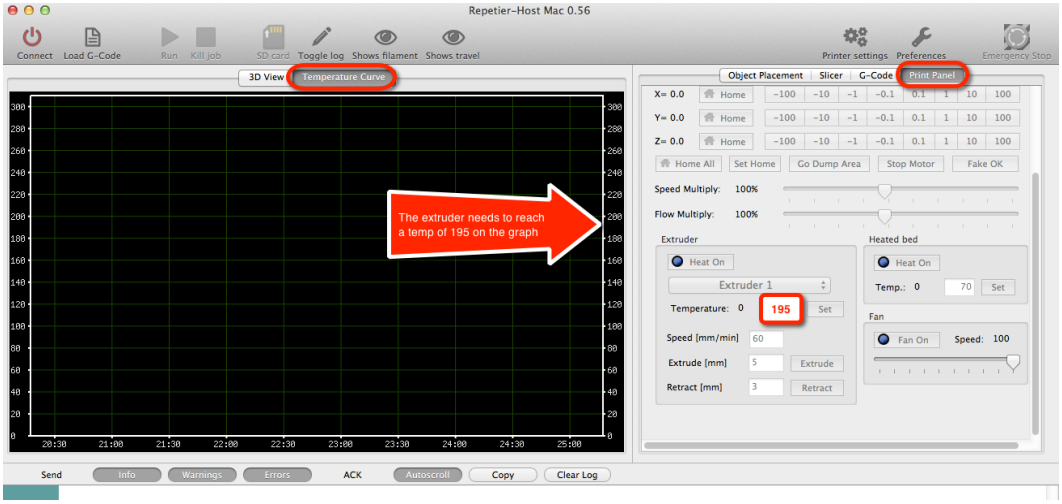

Printer Settings
Printer:
default
Connection
Printer
Printer Shape
Advanced
Connector:
Serial Connection
Port:
COM3
Refresh Ports
Baud Rate:
250000
Transfer Protocol:
Autodetect
Reset on Connect
DTR high->low
Reset on Emergency
Receive Cache Size:
Disabled
DTR high->low
DTR low->high->low
DTR toggle
From Arduino 1 on the receiving cache was reduced from 127 to 63 bytes!
Use Ping-Pong Communication (Send only after ok)
The printer settings always correspond to the selected printer at the top. They are stored with every OK or apply. To create a new printer, just enter a new printer name and press apply. The new printer starts with the last settings selected.
OK
Apply
Cancel

After you have disabled "Reset on Connect", click "OK" on the Print Panel and your commands will be executed.

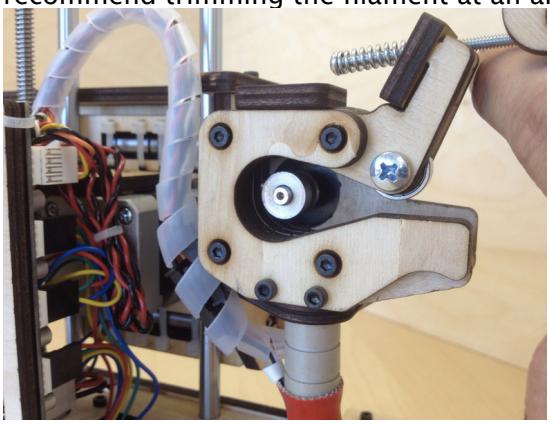


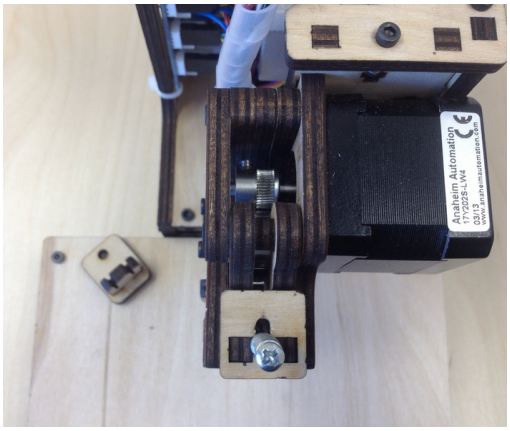
Test the extruder with initial burn-in temperature

Set your Extruder temperature to **195** and click Set. View the Temperature Curve graph and wait for the temp to reach 195 on the purple line.



When the temp has reached **195**, open the extruder latch and feed the filament down into the hot end opening at the hobbed bolt area. For easier access, we recommend trimming the filament at an angle.





