

Getting started with your Printrbot

Materials

To use this getting started guide to its fullest you will need the following items:

- A ruler
- Digital calipers (optional but recommended)

Prerequisites

1. Before you get started with this guide you should have fully assembled your printrbot. That means you have connected up the motors and bed, verified that everything moves smoothly and freely by hand and you have a working power supply connected and plugged in. If you haven't quite finished those steps yet then some of this guide will still be applicable but it will make more sense for you to come back once that is done.
2. This guide applies to both the printrbot and printrbot+, however for simplicity we will just refer to the machine as a printrbot.
3. You will need a flat, stable surface to work on.
4. Install Pronterface from <http://koti.kapsi.fi/~kliment/printrun/>
5. **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 Pronterface to connect to the printer.
6. Install Slic3r from <http://slic3r.org>
7. Connect to the printer with Pronterface and verify that you can move all four motors using the software buttons (X, Y, Z & extrusion).

Hardware Calibration

Bed Size

Before you begin the hardware calibration it is useful to determine the usable size of your print bed. While this is not strictly necessary (you can assume the standard printrbot has approximately 150mm x 150mm of X/Y capacity and the printrbot+ has a 200mm x 200mm capacity), depending on the model of printer, whether you have chosen to use a heated glass bed and how it is attached you may find that the workable print area varies slightly. If you plan to print objects that approach the limit of your machine's capacity then it would be worthwhile to complete this section. If you prefer to skip ahead to the **End Stops** section then you can jump to [Step 5](#) below using the standard values for X and Y and then come back and complete the additional steps later.

1. Using the X & Y controllers in Pronterface, move the print head all the way to the front left corner of the print bed. Don't use the "Home" buttons at this time. You can use the ring controls to move 0.1, 1, 10 or 100 mm at a time. Once the head is at the front left corner bring it down using the vertical Z controls until it is just above the bed. Don't worry about getting it exact right now.
2. Move the print head to the right using the **+X** control. Start with the 100 ring and then move it 10, 1 or 0.1mm increments until you reach the far right side. If you are using screws or clips to hold down glass on your PCB then make sure that the print head and other components are not hitting or binding on any component of the bed. If they are then move back to the left until there is no interference. If you hear the motor skipping or struggling you have reached the physical limit, back it off slightly. Make a note of how far you moved the head (subtracting any distance that you had to back off). This is your X size.
3. Next move the print bed forward using the **+Y** control using the same procedure as above. Again, make sure that no part of the bed, or mounting system interferes with the movement and that you are not trying to exceed the limit of travel. Make note of how far you moved the bed. This is your Y size.
4. Verify that your X size is valid at both the front and back of the bed and that your Y size is valid on both the left and right by verifying again that there is no interference with any part of the mounting mechanism. You should now have a known-safe square area within the perimeter of your print bed. This is the maximum usable size you can print.
5. Plug the X and Y size numbers into Pronterface's options screen. If you are using Slic3r then also divide each number in half and enter them into the **Print Center** option under the **Printer and Filament** tab.

End Stops

There are three end stops on the printrbot which stop the bed from moving when it reaches its "home" position. It's important to calibrate these so that the home position is correct for your machine.

1. Using the X position controller in Pronterface, move the print head all the way to the left side of the print bed (the Y & Z positions don't matter at this point). Make sure that no part

of the print head or carriage is rubbing or hitting any mounting bracket on your bed. Once you are satisfied, adjust the X stop screw so that it just engages the microswitch at this point. Test the stop by jogging the motors to the right by several mm and then using the **Home X** control in Pronterface. The X carriage should stop exactly where you expect. If it stops too early then loosen the stop screw slightly. If it stops too late or reaches the limit of travel (i.e. you hear the motors skipping or straining) then tighten the stop screw slightly. Repeat until the end stop is correctly calibrated.



*Home X
button*

2. Repeat the procedure for the Y end stop.
3. Repeat the procedure for the Z end stop. Your goal is to stop the print head just above the print surface so that you can slide a single sheet of paper between the bed and the print nozzle with only slight resistance. Use the .1mm Z controls in Pronterface to jog the head and then tighten or loosen the end stop screw as appropriate.

Leveling the print carriage

It's important that your print carriage is level relative to the bed and to the rest of the machine. Make sure that both captive nuts on the threaded rods are at equal heights and are seated fully into the corresponding wood or plastic parts.

1. Raise the Z height to approximately 20mm
2. If you apply slight pressure to either side of the X carriage there should be no vertical movement – the carriage should sit firmly against the nuts used to move it vertically. If there is any play then lift the carriage slightly and adjust one of the nuts on the threaded rod to correct. Both nuts should be at the same height so that the carriage is level relative to the rest of the machine.
3. Use Pronterface to move the bed up and down several times and then recheck.
4. This would also be a good time to lubricate the threaded rod. Repeated movement will eventually wear out the threads unless you occasionally add some lubricant. Use a light coat of spray oil containing PTFE. Be sure to cover your print bed so that no oil contaminates the print surface when you spray it.
5. Using a set of calipers or an accurate ruler, measure from the surface of the print bed to a point on the print carriage. This distance must be exactly the same on both the left and right side. Retest in several places on the print bed from front to back. If you find discrepancies then follow the next section in order to level the bed.

