

Monoprice Maker Ultimate 3D Printer Workflow

Computation + Construction Lab | Iowa State University

Monoprice Maker Ultimate

Build Volume: 7.87" x 7.87" x 6.89"



Hardware

After using these tools please make sure to return them to the grey container labeled Maker Ultimate.

1. Unclogging Tool
2. 3 Allen Wrenches
3. Card Reader
4. 1 MicroSD Card with adapter
5. SD Card Holder
6. Scissors
7. USB A to USB B cable
8. Scraper
9. Needle Nose Pliers
10. Tweezers



Software



RhinoCeros



**AUTODESK®
NETFABB®**



Design Setup

- RhinoCeros (make sure to export in mm) or whatever you feel comfortable using. A range of softwares can be used to setup an STL File.
- You may also choose to fix your STL or add supports in a variety of programs such as; Meshmixer, Netfabb, etc. These programs can be downloaded for no cost online.

Cura

- Cura version 15.04.6 for either PC or Mac (Free download online).

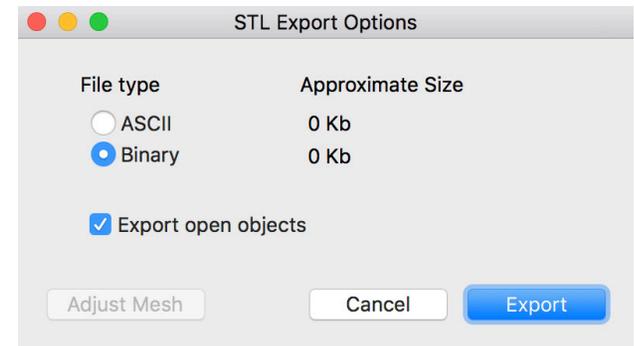
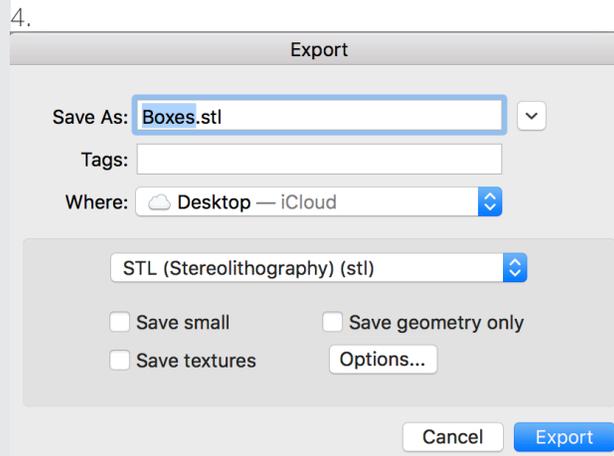
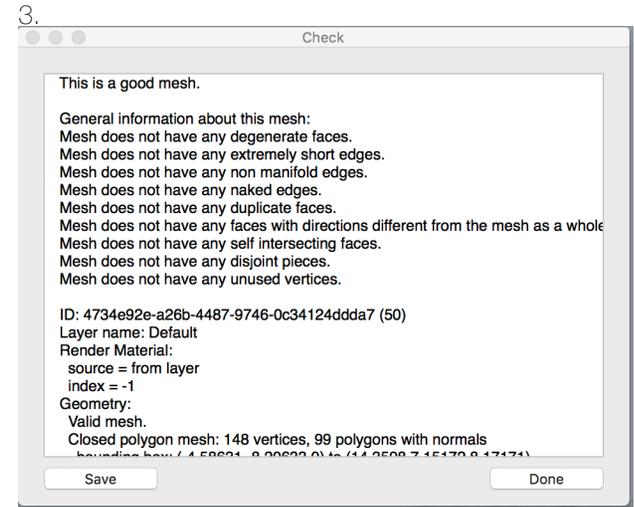
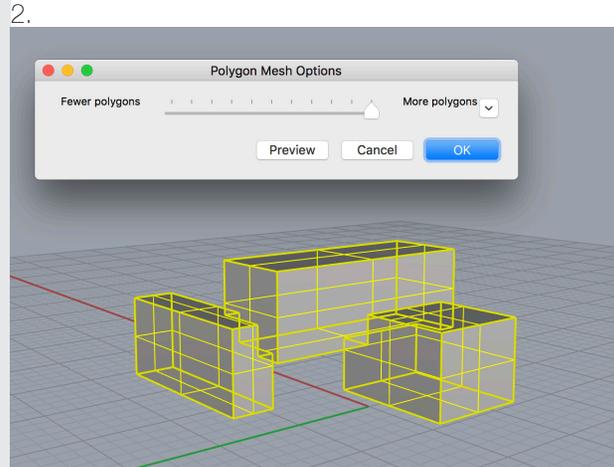
Printing

