

# MAKERBOT

For use with MakerBot Replicator Desktop 3D Printer (Fifth Generation Model)

---



# Table of Contents

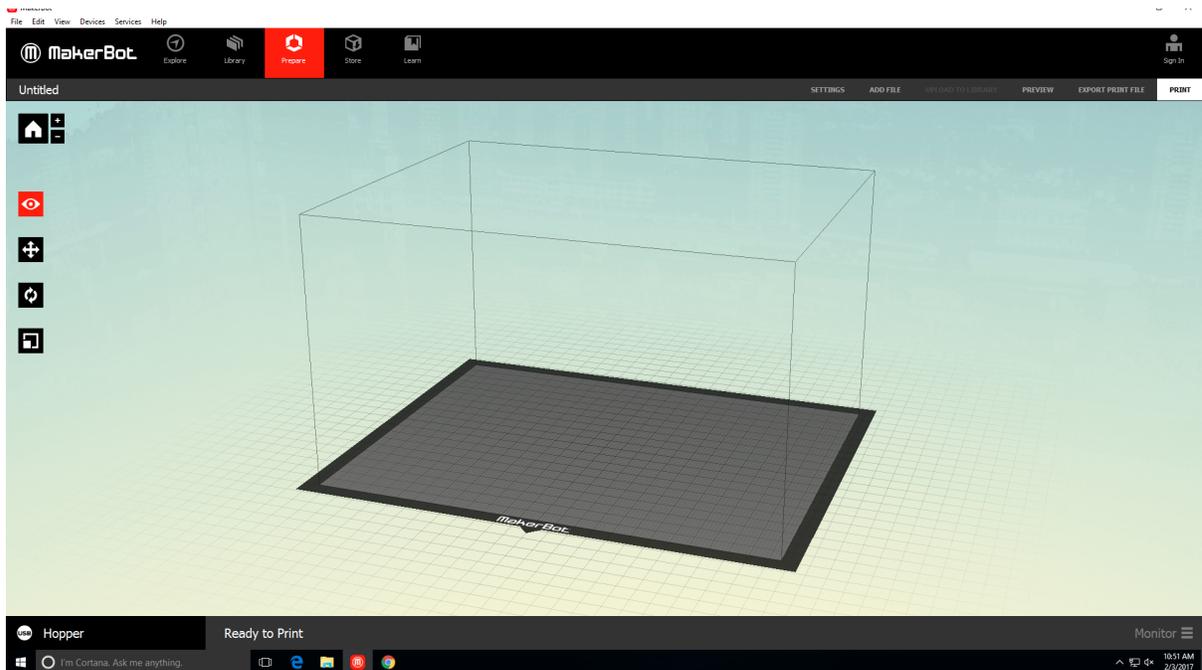
---

p. 3-5	.... Using the MakerBot Desktop
p. 6	.... Using the Right Settings
p. 7	.... Printing
p. 8	.... Troubleshooting

# Using the MakerBot Desktop

1. Obtain .STL file or save model as an .stl file from a CAD program.
2. Open up Makerbot Desktop. If this is your first time using the software, go to the **Windows Logo ->All Apps ->Scroll down to "M" and you should find the app called "MakerBot Desktop"**.  
It is recommended that you create a shortcut on the desktop or pin it to the task-bar.

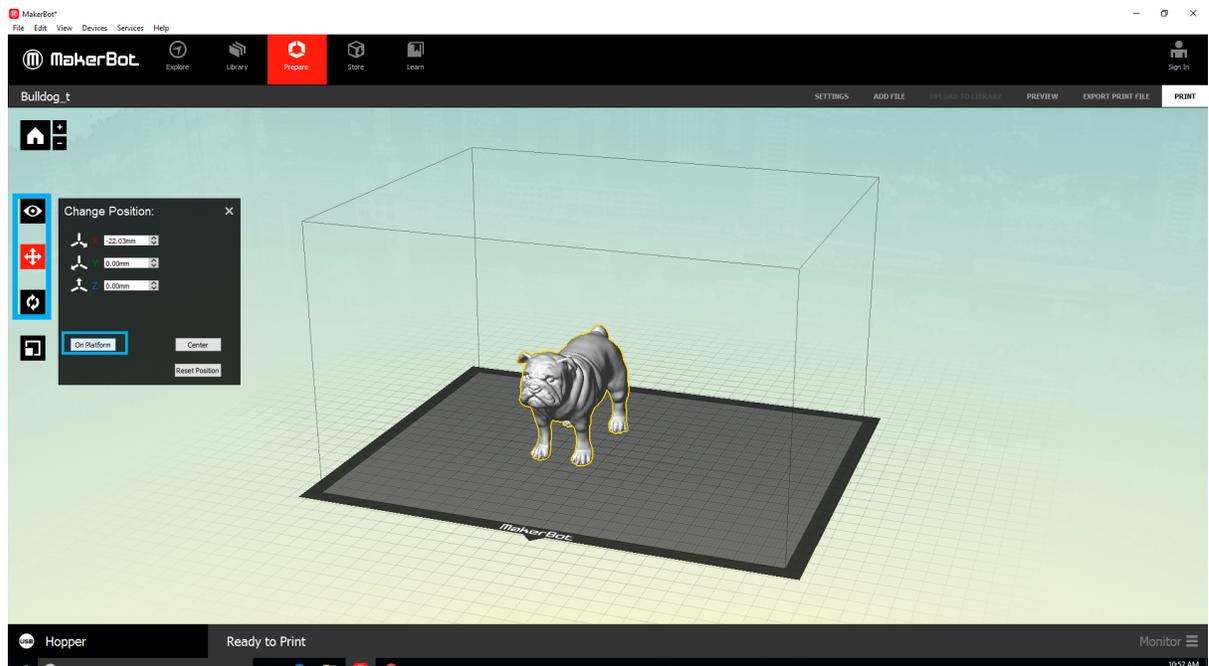
Your MakerBot Desktop should look like this:



