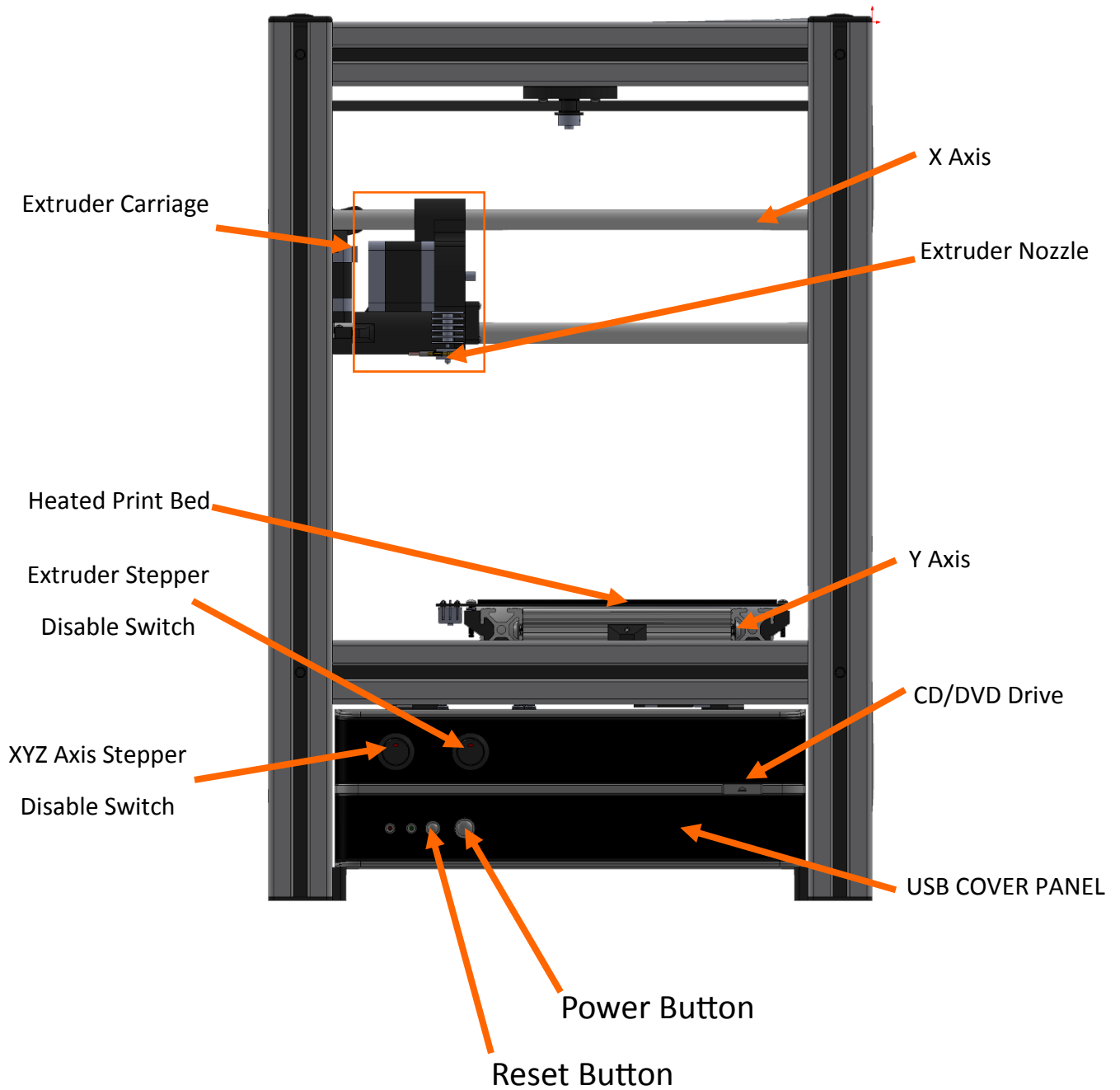


FABRICATOR

User Manual



The Fablicator at a Glance



Fablicator Specifications

Printing

Additive Technology	Fused Filament Fabrication (commonly referred to as FDM)
Build Volume	178 x 178 x 178 mm (7 x 7 x 7in)
Layer Resolutions	60um—300um 125um, 175um, and 250um are standard presets.
Mechanical Precision	X 12.5um 0.0005in Y 12.5um 0.0005in Z 0.06um 0.000002in
Typical Part Accuracy	0.005mm/mm (+-0.005in/in)
Typical Part Precision	+-.25mm (+- 0.01in)
Filament Diameter	1.75mm (.069in)
Nozzle Diameter	0.35mm (.014in)
Travel speed	25mm-300mm/s (1- 12in/s)

Power

AC Voltage	100-240V
Current	3.75A MAX
Wattage	200W Typical During Printing

Computer

Connectivity	USB CD/DVD Ethernet Wireless (optional)
Software	KISSLicer, Fablicator Interface Windows 7
Computer Hardware	AMD A4-4000 450W 12V PSU 2GB RAM CD/DVD 500GB HDD

Temperature

Operating temperature	12C to 32c
Storage temperature	0C to 45c
Maximum nozzle temperature	270C (520f)
Maximum bed temperature	125C (260F)

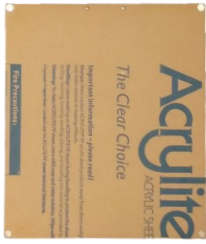
Size

Weight	16kg (35lb)
Size with Spool	43 x 43 x 56 cm (17 x 17 x 22in)
Wattage	200W Typical During Printing

Fabricator Standard Materials

Material	ABS	PLA	PETG
What is it?	A Mixture of Acrylonitrile, Polybutadiene, and Polystyrene.	Starch based, Biodegradable plastic	A type of Polyester Resin
What are its main benefits?	Higher temperature tolerance, slightly more elastic than PLA. Most easily drilled, tapped, sanded and painted.	High strength and least warping, cracking, and delamination when printing.	Strong and impact resistant material. Low warping, cracking, and delamination. High wear resistance.
Maximum Structural Temperatures	95C	60C	60C
Typical Printing Temperature	260C Head 125C Bed	240C Head 75C Bed	240C Head 75C Bed
Optical Properties	Opaque	Most colors opaque or very slightly translucent	Most colors semi-translucent
Common real world uses	Legos, Pipe, Toys	Small food containers, Biodegradable cups, Utensils	Water Bottles, Food Packaging
What are the difficulties printing this material?	Large parts may lift off platform (warp,) crack, or delaminate easily	Small cross section parts may deform if not enough time between layers	Small cross section parts may deform if not enough time between layers. Occasional problems with adhesion to build platform.
Density	1.04g/cc	1.2g/cc	1.38g/cc

Fabricator Accessories:



Front Panel



Calipers



Allen Wrenches and Steel Wire



Rear Panel



Part Removal Tool



Print Primer Bottle



Top Panel



Standard Spool Holder



Antistatic Material Spool Holder



Right Panel



Print Primer Pellets X3



Windows 7 Backup CD



Left Panel



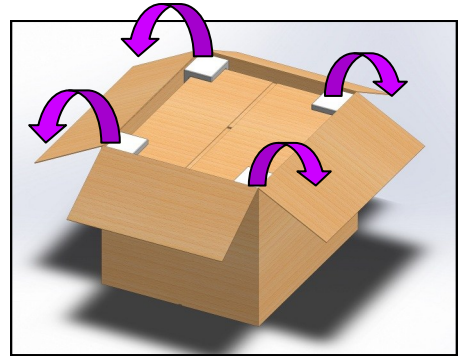
Filament Spool X3



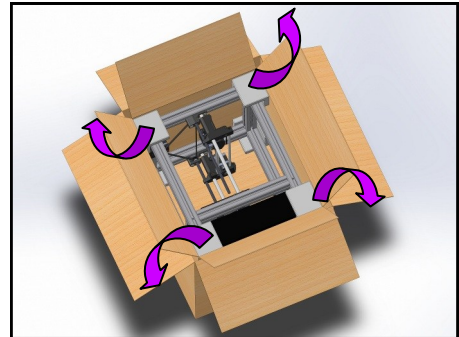
Power Cord

Unpacking the Fablicator

1. Open the large box containing the Fablicator, and remove the foam corners



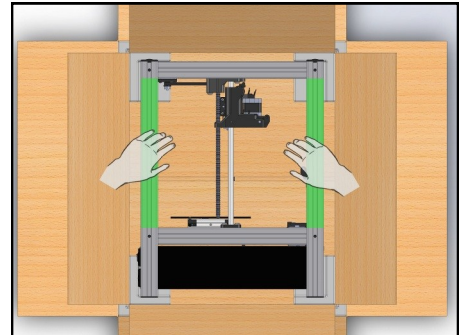
2. Open the Internal box, and also remove the foam corners