- Printing with an SD card. Put SD into computer > Select file > Save Gcode in Cura and choose the SD card location to save your Gcode to your SD card. Insert SD Card into the Graphical LCD Controller.
- Print directly from a computer using Cura and a USB A to USB B cable.

Creating a Mesh in Rhinoceros

1. BooleanUnion your model. Then select your 3D model.
2. Type mesh to convert your polysurface.
 - Choose more polygons
 - Tip save a copy of your polysurface.
3. Select your mesh, type check.
 - If your mesh is bad, type unifymeshnormals
4. Change your units to millimeters! Export the mesh as a binary .stl file.

Tip:
This example model was made quickly and was just an example of a model that would need support material. If your model is super basic like this consider hand cutting or using a laser cutter.



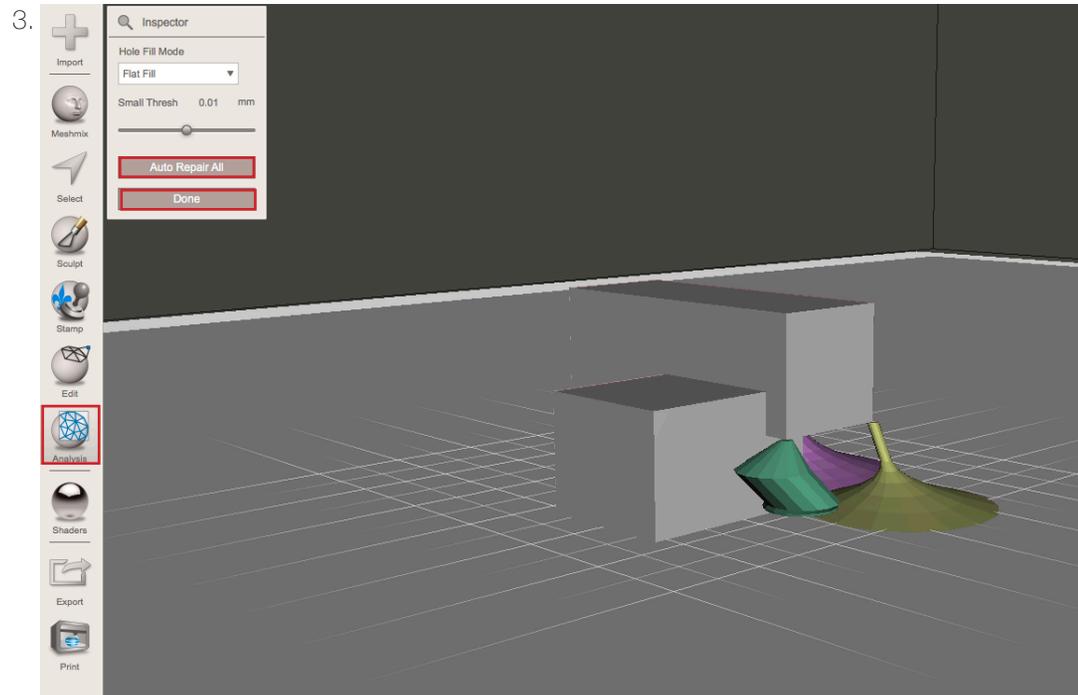
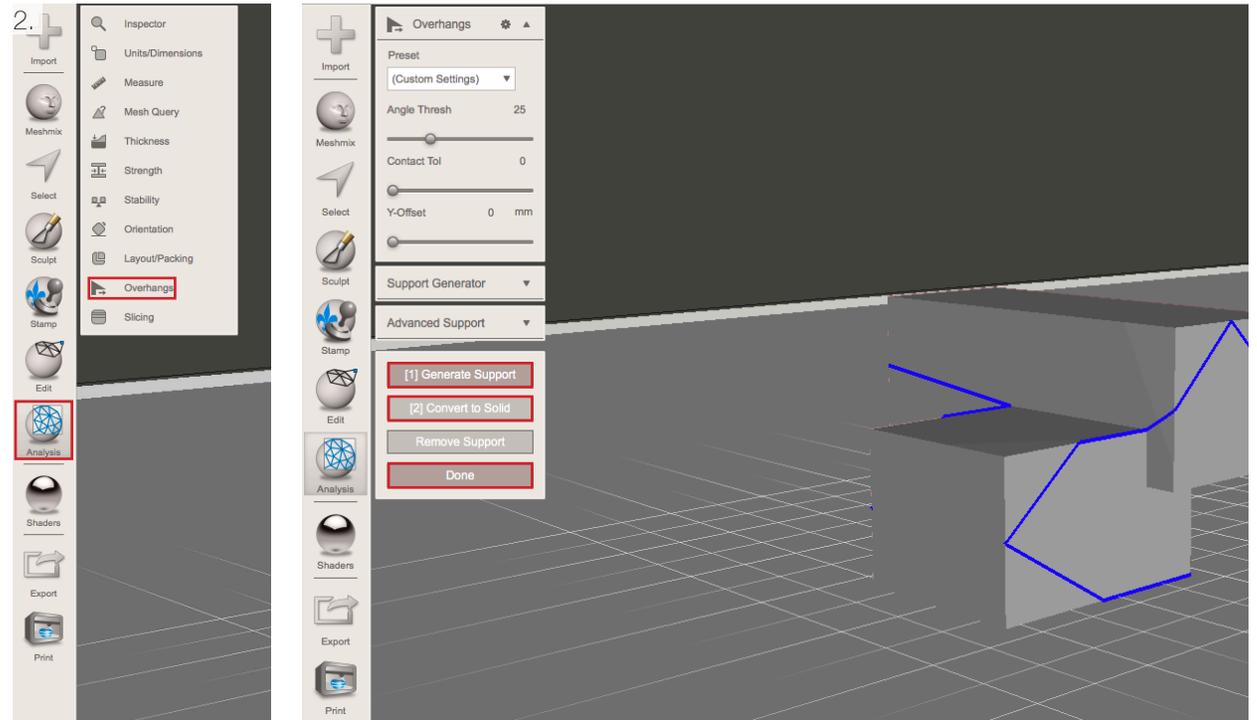
Meshmixer

Adding Supports + Fixing your .stl

1. Import your model to Meshmixer (free download, autodesk software). If you would prefer to use a different program such as Netfabb feel free. This is just a super basic way of adding suport material and fixing your .stl for print. There are many online resources that could teach you much more about this.
2. If your model has cantilevers/large overhangs or bridges,W Click Anaylsis> Overhangs > [1] Generate Support > [2] Convert to Solid > Done.
3. Then click Analysis > Auto Repair All > Done.
4. Export your model as a binary stl.

Tip:

This example model was made quickly and was just an example of a model that would need support material. If your model is super basic like this consider hand cutting or using a laser cutter.



Setting up Cura for Maker Ultimate

1. Download Cura version 15.04.6 for either PC or Mac (Free online).
2. Once installed find Cura on your computer and open it.
3. Check your Basic settings. Make sure that they match the settings in the figure.
4. Check your Advanced settings. Make sure that they match the settings in the figure.
5. Press Machine > Machine Settings. Make sure that they match the settings in the figure.