Troubleshooting

If you encounter any movement issues with your axis or extruder, double check the assembly instructions and check the technical forum at www.printrbottalk.com for possible quick fix solutions. If you are unable to resolve the problem quickly, please open a support ticket.

Printing your First 3D Model

8

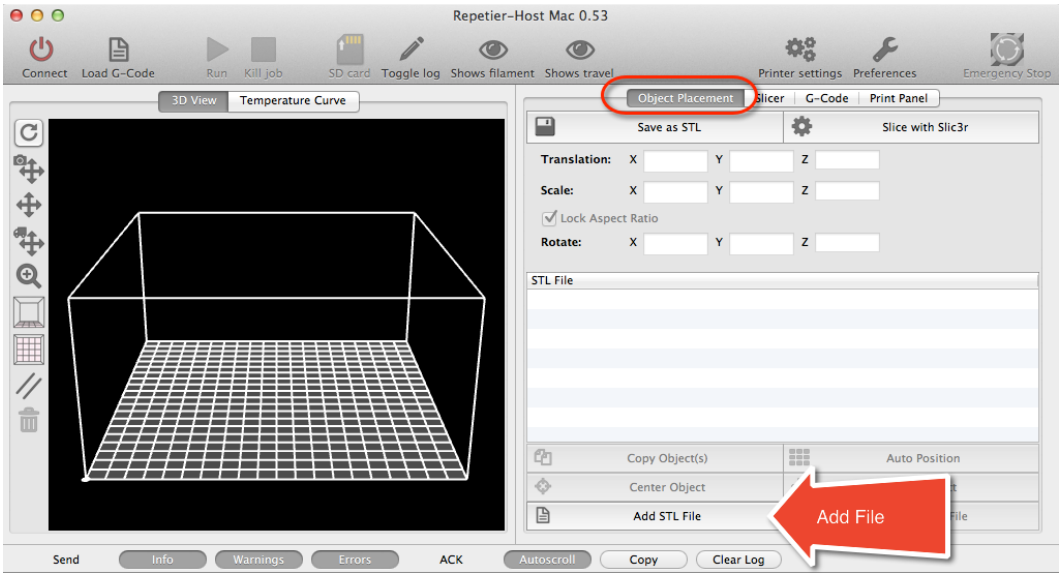
Now that your bot is assembled, connected to your computer, and the software installed – it's time to run your first test print!

Download File

Go to www.thingiverse.com and find something to print. For your first print, we recommend either the 5mm Calibration Cube or Mr. Jaws. Click on the big blue Download button and select either an individual file or the entire zip file.