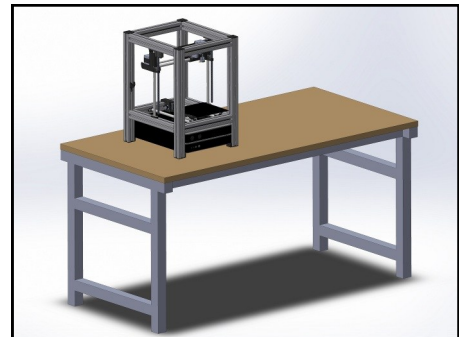
3. Gently lift out the Fablicator, holding it by the frame (highlighted in green)

Some assistance may be helpful for this step

Caution: **ONLY lift the Fablicator by the aluminum frame.**



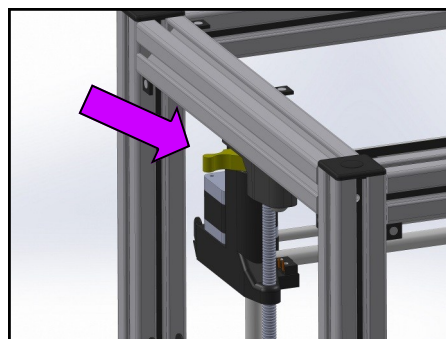
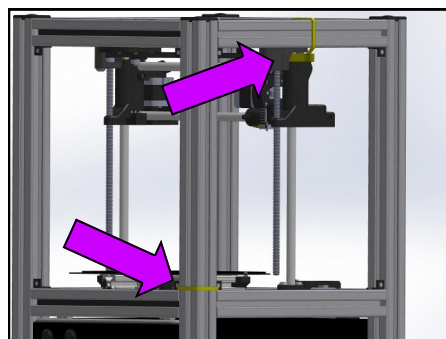
4. Place the Fablicator on a sturdy table or desk.



Unpacking the Fablicator

5. Carefully cut the colored zip ties and remove the yellow clips

Save the yellow axis stops in case the Fablicator ever needs to be shipped.

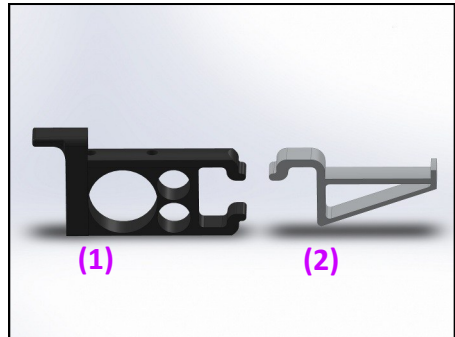


Setting up the Spool Holders

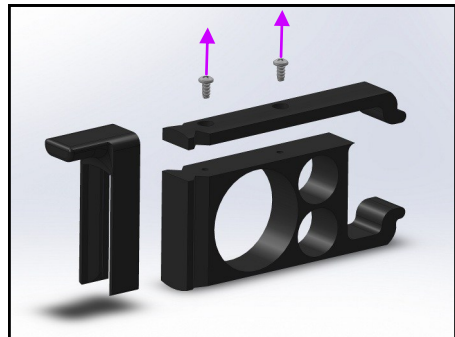
1. The Fablicator comes with two types of spool holders.

The standard spool holder **(1)** is for s ABS, PLA and PETG Spools.

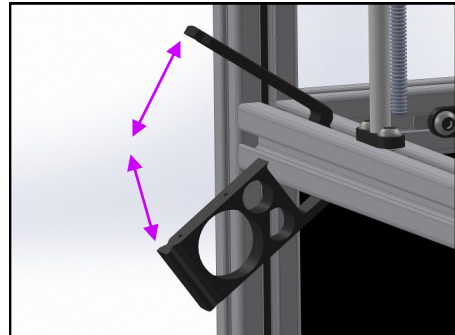
The alternate holder **(2)** is designed for antistatic material spools



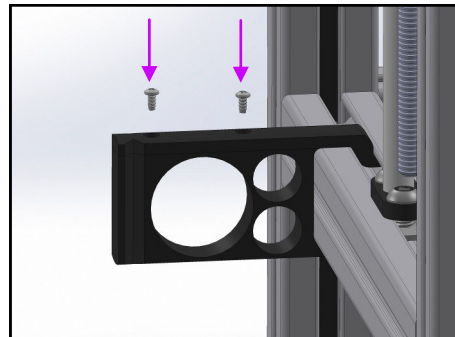
2. Remove the screws from the standard spool holder, so it can be attached to the printer.



3. Slide and Rotate the spool holder halves into the extrusion frame.

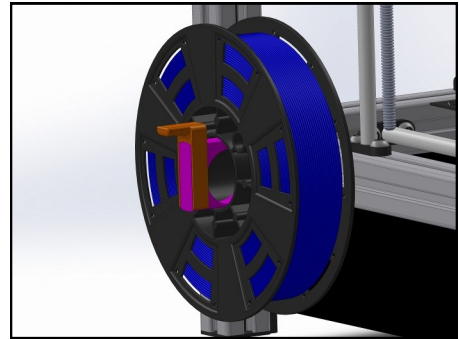


4. When the two halves are brought together, insert and tighten the screws.

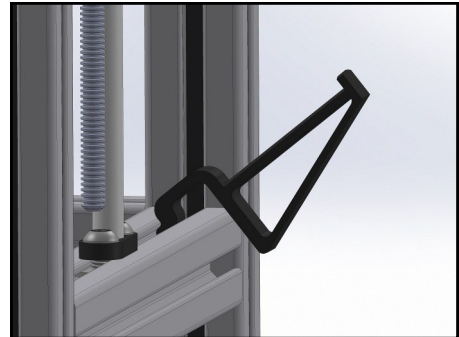


Setting Up the Spool Holders

5. A spool slides onto the holder and the clip is inserted to secure it.

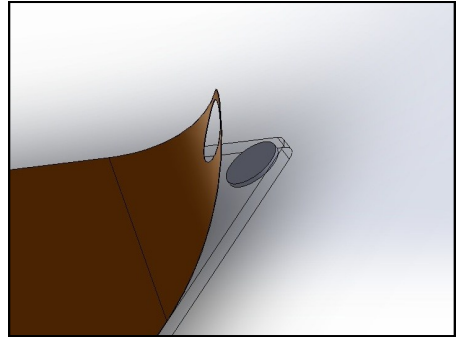


6. The antistatic spool holder is simply inserted and rotated into place on the opposite side of the printer.

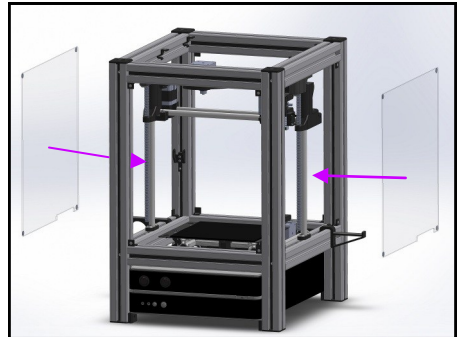


Attaching the Exterior Panels

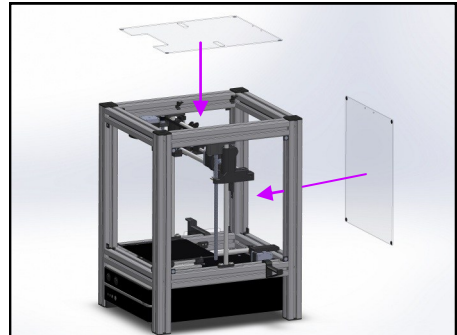
1. Remove the protective film from the 5 exterior panels.