3. **Basic** Advanced Plugins Start/End-GCode

Quality

Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>

Fill

Bottom/Top thickness (mm)	0.6
Fill Density (%)	20

Speed and Temperature

Print speed (mm/s)	50
Printing temperature (C)	210
Bed temperature (C)	50

Support

Support type	None
Platform adhesion type	Raft

Filament

Diameter (mm)	1.75
Flow (%)	100.0

Machine

Nozzle size (mm)	0.4
------------------	-----

4. Basic **Advanced** Plugins Start/End-GCode

Retraction

Speed (mm/s)	40.0
Distance (mm)	7

Quality

Initial layer thickness (mm)	0.3
Initial layer line width (%)	100
Cut off object bottom (mm)	0.0
Dual extrusion overlap (mm)	0.15

Speed

Travel speed (mm/s)	80
Bottom layer speed (mm/s)	25
Infill speed (mm/s)	50
Top/bottom speed (mm/s)	15
Outer shell speed (mm/s)	15
Inner shell speed (mm/s)	30

Cool

Minimal layer time (sec)	5
Enable cooling fan	<input checked="" type="checkbox"/>

5. **Machine settings**

E-Steps per 1mm filament	0
Maximum width (mm)	200
Maximum depth (mm)	200
Maximum height (mm)	175
Extruder count	1
Heated bed	<input checked="" type="checkbox"/>
Machine center 0,0	<input type="checkbox"/>
Build area shape	Square
GCode Flavor	RepRap (Marlin/Sprinter)

Printer head size

Head size towards X min (mm)	0
Head size towards Y min (mm)	0
Head size towards X max (mm)	0
Head size towards Y max (mm)	0
Printer gantry height (mm)	0

