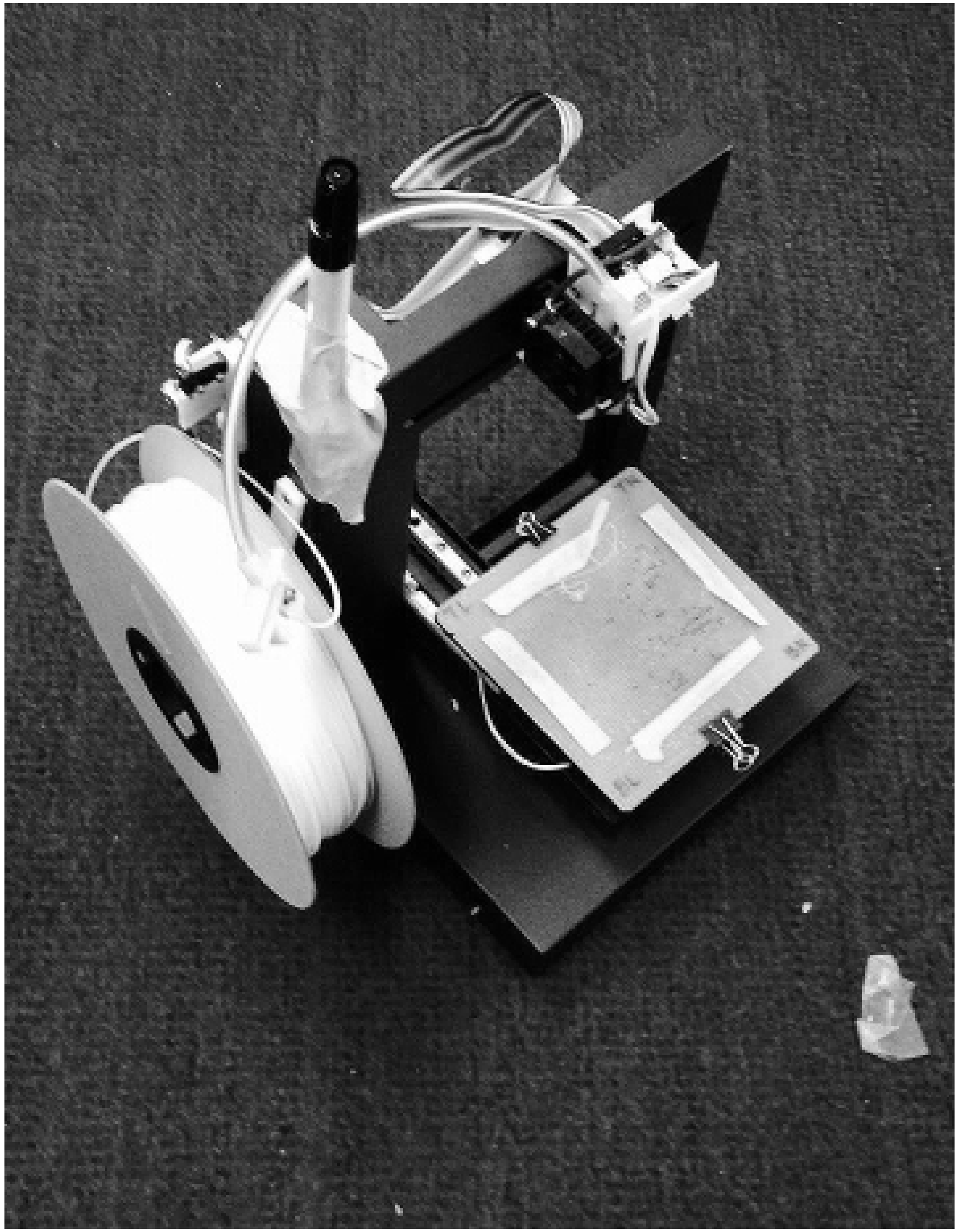
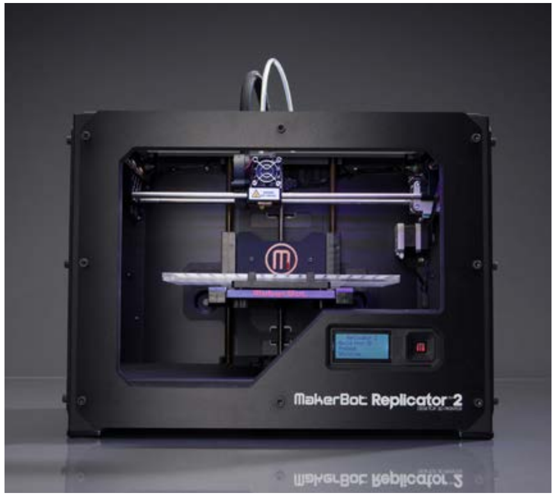


3D printers are rapid prototyping tools which make solid, three-dimensional objects out of melted plastic filament. Digital models are translated into instructions for the 3D printer (3d models are converted into a series of 2D slices) and read by the machine. The 3D printer then heats the plastic filament and squeezes it out through a nozzle onto a heated surface to build a solid object, layer by layer. This method is called Fused Deposition Modelling (FDM) or Fused Filament Fabrication (FFF).

WHERE DO WE PRINT ?

We have two types of plastic printer (Makerbot and Up) and one powder printer at School of Architecture.



Makerbot
in OML Office

up-printer
hirable in studio

powder printer
in workshop

WHEN DO WE PRINT ?

The Powder printer is operated by technicians in the digital fabrication workshop, multiple files are loaded and printed at the end of the day. Prints will usually be completed the morning after, but it is recomended you check with the technicians at the workshop 1-2 days before you print.