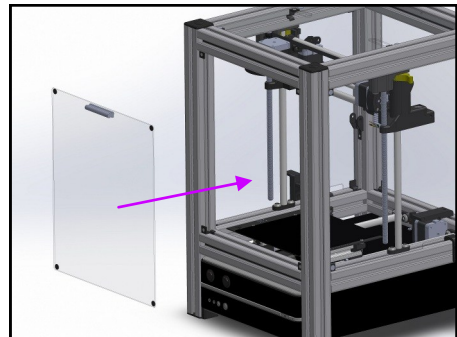
2. Attach the side panels, with the small notch aligning with the spool holder. The magnets on the panel should seat to the magnets on the frame



3. Attach the top and rear panel



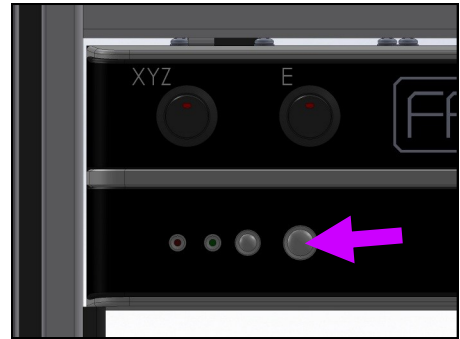
4. Attach the front panel



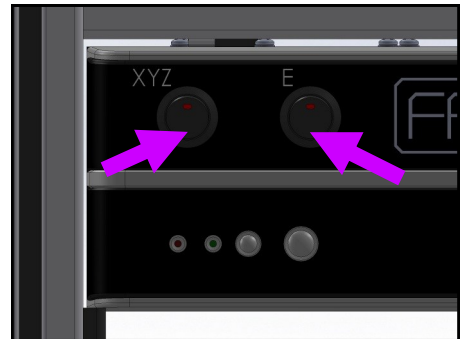
**If additional handles are desired, the .stl file is available in the sample .stl folder on the Fablicator's desktop.*

Loading Filament

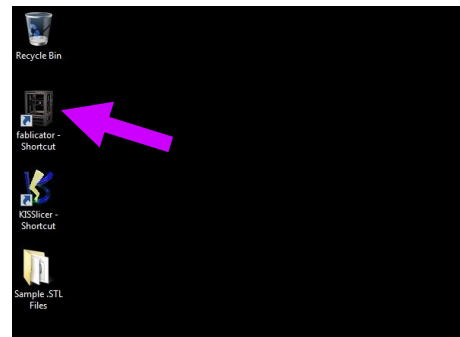
1. Turn on the printer



2. Turn off the XYZ and E Disable switches on the front of the printer. The red lights should be off.

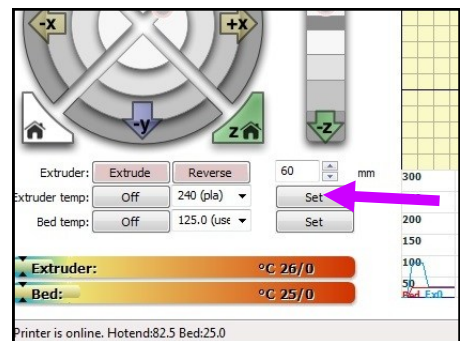


3. Open the Fablicator shortcut on the desktop



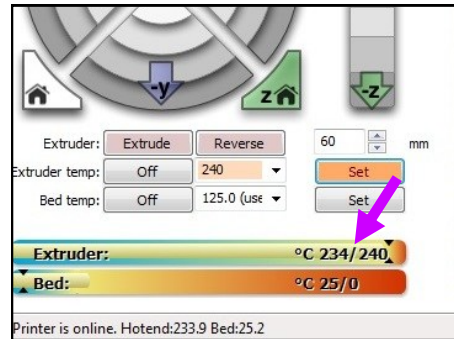
4. Once the software loads, press the “Set” button to the right of the *Extruder Temp* Label. This will heat up the extruder so filament can be loaded and unloaded.

Caution: After this step, the tip of the extruder will be extremely hot. Do not touch it.

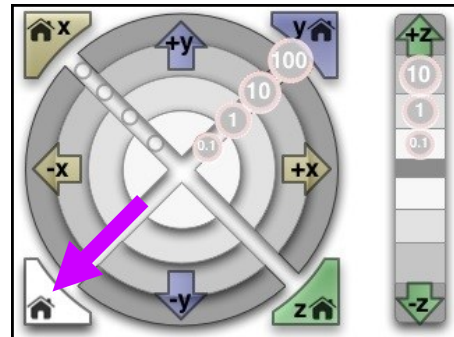


5. Wait for the Extruder temp to exceed 200 C

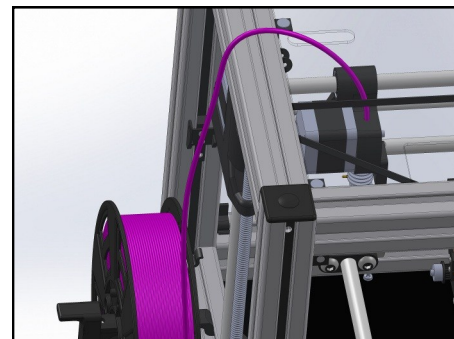
The extruder temp bar shows the current temperature vs the set temperature.



6. Home all axis to make sure the print head is in a convenient position.

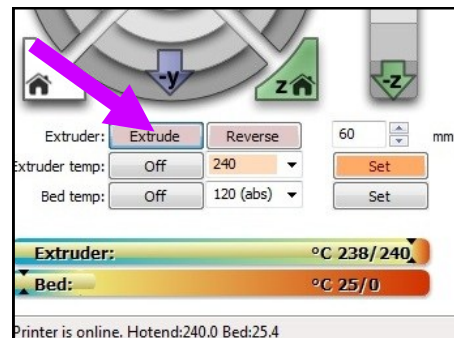


7. With a spool of filament mounted in a spool holder, route the filament up over the top of the frame, through the oval slot in the top cover, behind the belt, and into the small hole on the top of the extruder. The filament should go into the extruder about 3/4"



8. Press the “Extrude” button on the Fablicator interface, and push the filament gently into the top of the extruder until the drive wheel grips it.

Each press of the “Extrude” button sends 60mm of filament into the extruder nozzle.



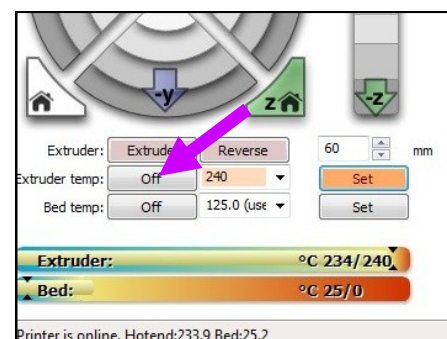
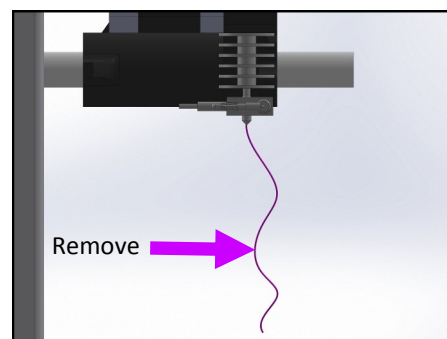
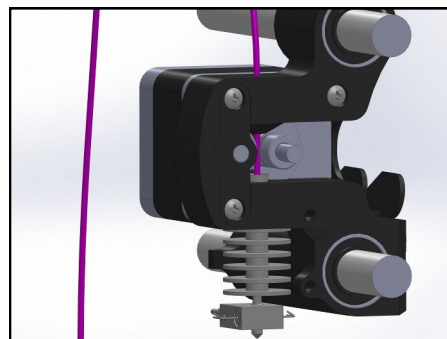
9. The extruder will stop on its own after 60mm of filament has entered the nozzle

10. Press the “Extrude” button again if filament has not been extruded at the nozzle tip by the time the extruder stops.

Once melted filament has come out the tip of the extruder, the filament is fully loaded.

11. Remove any excess extruded filament from the build area. It should be cool a few seconds after leaving the extruder nozzle.

12. Turn off the extruder by pressing the “Off” button next to the *Extruder Temp* text. This is only necessary if you do not plan to print an object in the next few minutes.