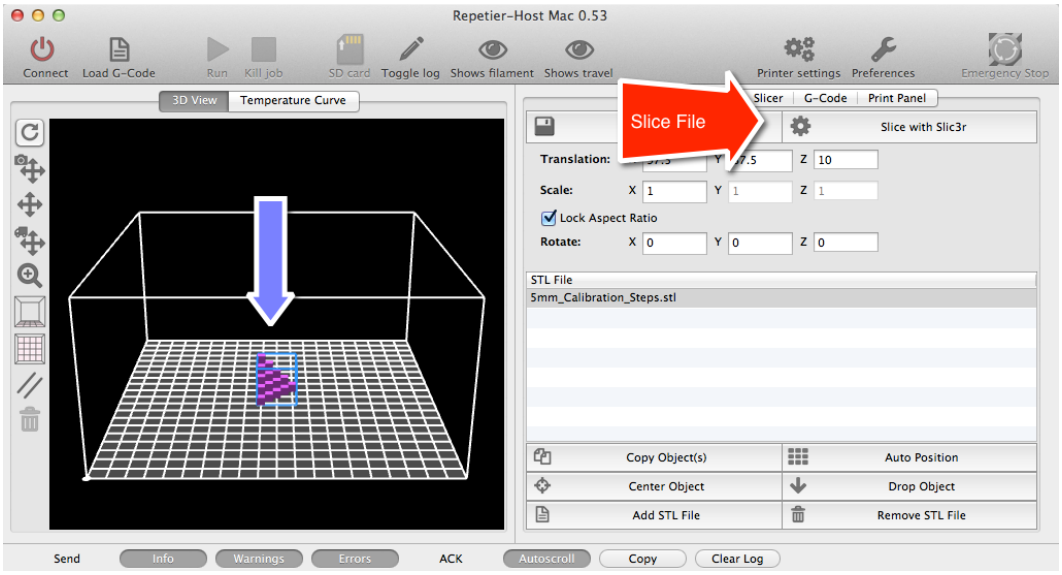
Add STL File

After you have downloaded a model, open Repetier and add the .STL file



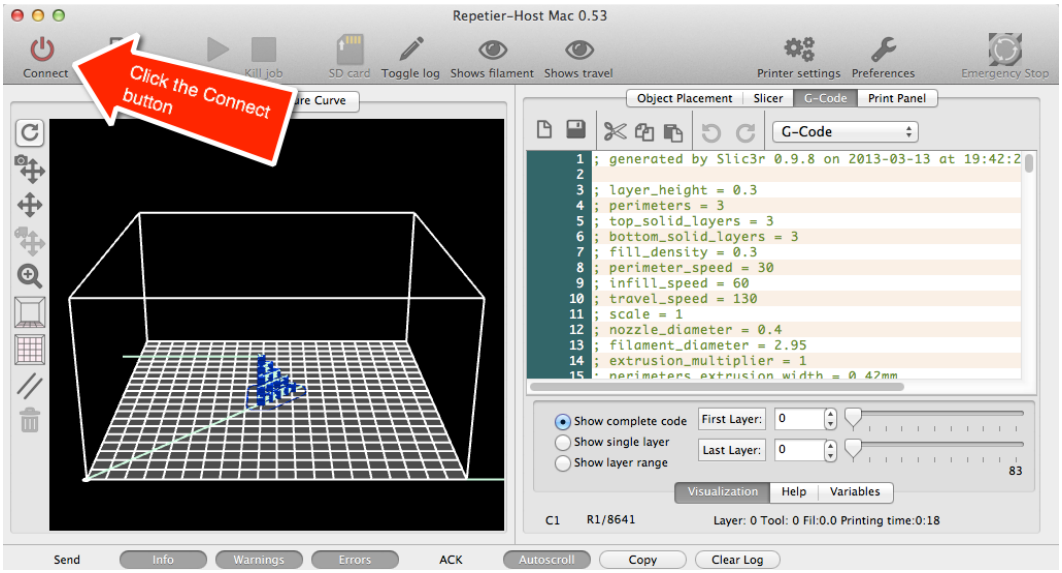
Slice

The file will drop into the print space, next click on Slice with Slic3r



Connect

When the slice has completed, click on the Connect button.



Set Temperatures

Next up , you need to set the extruder temp. Click on the Print Panel and scroll down to set the temperatures (Extruder **195 for PLA** and Heated bed **0**) be sure

to click the Set buttons. Also, please note that you should now have a green connect button to indicate that you are connected to your computer.

Temperature Curve

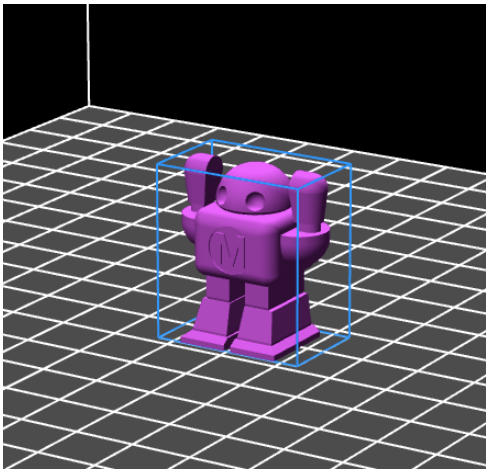
Now click on the Temperature Curve and watch the graph as the extruder and heat bed rise in temp. You need to make sure that both of them reach the purple lines before proceeding to the next step.

Run

Once the proper temperatures have been reached, you can then click on Run to start the print. Please note that it may take several seconds (or longer) before your print actually begins. Once it does start, the printer will move to home position (if it's not there already) and then begin to print. The first thing it will do is lay down a perimeter line around the area of your model and then it will start the final print.

After the print completes

The extruder and heat bed will turn off automatically. Allow several minutes to cool before you attempt to remove your print from the bed. Please note that the prints can often be firmly attached to the bed. To remove them, you can slightly twist the print when pulling up as this will help to break it free. You can also try to leverage a thin object underneath the print to pry it upward when lifting, but we recommend caution with this approach as you don't want to cut or damage the bed with a sharp object like an Exacto blade, etc.



Congratulations, you have completed your first 3D print on the Simple!

Advanced Printing

9

Now that you have done some basic prints, where do you go from here? Well, your journey has just begun as there are lots of advanced techniques for printing highly precise and complex prints; such as micro sized prints and complex objects.

Technical Forums

Please visit our technical forum at www.printrbottalk.com as we have several power users that have pushed the limits on what is possible with a 3D printer. They will be able to give you suggestions and advice on how to successfully print even the most complex objects.