Plastic printers are located in the OML office and can be hired by design tutors for studio use. Typically there are long queues for these printers due to long printing times and a high demand. It is unpredictable when your file will be printed, but we recommend you to load the file 3-4 days beforehand.

WHAT DO WE NEED ?

An **enclosed** 3d model saved as **STL** format.

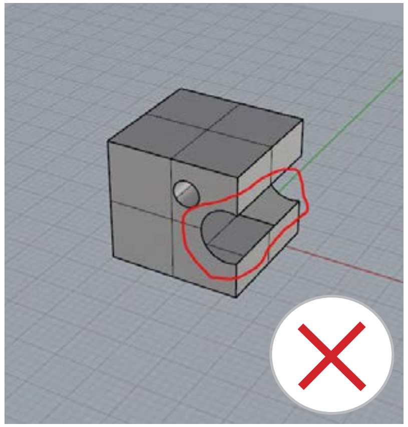
HOW DO WE PREPARE A FILE FOR 3D PRINTING?

This instruction guide will show you step by step how to prepare a printable file using the 3D modelling software Rhinoceros (this software package is avaliable on all the computers in architecture studio and lab). Your model must meet the requirements below to print:

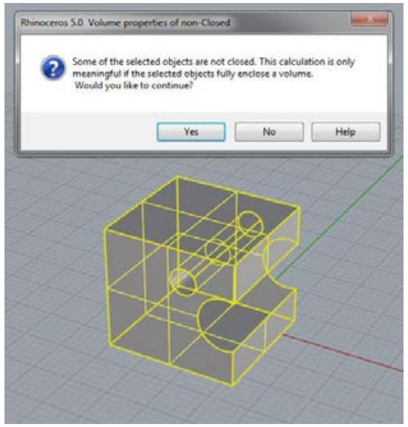
- Model must be a **solid volume**, it cannot print planar surfaces, i.e. it needs a **thickness**.
- Save your model in .**STL format** (mesh format – NURB surfaces will be converted into a collection of triangles).
- Make sure your model has no holes, the volume must be fully **enclosed**.
- Make sure surface normals are facing **outwards** and not inwards.
- Model dimensions must be able to fit in the printing bed constraints (width, height and depth).

Preparing your file in Rhinoceros

If you build your entire 3D model using NURBS surfaces in Rhinoceros, it is quite common to have non-enclosed volumes. This guide will demostrate how to check if an object, constructed from NURBS, is enclosed in Rhinoceros.



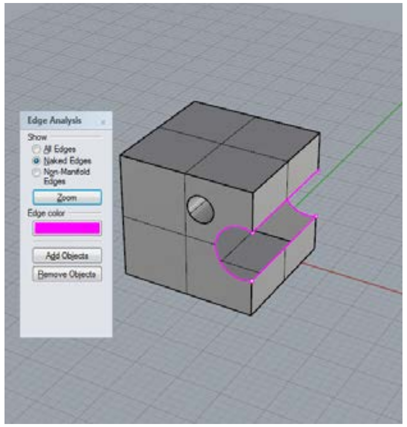
A non-enclosed volume.



To check if the object is enclosed, select the object, then go to:

[analyze->mass properties->volumn]

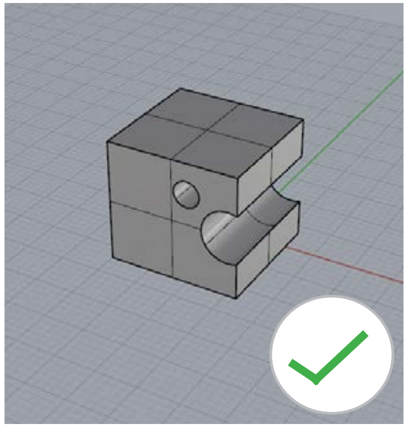
if there is a pop-up window like this, your model is not enclosed.



Select the object, then go to:

[analyze->edge tool->show edges]

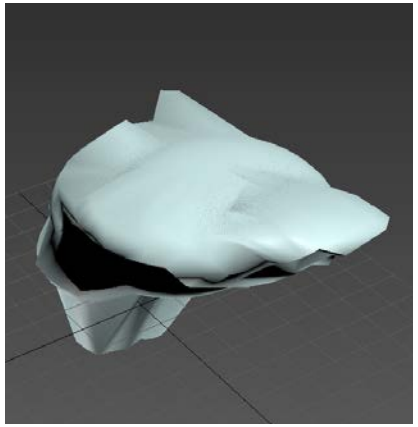
click show naked edges to show the unenclosed edge.



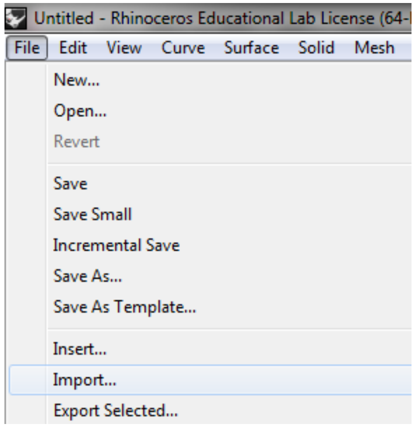
Close the volume appropriately, then use the command join to create one solid form, you will then get an enclosed volume that is printable.