Unloading Filament

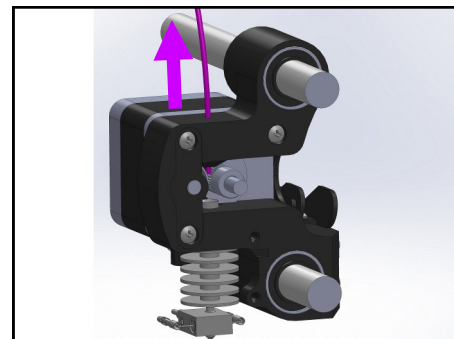
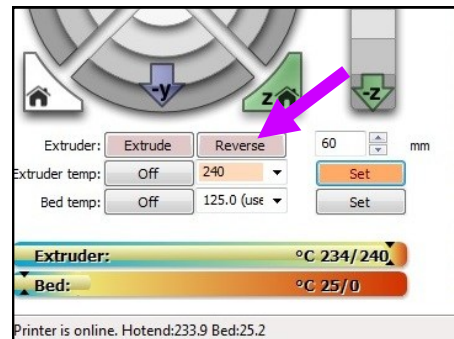
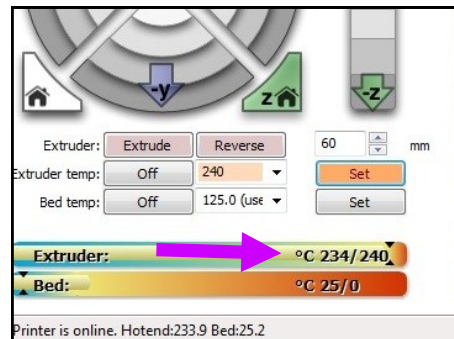
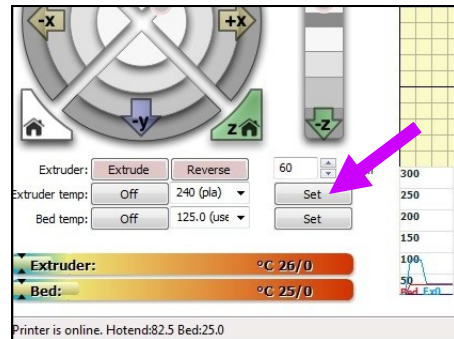
1. With the Fablicator.exe interface open, Press the “Set” button to the right of the *Extruder Temp* Label. This will heat up the extruder so filament can be loaded and unloaded. *If the Extruder is already warm, skip this step.*

Caution: **After this step, the tip of the extruder will be extremely hot. Do not touch it.**

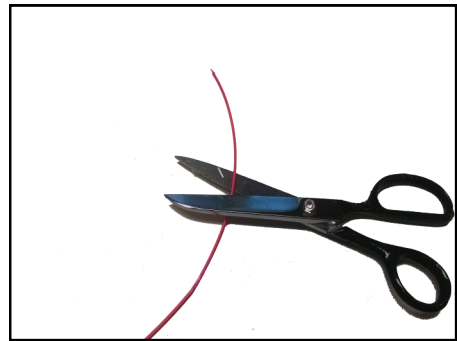
2. Wait for the Extruder temp to exceed 200Degrees.
The extruder temp bar shows the current temperature vs the set temperature

3. Once the extruder is above 200deg C, Press the “Reverse” button. The extruder will back out 60mm of filament from the extruder. The extruder will stop automatically after 60mm of filament has been retracted.

4. After the filament has been backed out, pull up on the filament to finish removing it from the extruder.

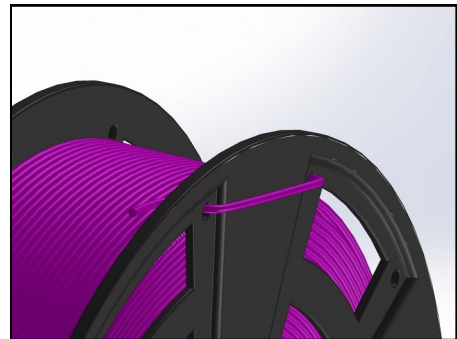


5. Clip the last few inches off the filament to remove the deformed end.

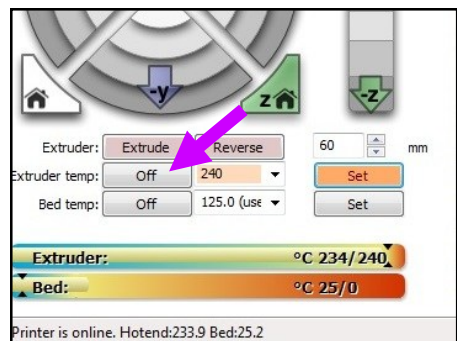


6. Neatly wrap the filament on the spool, making sure to secure the end of the filament.

Caution: **Not securing the end of the filament to the spool may result in a tangle, which will eventually result in a print failure.**

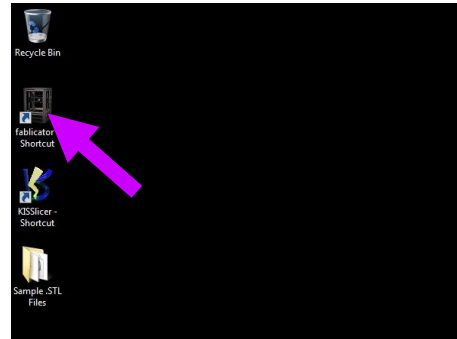


7. Turn off the extruder by pressing the “Off” button next to the *Extruder Temp* text. This is only necessary if the printer will be idle for the next few minutes.



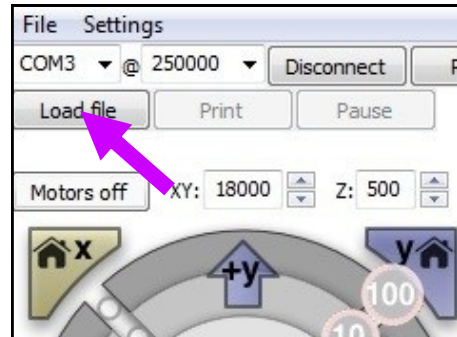
Printing a part.

1. Open Fablicator.exe



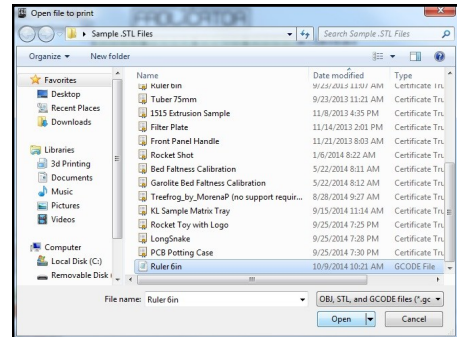
2. Click the “Load file” button in the top left corner

This will allow you to select a .STL file to prepare it for printing.



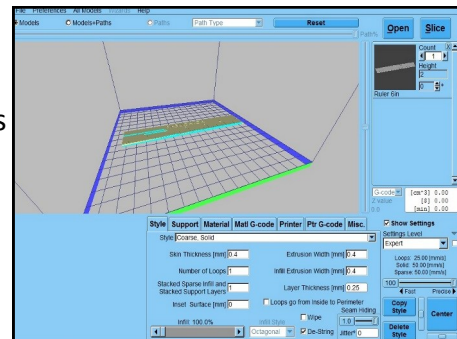
3. Select a .stl file (The example we are using is located in **Desktop > Sample .STL Files > Ruler 6in .STL**) and click the “Open” button

This will open the file with KISSlicer.



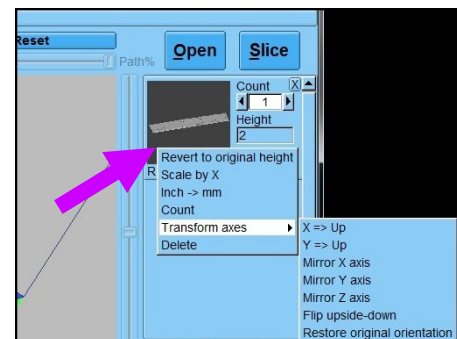
4. Once KISSlicer has loaded, a 3D representation of the part will be visible in the main window. The sample part is a short ruler in this case.