3. Open the .stl file by either using **File -> Open** command, or by dragging the file onto the platform.

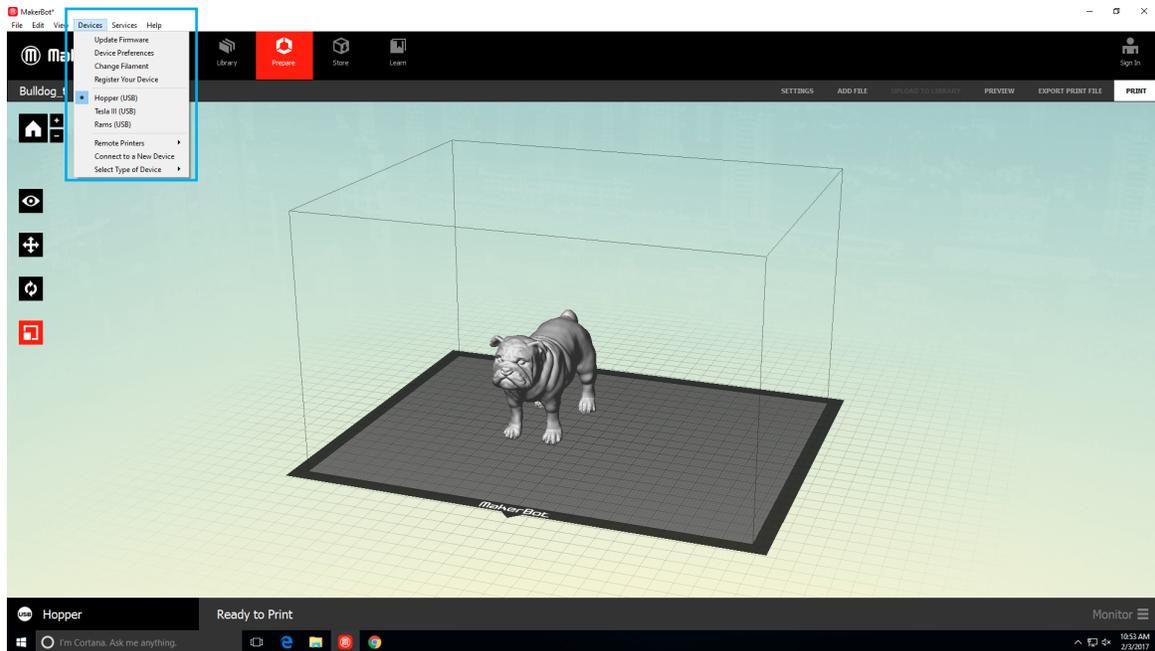
4. Once your object is on the virtual platform, select the object and make sure that it is correctly oriented for printing by using the **Move, Turn, and Scale** dialogs (located on the left of the desktop).