Importing models created in other programs

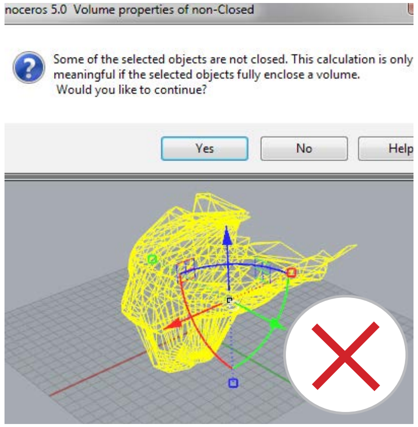
Files created from other programs (e.g. Archicad, Revit, 3dsmax etc.) can be imported to Rhinoceros for analysis.



A 3D model built in 3dsmax is used to demostrate the process.



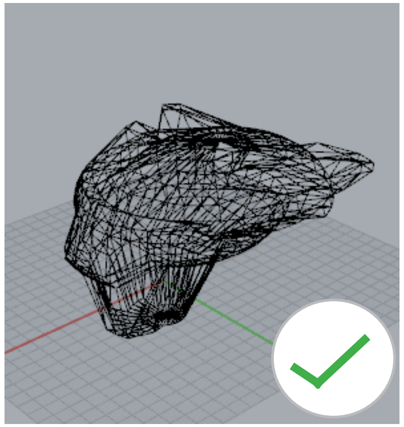
Import your 3D model to Rhinoceros to test whether it is enclosed or not.



To check if the object is enclosed, select the object, then go to:

[analyze->mass properties->volumn]

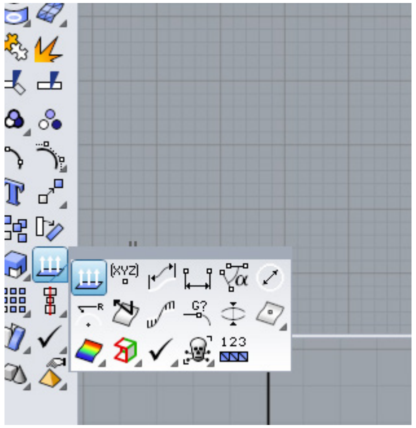
if there is a pop-up window like this, your model is not enclosed.



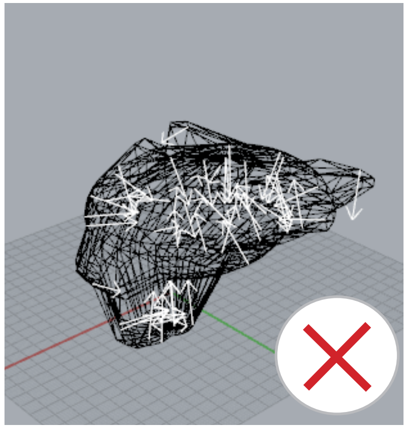
Fix the model in the program you built the 3Dmodel, then import the model again to check the volume. Repeat until the volumne is shown without pop-up window.

Making sure your normals are facing right!

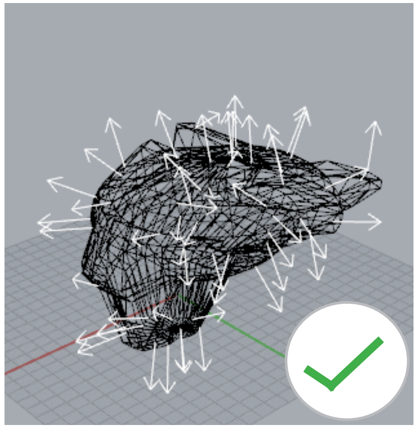
You need to make sure the nomals of meshes and NURBS surfaces are facing outward.



At last, make sure the normals are facing toward correct direction using “analyze directions”.



If it shows that any surfaces are facing inward, it might not be printable.

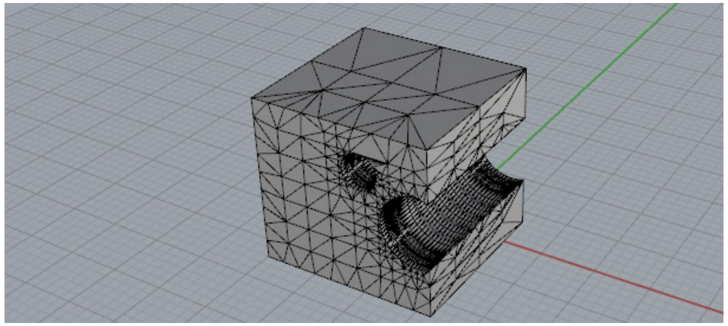


Click “flip” in the top bar to make surfaces face outward, to ensure your file will print.

Saving the file as STL

Accoring to Guide 1 and 2: after confirming that the object is a closed volume, select the object, go to [File -> Export Selected -> Export it as STL fomat], click "OK" at the "STL mesh export option window", and "Binary" at the next step.You will now have a printable STL file.

(If you open the STL file with Rhinocers again, the object will be converted to Mesh object and will no longer be editable as a NURBS)



Size of your object

Please make sure the object's size fits within the printer's boundary.

UP Plus 2

- Single extrusion only
- Maximum bed size W: 140mm, D: 140mm, H: 135mm

MakerBot Replicator 2X

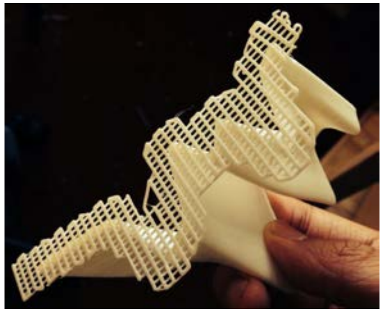
- Single or dual extrusion
- Maximum bed size W: 246mm, D: 152mm, H: 155mm

Hand in your file

If you are printing with powder printer, please hand in the file to a technician in our digital fabrication workshop.
If you are printing with plastic Makerbot, please hand in the file to OML.
If you are printing with plastic UP in our studio, please follow the following guide for file set up.