A Small thumbnail of the part will also be visible in the “part box” on the top right of the screen



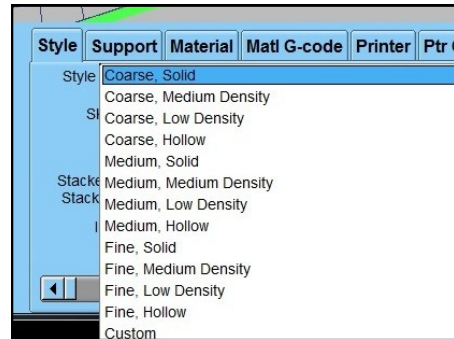
5. A right click on the part box allows a part to be scaled, multiplied, or transformed (rotated, flipped) if desired.

Additional parts can be added to the print using the Open button.



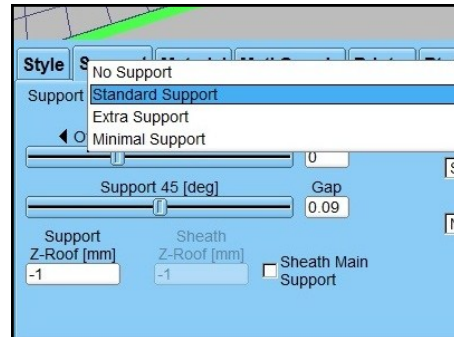
6. Select the desired print settings by clicking the **Style** tab. The example uses *Coarse, Solid* as selected from the drop down menu.

The style settings primarily control the layer height and Infill density of a part.



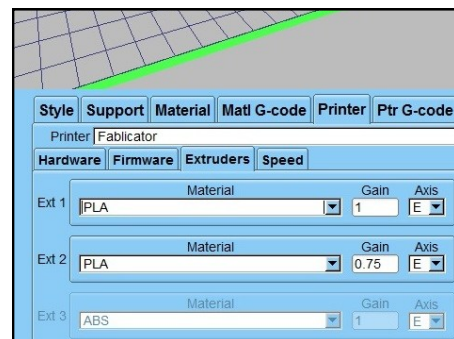
7. Click the **Support** Tab, and select a support setting. The example uses the *Standard Support*.

The support settings control where KISSlicer automatically creates support for a part, so overhangs can be printed successfully.



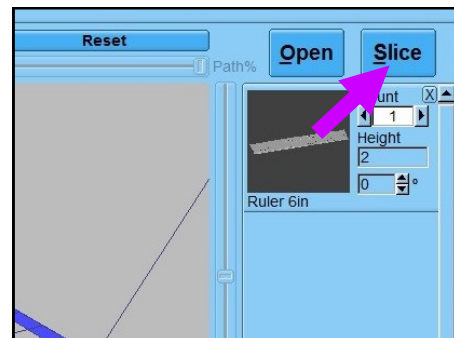
8. Select the **Printer** tab and **Extruders** sub-tab. Change both drop down boxes to the material you are printing with (*ABS, PLA, PETG*)

The material must be properly set in this window for a part to turn out as expected!



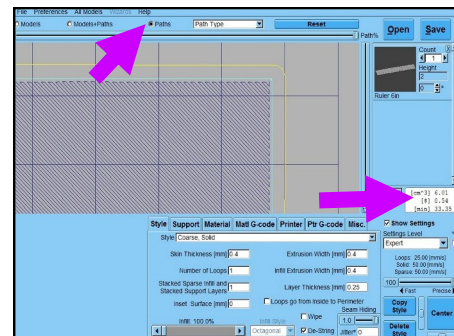
9. Once the settings are selected, Click the **Slice** button in the top right corner.

Choose a filename, and click **save** to create G-Code for the file. This process is called “Slicing” since it divides the part into many layers.



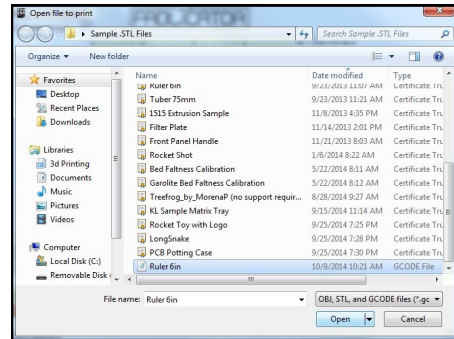
10. After the file is sliced, you can preview the layers using the paths radio button. Print time, material volume, and print cost are also now visible.

Once you are satisfied with the toolpaths, quit KISSlicer. You **MUST** quit kisslicer before the Fablicator interface will allow you to open a G-Code file to print.



11. Click the “Load File” button in the Fablicator Interface, select the G-Code file that was just created, and click Open.

In this example it is *Ruler 6in.gcode*



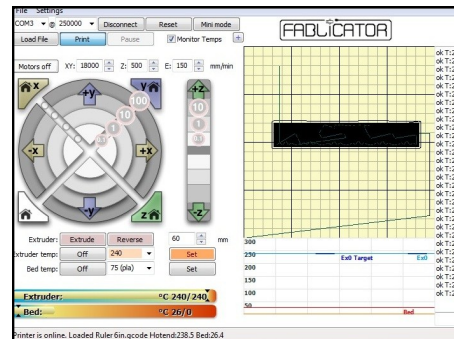
12. Before starting the print, always apply a thin coat of the primer solution to the print surface. Using a paper towel, apply 1-3 squirts and wipe the bed in a circular motion to achieve an even coat.

The bed should still be damp when you finish wiping, and it will air dry within a few seconds.



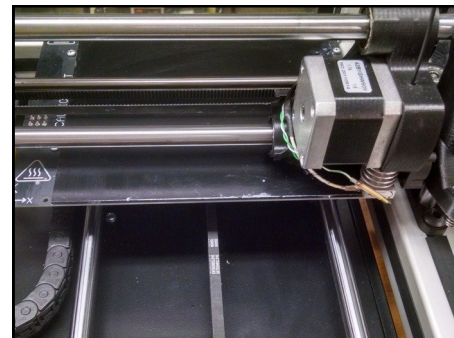
13. Once the bed is primed and G-Code file loaded, press the **Print** button

The Fablicator will automatically home all axis, and move the extruder to a corner of the bed. The printer will also automatically heat up the bed and extruder to the material settings from KISSlicer.

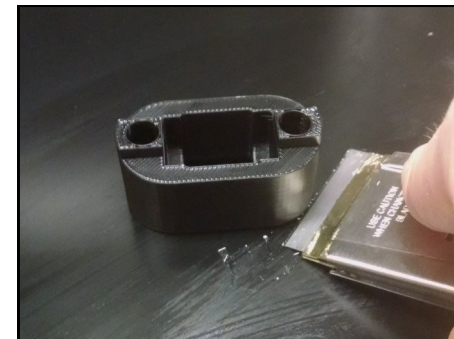


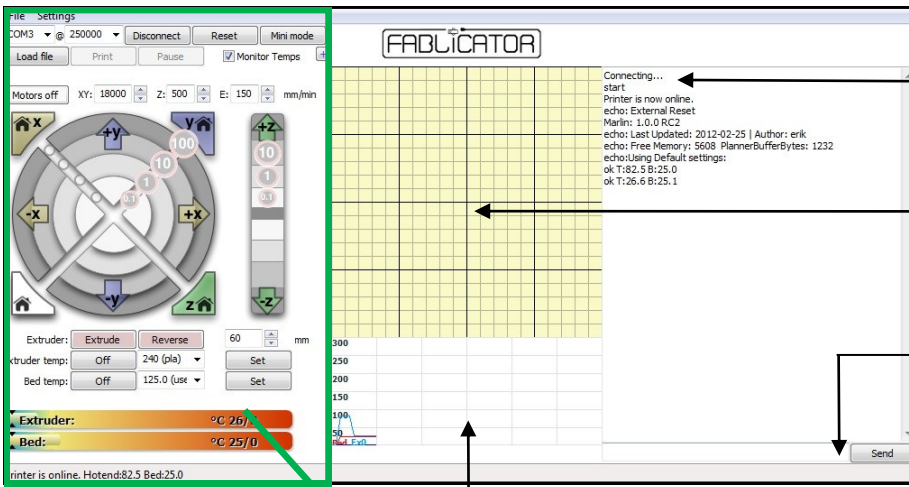
14. When the bed reaches the target temperature, the print will begin.

Once the print has finished, the extruder and bed will turn off automatically.