Communication settings

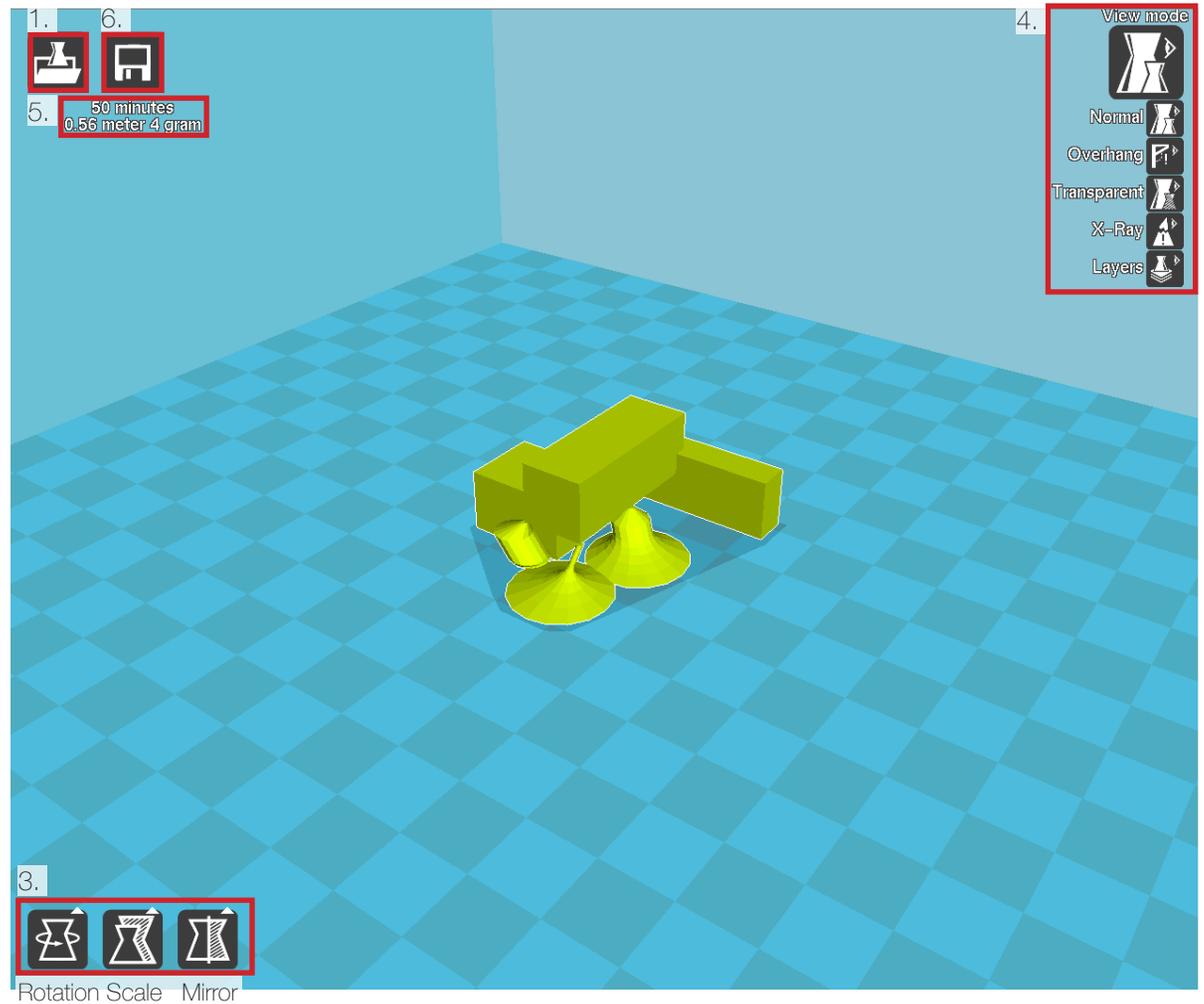
Serial port	AUTO
Baudrate	AUTO

Setting up your G-Code in Cura

1. Open Cura. Import/Load the .stl file you want to print.
2. Platform Size Restrictions
-If an object is too big or off the platform it will turn grey. Try to move or scale the object until they fit using the following commands in the Manipulating Objects section.
3. Manipulating Objects
-Click on object. Then three squares will appear in the bottom left corner which will allow you to rotate, scale, and mirror your object. Click any of the three squares to get more options and to use the command.
4. In the upper right corner is a box that reads view mode. If you click on this you will get a variety of viewing options.
5. A rough time and filament amount estimate is given in the left-hand corner. It is not always very accurate.
6. Save your G-Code on the Maker Ultimate's SD card by clicking the floppy disk icon.