SO MY OBJECT IS GUARANTEED TO PRINT, RIGHT?

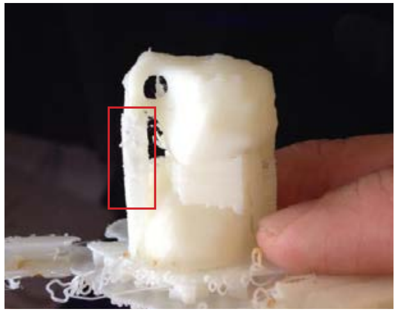
We can not guarantee your model will successfully print as there are many factors which can affect it. The following guideline will help you understand FDM printing at a greater detail to help you achieve a successful print.



Raft

A raft is a flat base that is printed before the object. A raft can help your object stick securely to the build plate. A raft consists of several layers forming a flat surface for your object to rest on. Objects printed with rafts sit on this base of disposable material instead of directly on the build surface. If your build plate is uneven or has imperfections, the raft might provide a better build surface. The base layer of the raft is thicker than a regular layer and prints more slowly, so it adheres better to the build plate. The option to build a raft can be toggled via the 3D printing software.

NOTE: It is recommended that you include a raft for best results when printing with support. Support structures might not adhere well to the build plate without a raft.

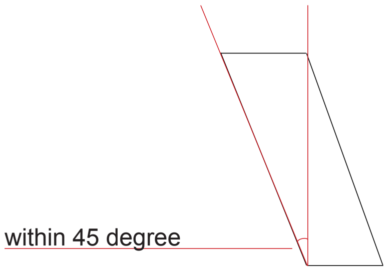


Support

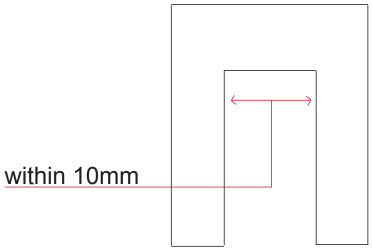
Supports allow you to print overhanging parts. Because 3D printers cannot print into thin air, supports provide a base for overhanging parts of a model. You can easily remove supports after building the object. Remove support material by tearing it away with a pair of pliers or your fingers. Supports can leave surface imperfections that require finishing of the object.

You can build an object of any shape using tearaway support structures, but supports can be difficult to remove and use up plastic. Make sure your object has a flat side to rest on. Orient your object to minimize overhangs and bridges.

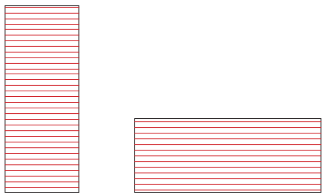
EXTRA TIPS FOR PRINTING



Overhangs (Applied if you do not want to print with support structure)
As the printer prints your object, each layer of plastic rests on the one below. When an object has straight sides, a new layer will be fully supported by the previous layer. But when an object has parts that angle outwards (overhangs) some part of the new layer is unsupported. If it's just a thin sliver of the perimeter, the layer will still have enough support, but if more than half of that outer perimeter has nothing to rest on, you might see drooping or dropped threads of plastic. To make sure at least half of each layer's outer perimeter is supported, avoid creating overhangs that form an angle greater than 45° from the vertical.



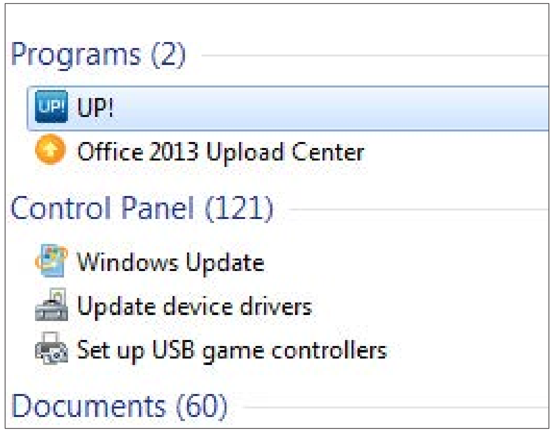
Bridges (Applied if you do not want to print with support structure)
A thread of extruded plastic that crosses from one supported area to another supported area over an unsupported area is called a bridge. Because the thread is supported at both ends, the unsupported middle does not cause problems. But if the unsupported section is too long, you might see sagging in the middle. 3D printers should handle bridges of 10 mm well, but you will probably see some sagging on bridges of 20 mm.



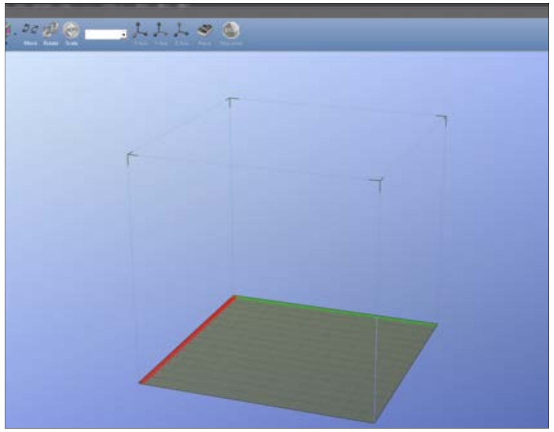
Maximizing Object Strength
Another thing to consider is the "grain" of the printed object. A thin cylinder printed upright will be composed of a series of circles placed on top of one another. If you try to break it in half, it will have as many natural breaking points as it has layers, and will snap easily. If, on the other hand, it is printed on its side, the layers will run up along the cylinder, and every layer will have to bend in order for the cylinder to break. Keep this in mind when designing objects, especially functional ones. A spool holder is a good example of an object that should be printed on its side for strength.