15. When the print is complete, wait at least 5 minutes for the bed to cool down before removing the part with the safety razor. **If you do not wait for the part to cool down, it may deform when you try and remove it, or be extremely difficult to remove.**



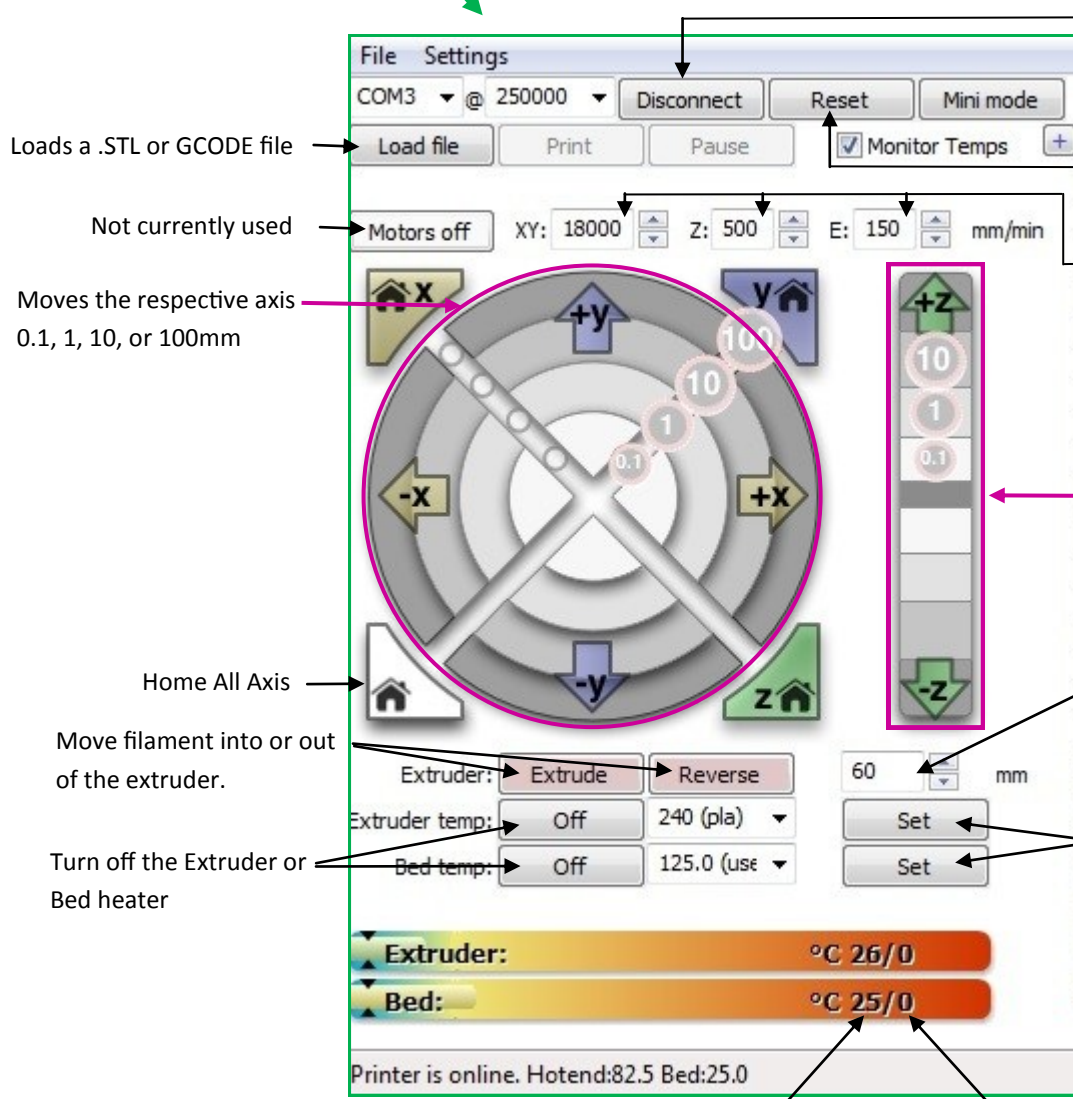


Displays some communications between the printer and the computer

Shows a 2D view of the current print as it is created

G-Code can be manually entered here while the printer is idle.

Chart showing current and set temperature of bed and extruder



Disconnects Printer from computer

Loads a .STL or GCODE file

Hard Reset Printer– Used to cancel print in progress

Not currently used

Feed rate for manual moves and extrudes

Moves the respective axis 0.1, 1, 10, or 100mm

Moves the Z axis in 0.1, 1, or 10mm increments.

Home All Axis

How much filament is moved for each press of the Extrude/Reverse

Move filament into or out of the extruder.

Set the Bed or Extruder temperature to the selected temperature. These settings will be overridden when a part is printed.

Turn off the Extruder or Bed heater

Current Extruder and Bed Temperature

Target Extruder and Bed Temperature

Style Settings

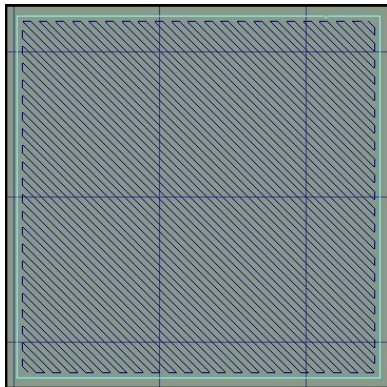
The style settings are broken down into two parts, a layer thickness, and a infill density.

The layer thickness is how tall each division of the part is, and the infill density is how hollow or solid a part will be.

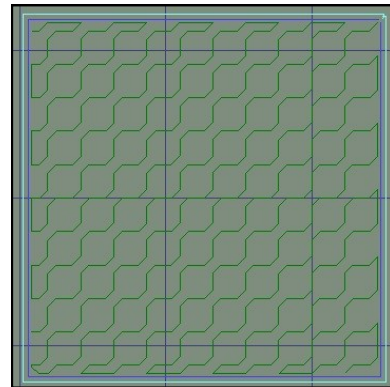
Thicker layers mean much faster print times and stronger parts, and thinner layers will generally have slightly better surface finishes .

Solid parts will be the strongest, while medium or low density parts will print more quickly, and use less material. The hollow setting is usually reserved for special purpose parts which are designed to be hollow. Many parts will collapse onto themselves if printed hollow.