Leveling the bed

The bed must be level relative to the print head so that when you home your Z axis at one corner it stays the same distance from the bed throughout its range of motion.

1. Home your X, Y & Z motors so that the print head is at the front left corner of the board and is approximately the thickness of a sheet of paper above the print surface. This will be your reference height.
2. Without moving Z, move the print head to the back right corner of the bed and recheck the height.

3. Move the print head to the back left corner of the bed and recheck the height.
4. Finally, move the print head to the front right and check the height once again.
5. If any of the three corners differ from your reference height then you must level the print surface by shimming it slightly. You can place thin washers or a similar material underneath the wood print bed, between the wood and the “P” shaped mounting brackets. Recheck the bed from Step 1 and make any other adjustments until the bed is level.

Software

Calibrating your software

It is important to make sure that the motors rotate the correct number of times to move the print head an expected distance. This is often called the “steps per mm” value and can vary from printer to printer. The settings can either be burned into the firmware in order to provide a default value or they can be overridden at print time. For now you will want to enter some standard values and then evaluate their correctness by making some calibration prints. Examples of calibrating your steps per mm are given in the **Troubleshooting** section below.

Use Pronterface to send the following commands to the printer:

- M92 X64.8592 ; calibrate X
- M92 Y64.8592 ; calibrate Y
- M92 Z2387.0719 ; calibrate Z
- M92 E569 ; calibrate E

These lines should be placed into the “**Start GCODE**” section of Slic3r as noted below.

The X, Y, and Z calibration settings given above should be close enough to get you started, however your E value can vary widely depending on your filament type & color, how tight your extrusion idler is, etc. You will need to recalibrate it to before you begin, and then again any time you change your filament type. To correct the E calibration:

1. Load filament into the extruder and bring it up to temperature. Extrude a few mm to make sure that the filament is moving freely and that the extruder is primed.
2. Mark the filament with a piece of tape or other mark several inches above the point where it enters the machine.
3. As accurately as possible (preferably using digital calipers), measure the distance from a fixed point on the extruder to your mark.
4. Extrude 30mm of filament using Pronterface, 5mm at a time (click the extrude button 6 times slowly)
5. Measure the distance to the tape again from the same point on the extruder. If it does not equal 30mm then do the following calculation:

$(\text{E steps per mm} * 30) / \text{actual extrusion length} = \text{new E step value}$

For example, if you are using the initial E step value of 569 and you actually extruded 27mm of filament in the test:

$(569 * 30) / 27 = 632$ (rounded to closest whole value)

Use the result as your new E steps per mm value in your **Start GCODE**. This value will be different for different materials, in particular if you switch from PLA to ABS or vice versa you will need to recalibrate this value. Sometimes different colors or batches of the same material may require different settings, so make sure to become comfortable with this procedure. You will likely want to perform this calibration step each time you change materials.

Initial Slic3r settings

Slic3r will save its settings in a file called config.ini by default, although you can choose to name this anything you like. You may wish to have different configuration files for different filament types or quality settings. Choose a filename and path for your config file (choose an existing directory, we will be creating the file in the next step) and verify that Pronterface is configured to use it by default:

1. Open Pronterface and go to the Settings -> Options menu
2. You will be modifying the slicecommand and sliceoptscommand options to replace **--load config.INI** with **--load /full/path/to/your/INI_file**
For example, on Windows it might be something like: C:\Documents\slic3r.ini
On Mac OS it might be: /Users/myuser/slic3r.ini
3. Close the Options window
4. Choose **Slicing Settings** from the Settings menu in Pronterface to continue configuring Slic3r

Slic3r has many configuration options which you can tweak, the minimum that we recommend depend on your print material. The printbot ships with a spool of ABS but you may also choose to print with PLA. These settings are by no means optimized but should provide an acceptable and conservative base from which to start.

ABS

- Bed temperature: 70
- Hot end temperature: 210

PLA

- Bed temperature: 60
- Hot end temperature: 185

Common settings

- Nozzle diameter: 0.5

- Filament size: measure with a set of calipers if possible. Enter 3.0 if you are not sure
- Layer height: 0.4
- Skirt loops: 2
- Perimeters: 2
- Fill density: 0.4

Speed settings:

- Perimeters: 65
- Small Perimeters: 30
- Infill: 80
- Solid infill: 70
- Bridges: 60
- Travel: 195
- Bottom layer speed ratio: 0.35

Add the appropriate calibration commands from the previous section to the **Start GCODE** section under the **Start/End GCODE** tab in Slic3r and then save your config file. If you configured Pronterface to use Slic3r then this config file should be saved into your default location, although over time you may want to develop different configurations specific to different materials and prints.

Your First Print

Choose an easy print for your first object. There are thousands to choose from on [thingiverse.com](http://www.thingiverse.com) or you can design your own. For now, avoid any objects with large overhangs or any bridging (free air extrusion). A good first example is Mr. Jaws v2 available from <http://www.thingiverse.com/thing:14702>. It is a relatively quick print and the finished product is useful for keeping your potato chips fresh.