SELF-PRINTING GUIDELINE FOR UP PRINTER

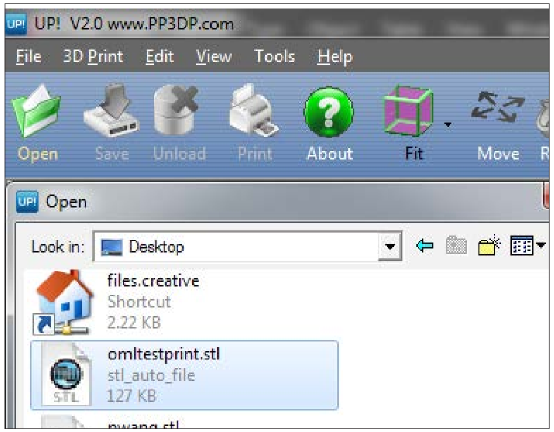
1. Open **Up!** interface



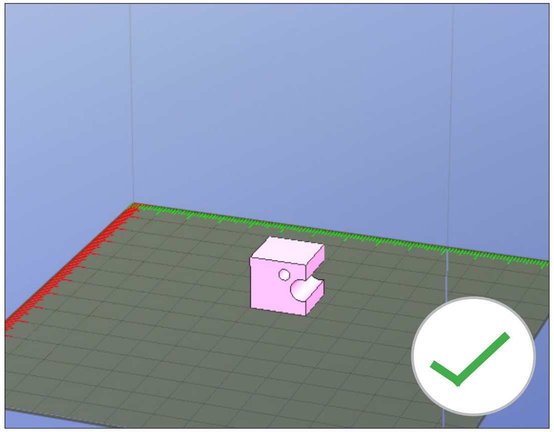
2.This is how it looks.



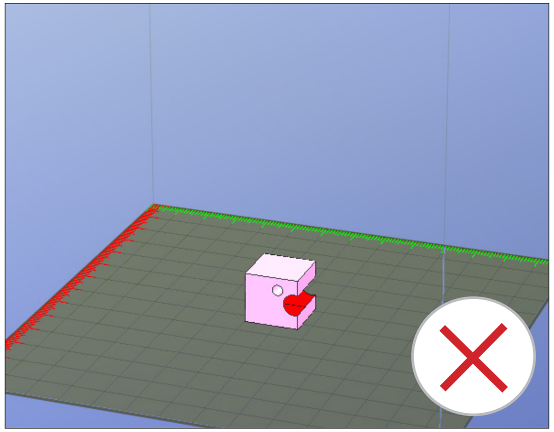
2. Click Open, then select the STL file.



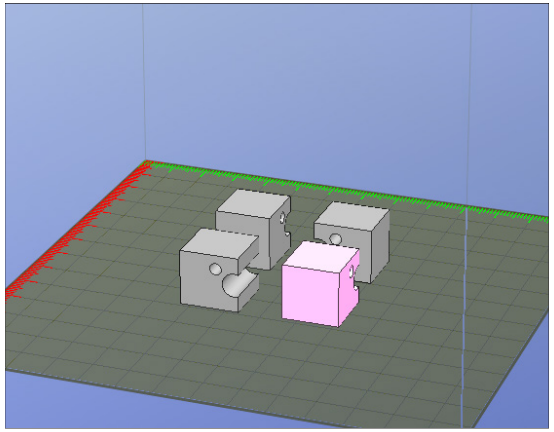
4. Your STL object should be a pink object to be printed.



5. The object is not enclosed if there is red. Fix your model before proceeding.

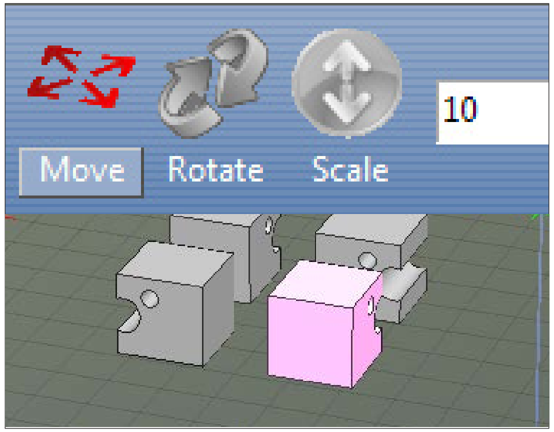


6. You can load multiple objects into Up!.

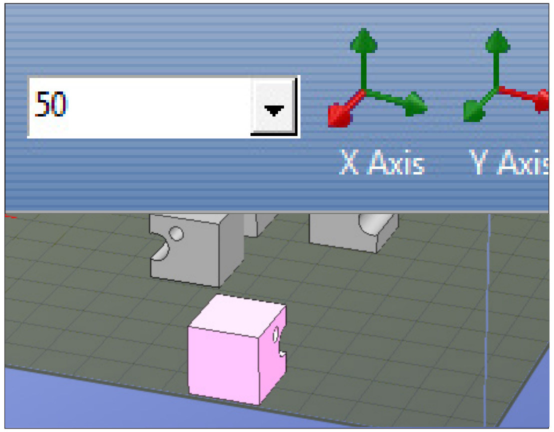


How to move objects?