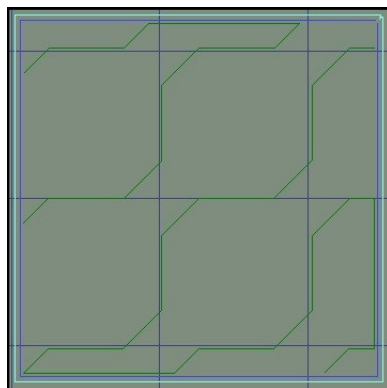
Infill Examples



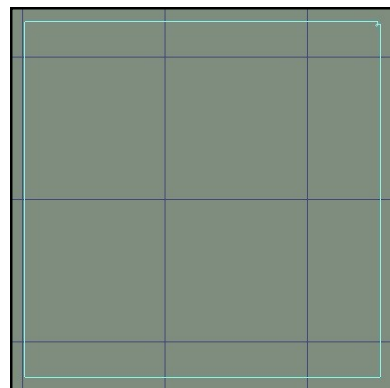
Solid Infill



Medium Infill

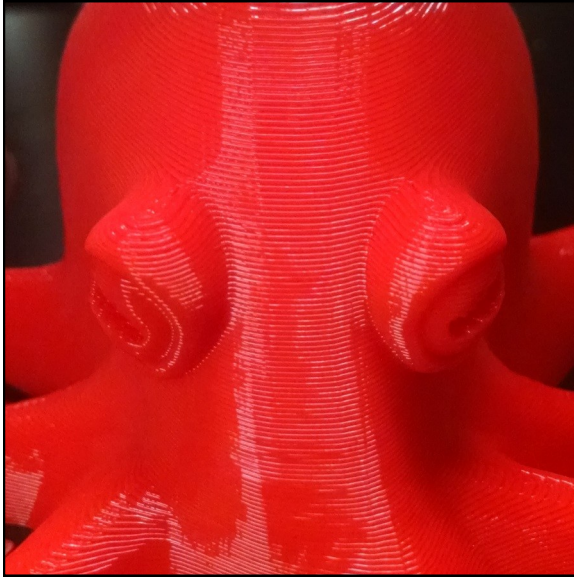


Low Infill

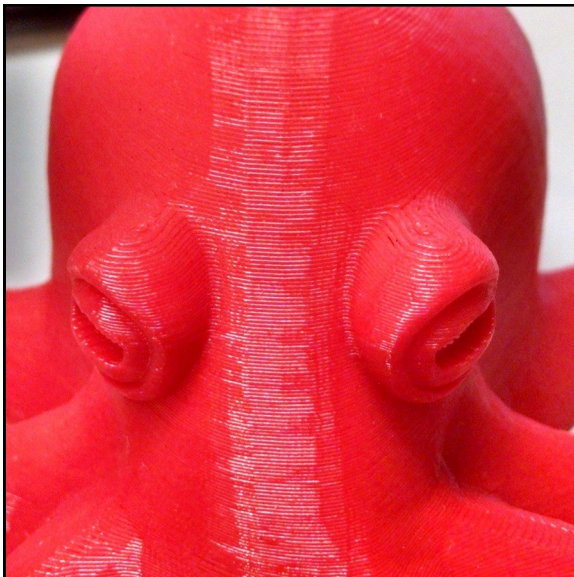


Hollow Infill

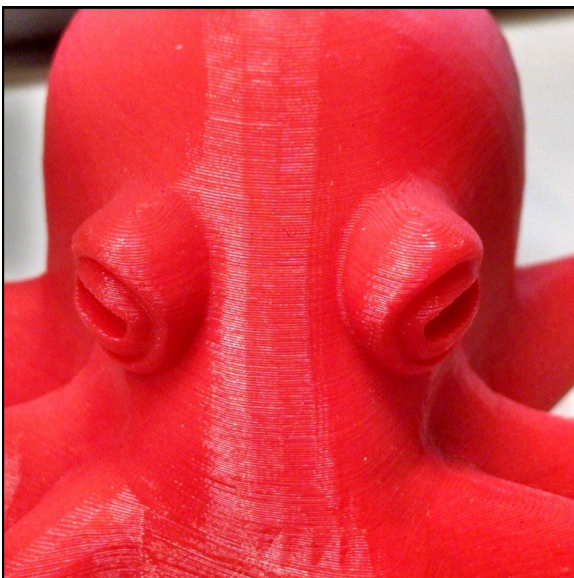
Layer Height Examples



Coarse (0.25mm)



Medium (0.17mm)



Fine (0.125mm)

Support Settings

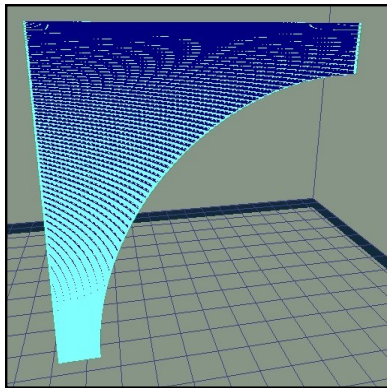
Support allows parts to be created which have nothing else below them.

There are four default settings for support: No Support, Standard Support, Minimal Support, and Extra support.

These primarily differ in what angle of an overhang can exist before support is automatically generated. Generally a part can have a 45deg overhang before support is required, but there are many exceptions to this rule.

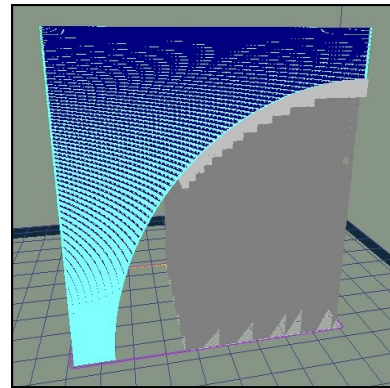
Examples of how support is generated for a half arch are given below.

Support Examples



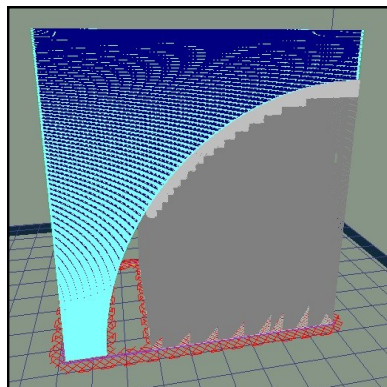
No Support

(Print will likely fail due to material with nothing below it)



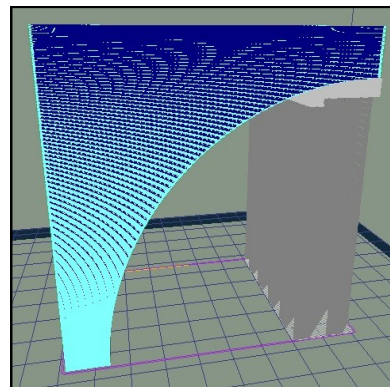
Standard Support

(ideal support for most objects)



Extra Support

(Object well supported, but will take longer to print and may be more difficult to remove)



Minimal Support

(Easier to remove support, but part of the arch may sag or collapse)

Clearing an Extruder Clog

1. If the extruder is making a clicking noise and/or is not extruding filament properly, you likely have an extruder nozzle clog.

This can happen when foreign material enters the extruder and blocks the tiny extruder nozzle.

PLA or PET filament is required for the unplug procedure.

2. First, set the extruder temperature to 240C and wait for it to get near the target temperature.

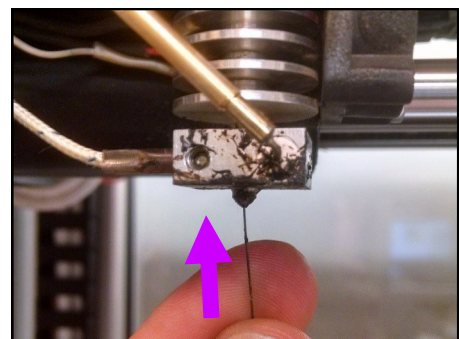
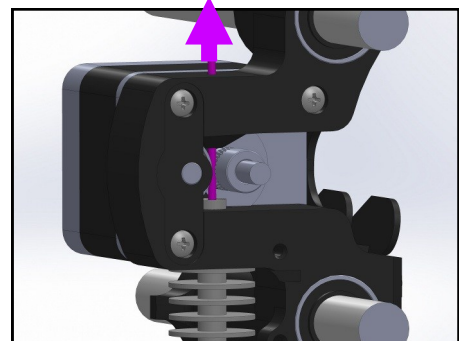
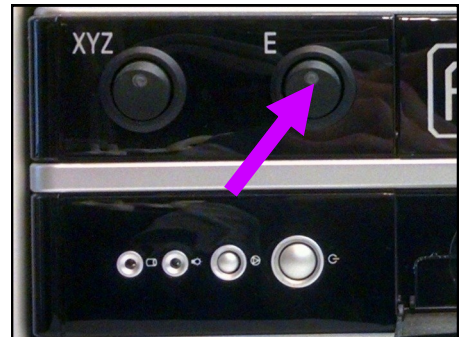
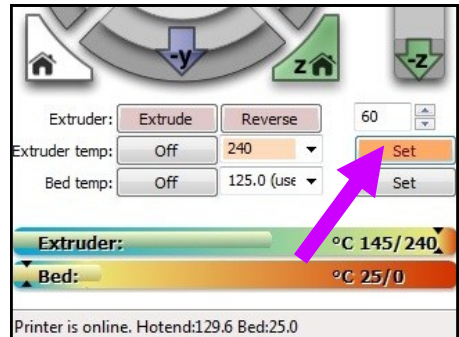
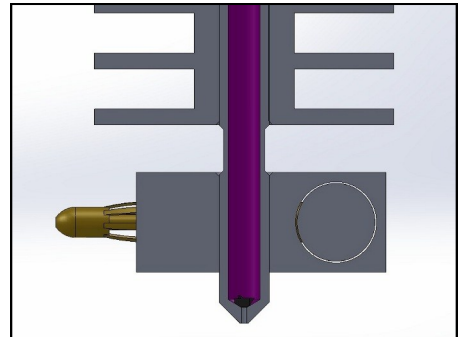
3. Switch on the extruder disable switch on the front of the printer. The little red light should turn on.