5. When you are in the Move dialog, make sure to hit the **On Platform** button.



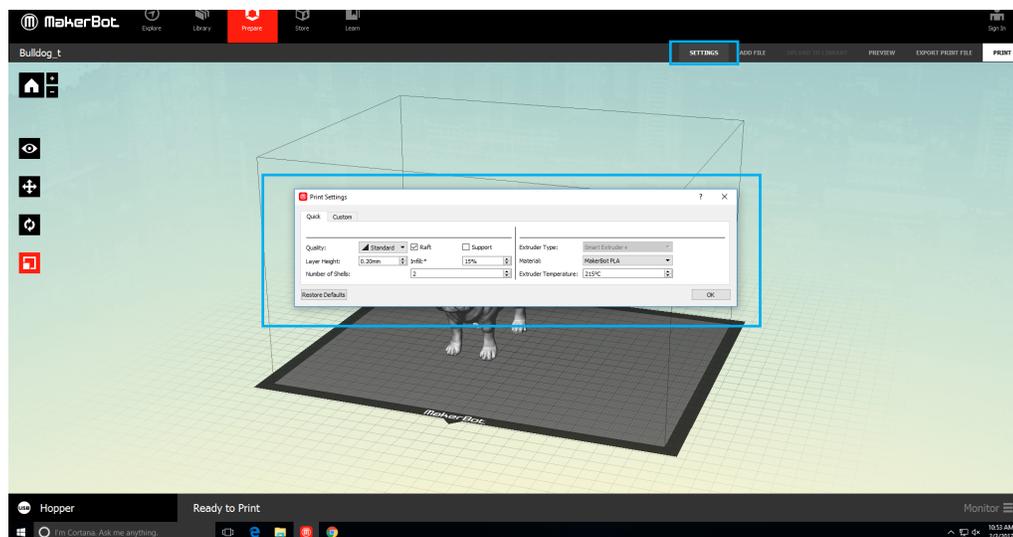
6. If you are printing multiple objects and they are different, you can repeat steps 1-5 in the same window. If you want multiple copies of the same object, use the copy and paste commands (ctrl + C, ctrl V) (after scaling/adjusting your model).

7. Select which machine you are printing to by clicking on Devices.



\*At this point, make sure that the plate of the printer is cleared. Try to place your object on a part of the build plate which has tape that is nice and flat.

8 Once the object(s) on the platform is ready to print, go into SETTINGS, located at top right.



# Using the Right Settings

---

**1. ALWAYS MAKE SURE TO HAVE “RAFT” CHECKED ON!** This will make sure that your piece has a solid foundation to print on and won't peel off the base.

**2. Check Support if the object has large overhangs or needs support materials.** Typically, any overhangs that are 45° or lower need support.

**3. Infill is the percentage of the model that will be filled in. 0 will only produce the outer shells, while 100 produces a solid object.** Any number in between will result in a honeycomb lattice in parts of the model that are designed to be solid. The infill percentage changes the density of the lattice. We recommend that you keep the infill at minimum of 10%

**4. Number of Shells refers to the number of surface layers added to the inside of the perimeter.** This is to strengthen the outside shell of the model, particularly with a lower infill percentage and improves the finish of the model. Usually 2 shells gives you a good finish. Increase the number to 4 or 5 if you plan on sanding the piece afterwards.

**5. Layer height corresponds to how tall each layer is during one pass of the extruder head.** The larger number means a faster but lower resolution print. Do not go below 0.1mm in layer height.

# Printing

---

6. Before you print, press the PREVIEW button to find out how long your print will take. To ensure that the machines are open to others, please keep the print time to three hours or less. **If you are unable to do so, please talk to a staff member.**
  
7. To choose which device to print to, navigate to Devices -> then choose which printer you would like to print to between Tesla, Rams, and Hopper.
  
8. Click the PRINT button to send it to the printer.
  
9. Note down the estimated time on the Log Sheet, along with starting time, name, and description.
  
10. Once the machine has gone through its starting sequences, **make sure the filament extrudes smoothly and at least two layers of the print are completed before you leave (around 15 minutes).** You can log off the computer once you have sent the file to print.
  
11. Once the object has finished printing, go ahead and try to pry it from the plate. If the model sticks too well to the plate, use the scraper to pry the model off. Smooth the tape down afterwards.

# Troubleshooting

---

*There are gaps on the outside surface of the print/print quality is poor :* Increase number of shells

***Filament Jammed Error:*** If it looks like nothing has been extruding and it is the first layer of the job, cancel the job, re-level the plate (instructions below), and send the job to the printer again.

If it is currently in the middle of a job, unload and reload the filament following the instructions on the screen closely. DO NOT CANCEL YOUR PRINT, you can reload the filament and continue your job.

***How to re-level the build plate:*** On the printer LED screen, under the Main Menu, go to the SETTINGS tab, scroll down to CALIBRATION -> LEVEL BUILD PLATE -> START ASSISTED LEVELING. You will then be prompted to unload the filament and turn knobs underneath the build plate.

***How to change filament:*** On the printer LED screen, under the Main Menu, go to the FILAMENT tab and go through the prompts for each option.

As always, if you feel uncomfortable or unsure, get help from a staff member!