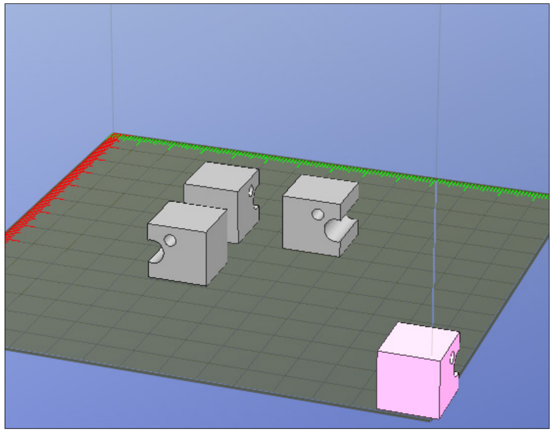
1. Select a object, click on “move”.



2.Type in 50 and click “X Axis”, then it will move down 50mm in the X direction.

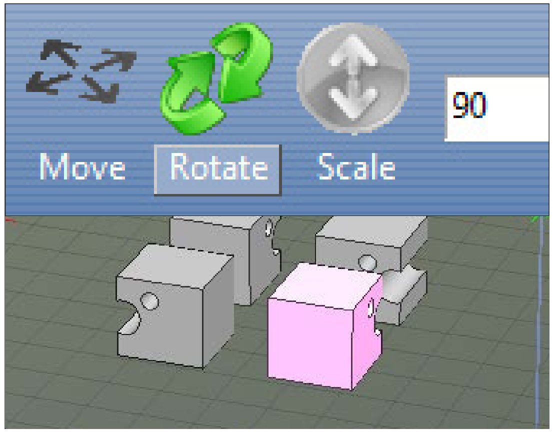


3. Click Y Axis, then it will move 50mm in the Y direction.

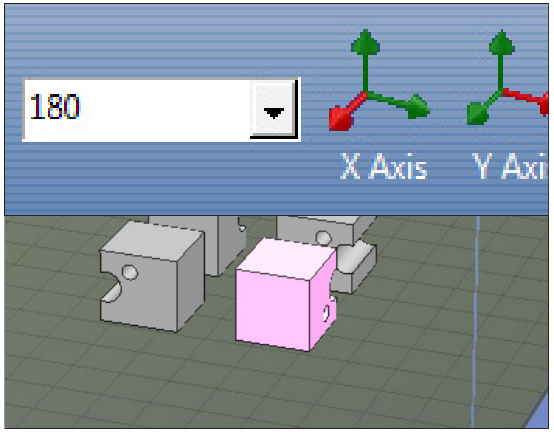


How to rotate objects?

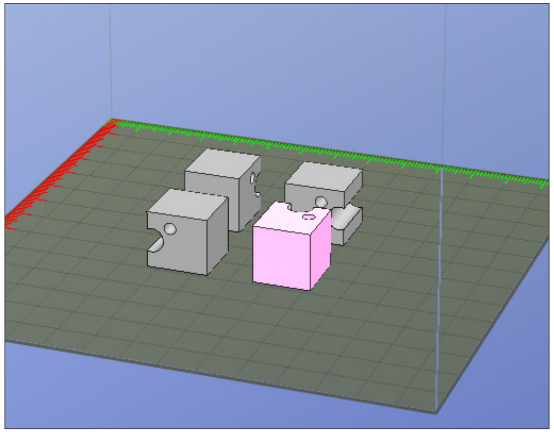
1.Select a object, click on rotate.



2. Type in 180 and click “X Axis”, then it will rotate 180 degree in the X axis.

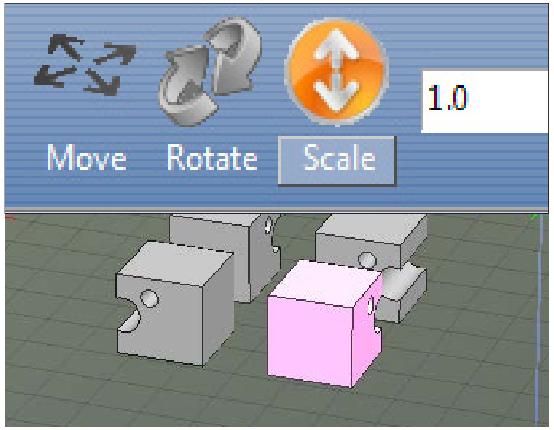


6. Following the same method, objects can rotate in the Y and Z axis also.

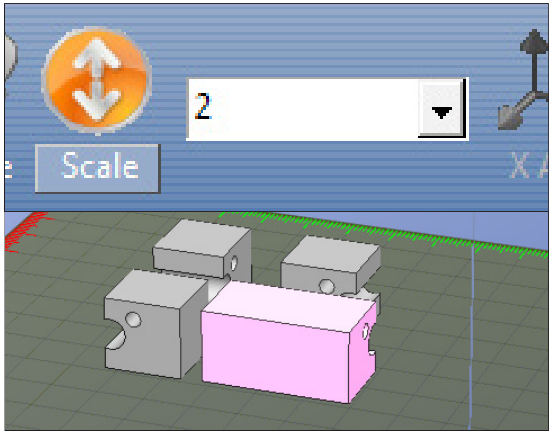


How to scale objects?

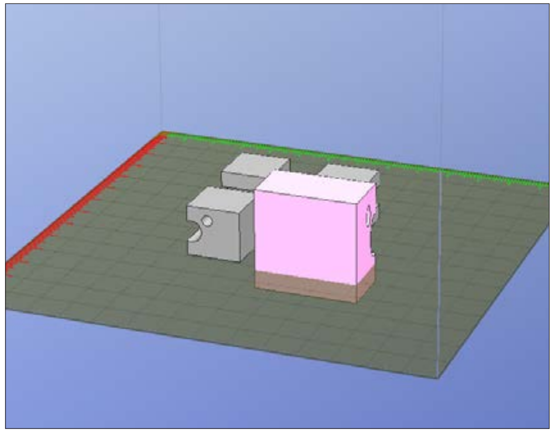
1. Select a object, click on “scale”.