Community

We also suggest checking out www.reprap.org and interacting with the rest of the 3D community. Their community portal and New Development pages have more information on how to get involved.

Common Issues and Troubleshooting 10

Filament is not extruding

The possible issue may be that either the hot end temp wasn't set to 15 for ABS or 195 for PLA and/or the filament wasn't being pulled down into the extruder. If the filament is still in the hot end, heat up the hot end to 15 for ABS or 195 for PLA degrees and then open the top latch on the extruder and pull the filament out.

Check the small grooves in the hobbed bolt where the extruder goes into the hot end. The grooves may be clogged with filament as this happens when the temp isn't hot enough and then the filament just ends up grinding against that hobbled bolt. Use an Exacto knife to clean out all of the grooves completely. Then snip off the end of the filament at an angle and push it down into the hot end with medium pressure and you should see it extrude out the bottom. Just make sure that your hot end temp has reached 215 for ABS or 195 for PLA before doing this. You can click on the temp graph in Repetier to view the current temps.

If you see filament coming out the bottom when manually pushing, pull out the filament, re-trim the tip, and then feed the filament between the two long bolts on the top latch down into the extruder hole where the hobbled bolt is. Then close the top latch to lock it into place. Make sure your computer is connected to the printer and that you are connected in Repetier. Then click on the manual extrude button to make sure it comes out.

Print not sticking to bed

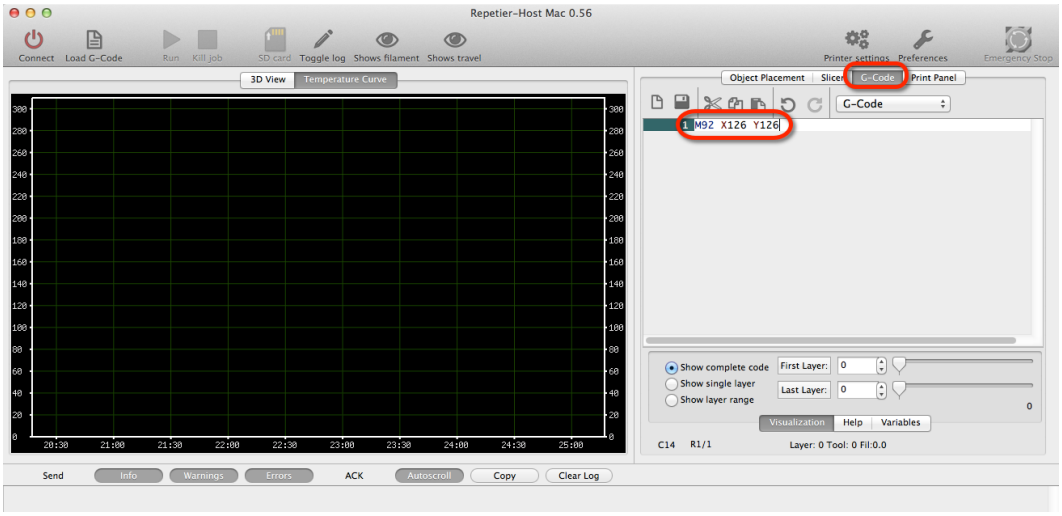
We recommend using common masking tape when printing PLA. Be sure that the tape is covering your desired printing surface. If the print is still not sticking, try lightly sanding the masking tape with a fine grit sand paper.

Head is dragging across the print

If the nozzle seems to drag through the plastic of the previous layer as it is printing then first check to be sure that the thermal wrap on the print head is secure and is not hanging below the level of the nozzle. If it is then slide it up slightly so that it is clear of the print surface. Be careful, if your heater is turned on then it will be hot! Make sure there is no plastic debris on or around the nozzle that may have been left over from a prior print -- even a small piece of plastic stuck to the side of the nozzle can ruin a print.

Print not correct size (G-Code)

Verify values for X,Y,Z by typing command M501. Values should be X119 Y119 Z2020 E100. If for some reason values are not this, change with G-code command M92 X119. Save settings on firmware with M500 command. Type M501 to verify change has been made.



Other Problems

For more in depth calibration troubleshooting, please review the Hardware Calibration and other settings in our original Getting Started Guide – Version 1 http://bit.ly/YX8nct

For all other issues, please visit our technical forum at www.printrbottalk.com for further assistance and troubleshooting. If needed, please open a support ticket for all broken and faulty parts at www.printrbot.com/support/