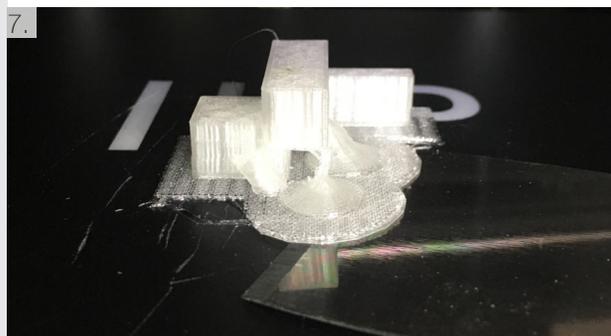
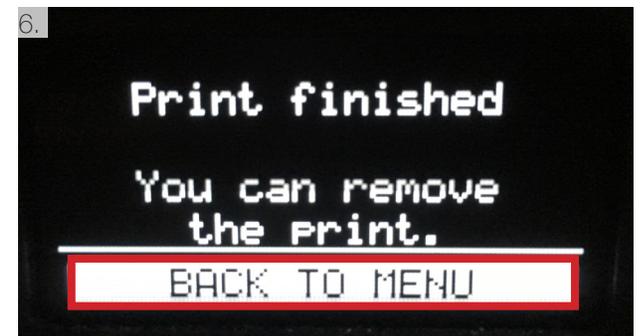
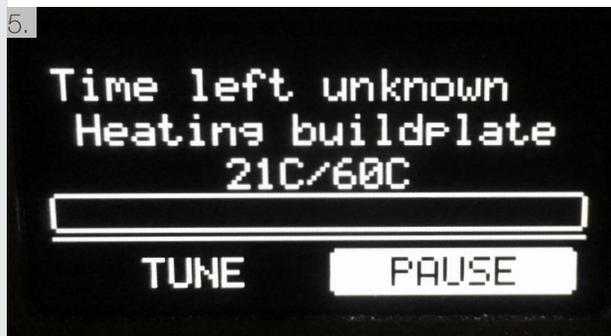
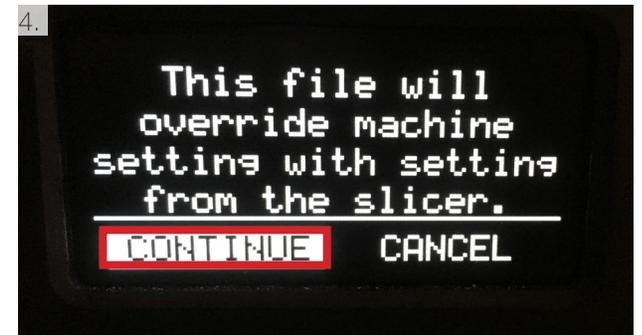
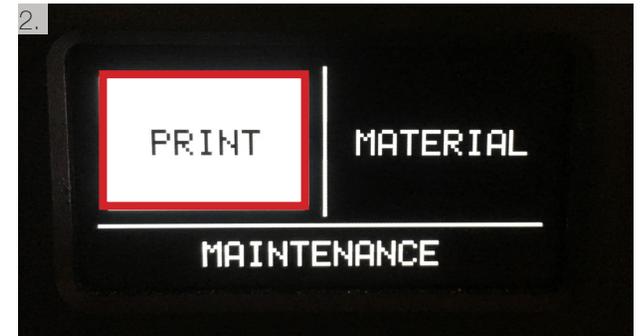
Tip:

This example model was made quickly and was just an example of a model that would need support material. If your model is super basic like this consider hand cutting or using a laser cutter.



Printing via SD Card

1. Put the SD card back into the printer. Located to the left of the screen.
2. Use the control knob to the right of the screen to highlight and select PRINT.
3. Locate and select your file on SD card. In the accompanying images, the file is Boxes, but instead select the file you created.
4. If the prompt shown in photo 4 appears, select continue.
5. While it is printing the model the printer will display a progress bar (the time is not always accurate) as well as a TUNE and a PAUSE option. Select TUNE > ABORT to cancel the print operation.
6. When the print operation is completed, the printer will start to cool down. A progress bar will help you predict the amount of time needed for cool down. Once the screen reads Print finished you can remove your print. Select BACK TO MENU.
7. You can now remove the print from the build plate. The best way to do so is with the included scraper.
8. Remove any additional material such as rafts or support material with the provided needle nose pliers. Make sure that all small equipment designated to this machine is returned to the grey container labeled Maker Ultimate.