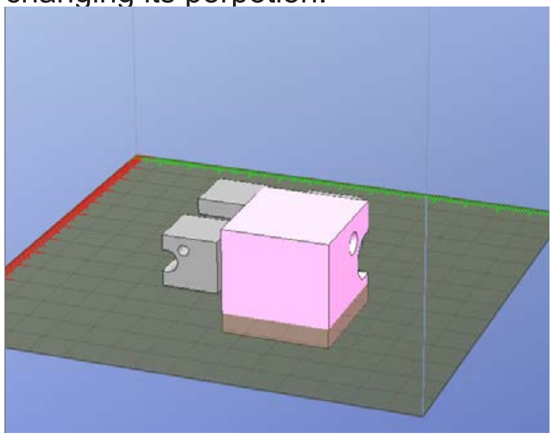
2. Type in 2 and click “Y Axis”, then it will scale by a factor of 2 in the Y Axis.



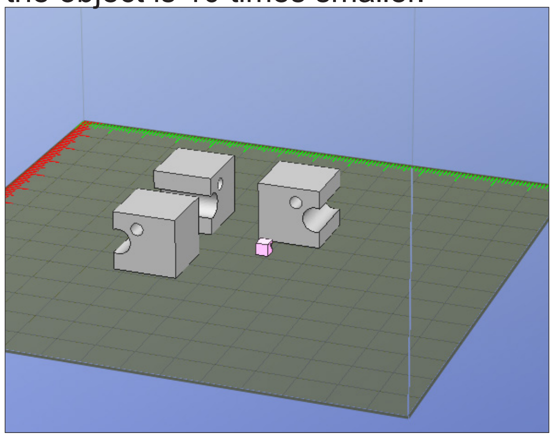
3. Click Z Axis, then it will scale vertically in the Z Axis.



4. Click X Axis, then it will scale horizontally in the X Axis. The object is now 2 times bigger than the original size. You will need to scale in all 3 axis to enlarge or reduce its size without changing its porpotion.

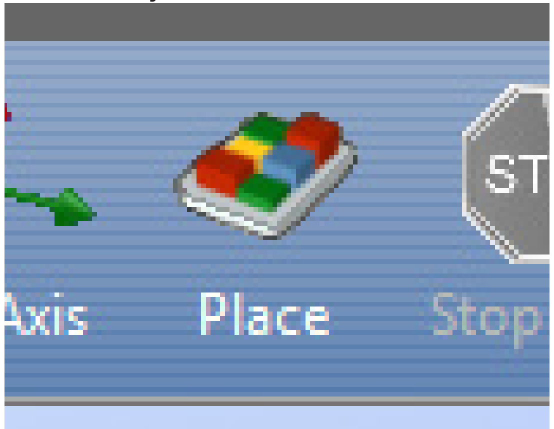


2. To reduce the object, type in 0.x then select the axis. e.g: we type in 0.1, then the object is 10 times smaller.



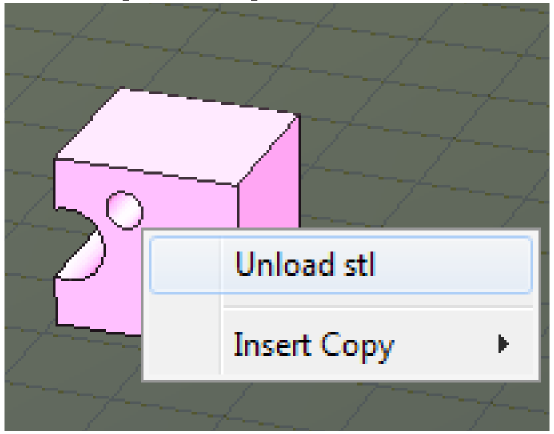
Placement of objects

Always click [Place] before printing to ensure objects are on the base.



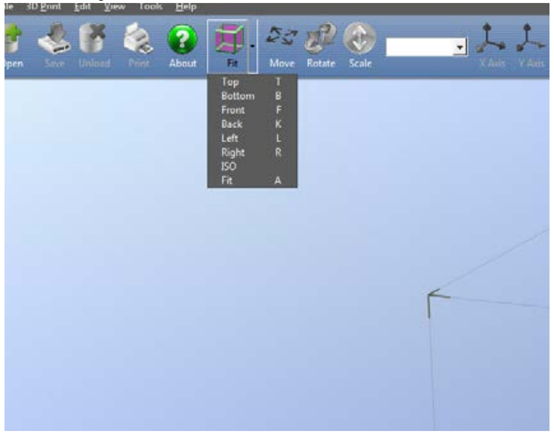
How to remove objects?

Right click on the obeject, then click “unload [filename]”.