Turn Up The Heat

Use Pronterface to set the bed and nozzle temperatures to their initial values and allow them time to heat up. If you are using glass on your bed then you should allow at least 5 to 10 minutes for it to get up to temperature. Even when the printer reports that the heat bed is ready it may take several more minutes for the glass to fully heat. Remember that the temperature is being measured at the red PCB, not at the print level.

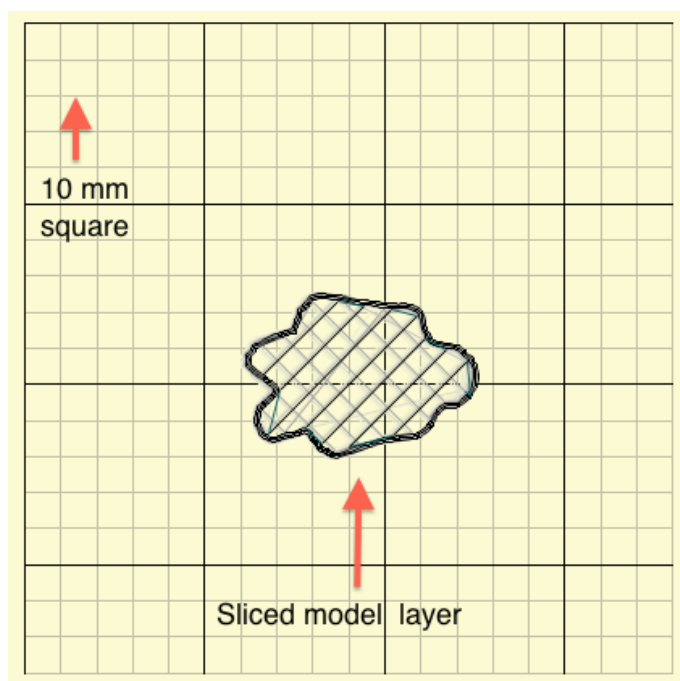
When the hot end is up to temperature move the head several centimeters off the bed and try extruding some material in 5mm increments using the **Extrude** button. Make sure that the filament is moving freely and that it extrudes uniformly.

Prepare The Bed

If you are printing with PLA the plastic will stick very well to the plain glass bed. Wipe it down with some rubbing alcohol and a paper towel to remove any dirt or fingerprints. If you are printing with ABS then you will most likely find that your prints do not stick to plain glass. Place a layer of either blue painters tape or kapton tape (preferred) carefully across the bed. Wider rolls are preferred so that there are fewer seams. Wipe down the surface of the tape with rubbing alcohol as above.

Slice And Print

Use the **Load File** button in Pronterface to load an STL. If you have configured Slic3r as noted earlier then the model will be automatically sliced and loaded into the yellow print area. You can click on the preview and then spin the mouse wheel while holding Shift in order to see the layers as they will be printed. The size of the final object is also indicated via the grid pattern. Each small square is 10mm wide.



Once you are satisfied with the proposed print, click the **Print** button and the printer should begin after a few moments. Congratulations, you've just made your first print!

Troubleshooting

Prints not sticking to the bed

Make sure that your bed temperature is set to the right value. Make sure that the bed is clean of any fingerprints or other material. Use rubbing alcohol to clean it thoroughly. Make sure the head is not dragging the print. Make sure that your Z home height is approximately the thickness of a sheet of paper above the print surface.

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.

If you have checked for physical obstructions then either the Z or E steps per mm are probably incorrect. To correct the E calibration see the steps in the **Calibrating your software** section above. The procedure to correct the Z calibration is the same except you will use the initial Z steps value and instead of extruding filament you will move the Z axis by 30mm. Accurately measure the distance between the bed and a fixed point on the print head and calculate:

$$(Z \text{ steps per mm} * 30) / \text{actual distance}$$

When I home the Z axis the threaded rod just spins forever and bottoms out the captive nuts

Your Z end stop is not correctly calibrated, see the **Hardware Calibration** section.

My machine makes a lot of squeaking or rattling noises

Check to see that everything is lined up and perpendicular. With the motors off you should be able to easily move X and Y by hand. If not then determine which component is binding up or getting stuck. Apply a light coat of oil containing PTFE to the threaded rod.

My objects are coming out too small

Use a calibration print to check which axis is incorrect and recalibrate the steps per mm value in your **Start GCODE** as explained above.

Filament is not extruding consistently

Check to see that the hobbled bolt is clear of any debris or ground up plastic. Make sure that the filament idler screws are tight – for PLA they should be tightened down almost all the way. For ABS back them off a little bit. Make sure that the action of feeding in filament does not cause it to be dragged sideways against the hobbled bolt as this can cause it to run out of alignment. Consider printing a filament guide such as the one at <http://www.thingiverse.com/thing:10735> to fix this case. Try a slightly hotter or colder extrusion temperature. You may have to experiment to find the ideal temperature for each type & color of material that you use. Adjust the temperature in 5 degree increments until you can extrude a line of plastic on the bed cleanly. There are some calibration objects available on thingiverse which can help with this (you can adjust the temperature on the fly using Pronterface while the printer is printing.)