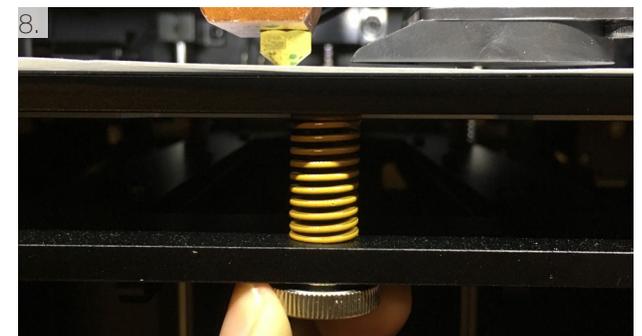
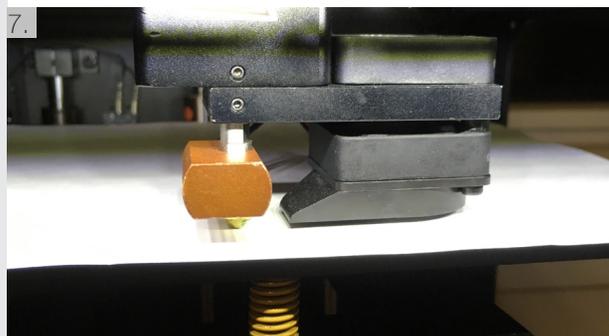
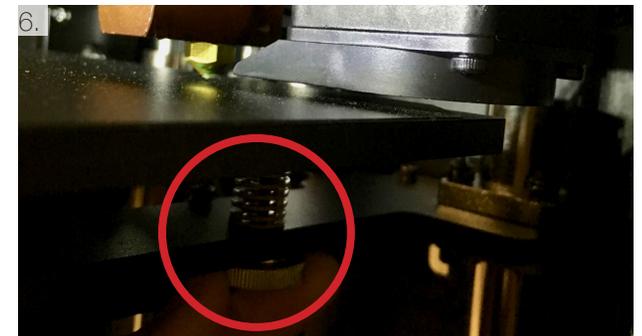
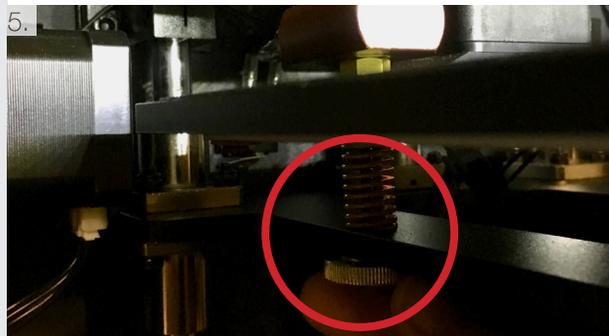
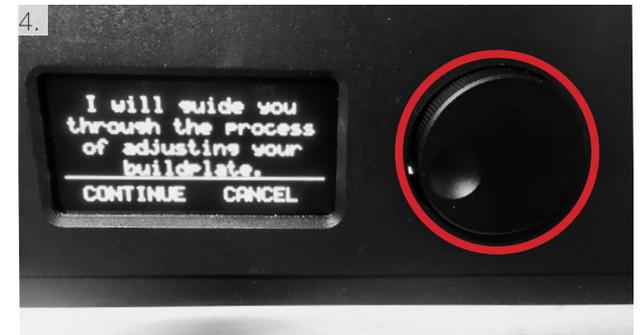
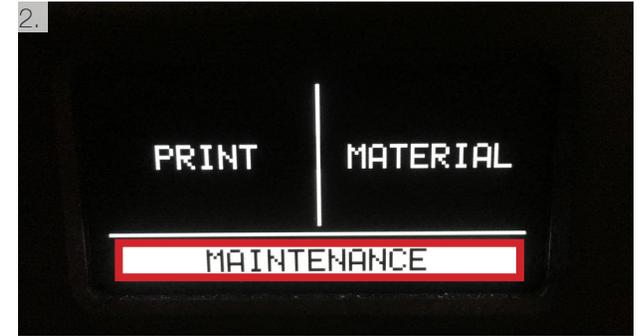
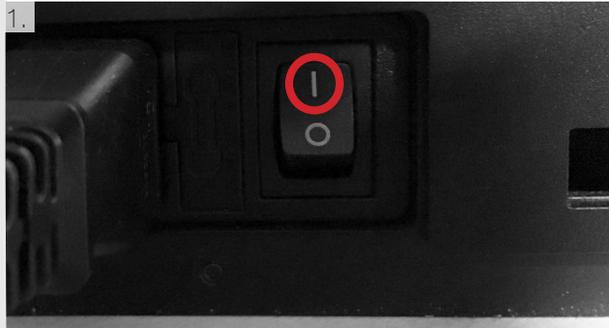