This means the stepper motor controlling the extruder can rotate freely.

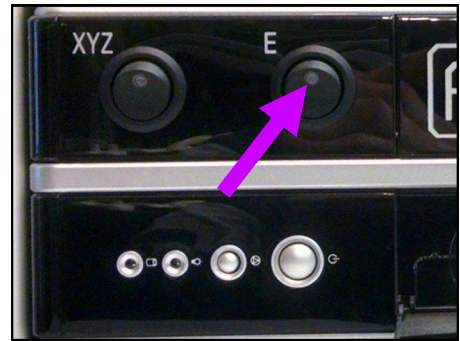
4. Pull up on the filament going into the extruder, this should remove the filament.

5. Use a piece of the music wire from the starter kit and carefully insert it through the tip of the nozzle. Try and push it all the way through the extruder assembly and out the top.

Pull the wire completely through the nozzle assembly.

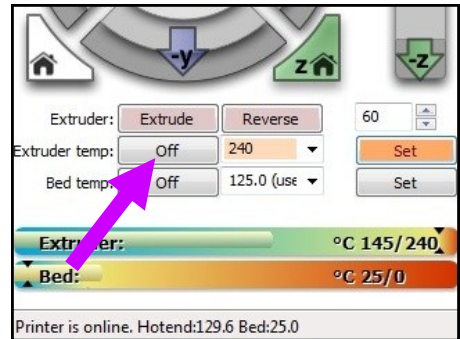


6. Turn the E disable switch off (red light not lit) and load a piece of PLA or PETG filament into the extruder as you normally would. (with the extruder hot, insert the cut end of the filament into the extruder top and press extrude)



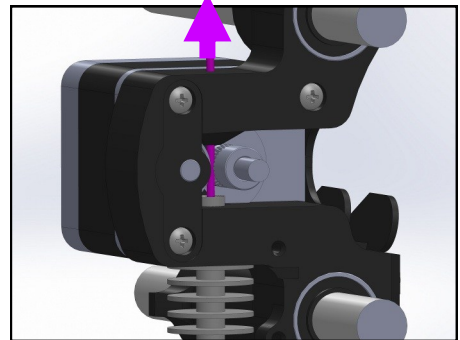
7. When the extruder stops clicking and/or extruding turn the extruder heater Off.

Also, flip the Extruder Disable switch on the front of the printer (red light should now be on)



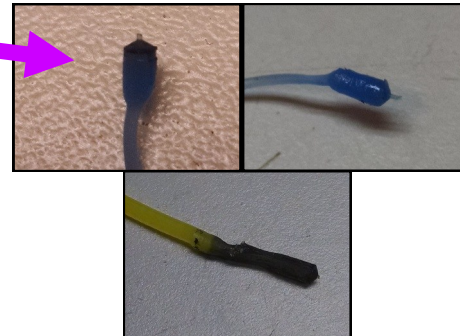
8. When the extruder temperature reaches 90C for PLA (120C for PETG) pull up on the filament above the extruder.

This should pull out all of the filament inside the extruder head.



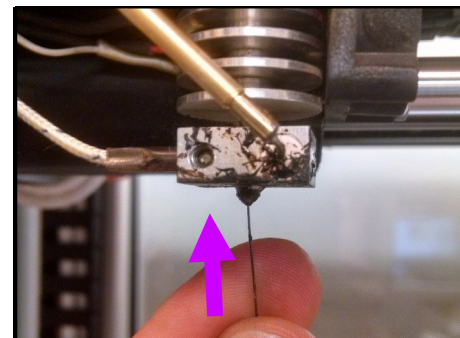
9. The tip of the filament should look like this.

If the tip of the filament does not have the conical shape, the filament was not completely removed. You will need to repeat steps 6-8 until the tip of the filament looks similar to the example.



10. Once the filament is removed, set the extruder temperature to 240C and run the .012 steel wire up through the nozzle and out the top of the extruder.

Load filament as normal and press the extrude button a few times to ensure material is being extruded properly.



11. If the clog has not been cleared, repeat steps 6 through 10. Most clogs will clear in 1-2 tries but sometimes it may take additional attempts.

If you can't seem to clear a clog, don't hesitate to contact technical support :

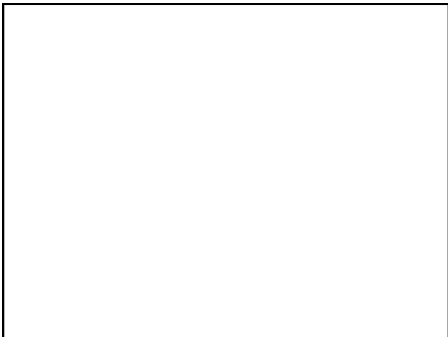
technicalsupport@fablicator.com

or

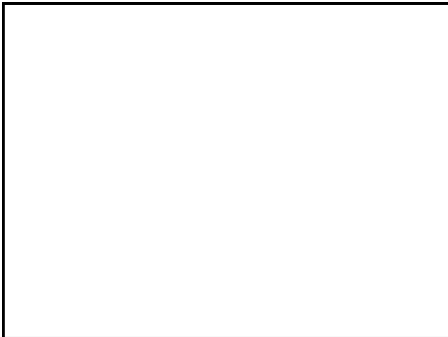
610-295-7867 between the hours of 9am and 5pm EST.

Title Text

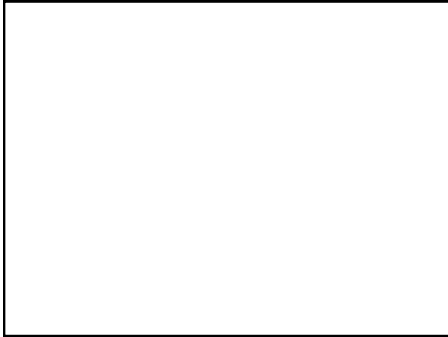
TEXT



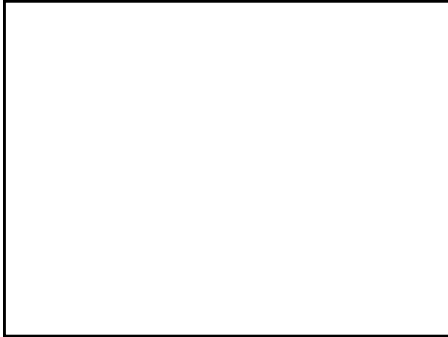
TEXT



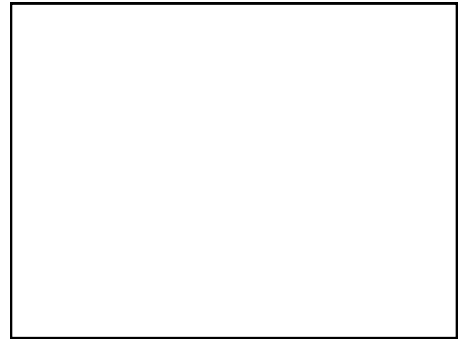
TEXT



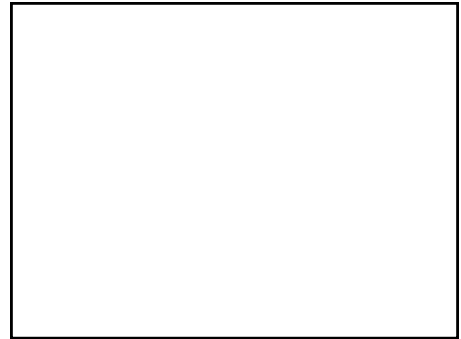
TEXT



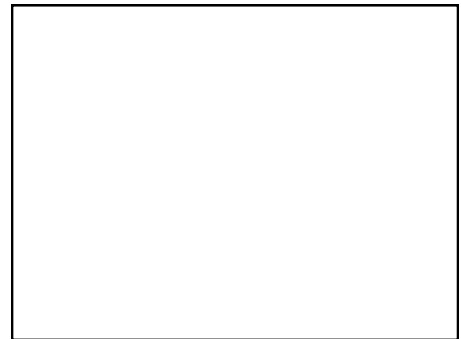
TEXT



TEXT



TEXT



TEXT



TEXT