Leveling the Build Plate

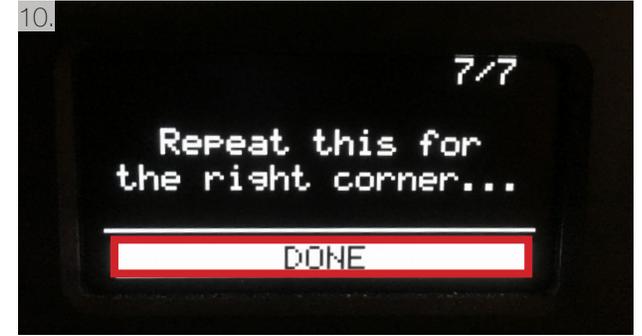
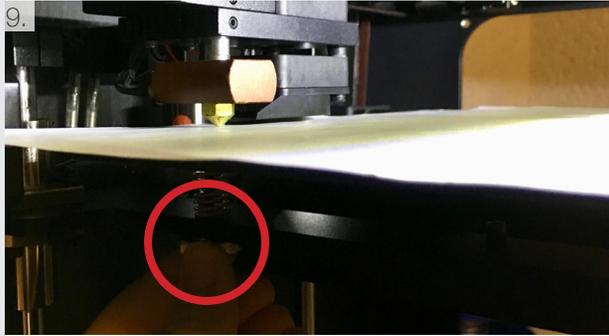
Build plate leveling is crucial to getting a good print. A build plate that is not level will have areas where the nozzle is either too far or too close to the build plate. If the nozzle is too far from the plate, the extruded material will be unable to stick to it. On the other hand, if it is too close to the plate, it could prevent the material from extruding and could cause damage to the nozzle and/or build plate.

Perform the following steps to level the build plate.

1. Turn on the 3D printer. The switch is on the back of the machine.
2. Once startup is complete, the built-in menu system is displayed. Use the navigation wheel to move the highlight to the MAINTENANCE entry, then press the wheel to select it.
3. Select the BUILD-PLATE entry. The system will start the 7-step leveling process.
4. For step 1, use the control knob to raise the build plate, as directed. Press the control knob to select the CONTINUE option when complete.
5. For step 2, adjust the screw under the left-rear corner of the build plate until the plate is 1 mm from the nozzle. Turn the screws clockwise to lower the build plate and counterclockwise to raise it. Select the CONTINUE option to proceed.
6. For step 3, adjust the screw under the right-rear corner of the build plate until the plate is 1 mm from the nozzle. Turn the screw clockwise to lower the build plate and counterclockwise to raise it. Select the CONTINUE option to proceed.
7. Insert a sheet of ordinary printer paper between the nozzle and the build plate, then readjust the screw under the right-rear corner of the build plate until there is a 1 mm gap. Select the CONTINUE option to proceed.
8. The extruder will move to the front center of the build plate. Adjust the screw under the center-front of the build plate until you can move the paper with a slight bit of resistance.



9. The extruder will move to the left-center of the build plate. Adjust the screw under the left-rear corner until you can move the paper with a slight bit of resistance. Select the CONTINUE option to proceed.
10. The extruder will move to the right-rear of the build plate. Adjust the screw under the right-rear corner until you can move the paper with a slight bit of resistance. Select the DONE option to finish the process.



Changing Filament

Before you can print any objects, you will need to insert filament. Perform the following steps to insert filament.

Perform the following steps to level the build plate.

1. Depress the I side of the power switch to power the printer on.
2. Once startup is complete, the built-in menu system is displayed. Use the navigation wheel to move the highlight to the MATERIAL entry, then press the wheel to select it.
3. Select the CHANGE entry.
4. Wait for the printhead to heat before removing the existing material.
5. Remove existing material by pressing down the lever to the left and pulling upwards on the filament. Cut the end of the filament with a scissors at a slight angle. Put filament away. Then select READY entry.
6. Insert new material. Then select READY entry.
7. Wait until your material comes out of the nozzle. Purge all of the prior material.
8. Select which material you are using on the screen. Then select OKAY entry.