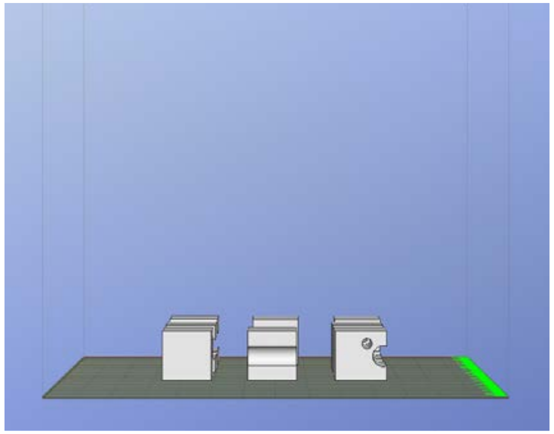


Observing placement

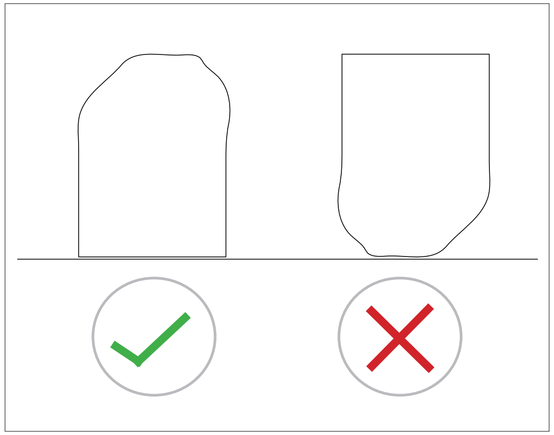
Hover over the arrow next to “fit” to check your model from different views.



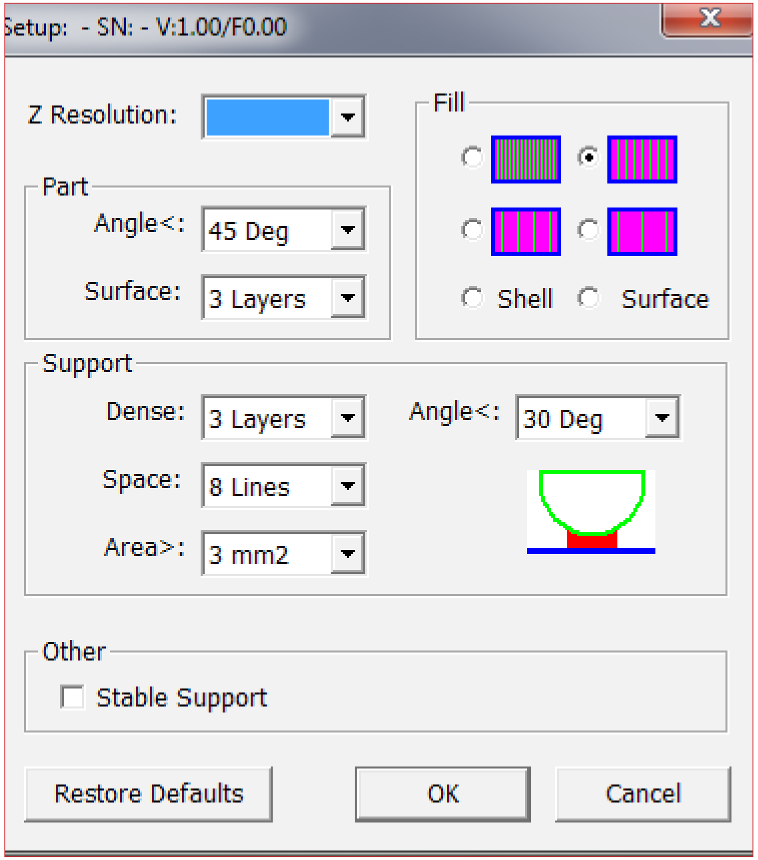
We suggest you to go to sideviews to make sure that the objects are above the plane.



If you want to get the print done **faster**, we also suggest the bottom of the object to be as flat as possible.

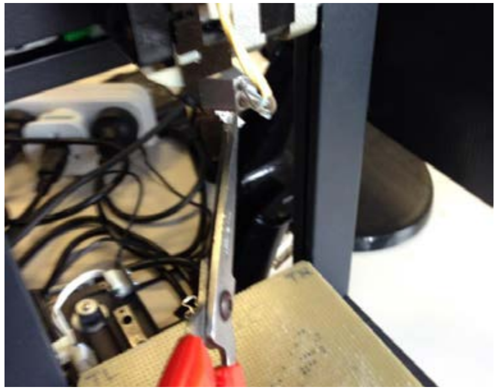


FINAL SETUP FOR PRINTING



Once the above steps have been completed you will be ready to print, click “printer setup” to get this window. Click “OK” to start printing. We suggest using the default settings to print.

Filament and Nozzle



Cleaning the Nozzle
Make sure the nozzle is cleared of any excess plastic build up before and after printing. Be careful though as the nozzle is extremely hot during and after printing and can reach temperatures of over 200 degrees Celsius. Do not remove excess plastic on the nozzle heads with your bear hands, use a tool instead and gently remove excess plastic.

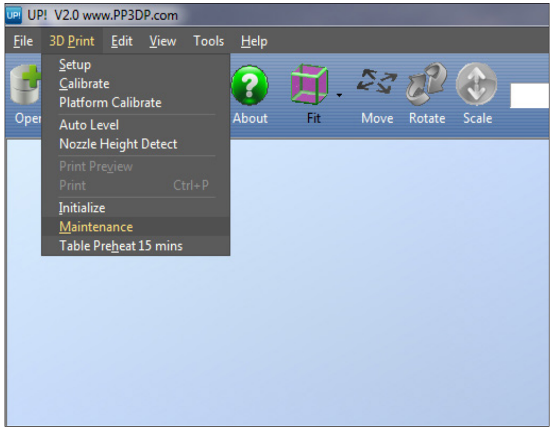
NOTHING WILL COME OUT IF THE NOZZLE IS BLOCKED

Replacing Filament
To replace the filament, you first need to detach the guide tube and feed the new filament through it. Make sure the new filament is not bent or the filament may snap and the printer will cease extruding plastic. After feeding the new filament through the guide tube, feed the filament into the extruder.
NOTE: The spool of filament should always feed from the bottom of the spool toward the top.

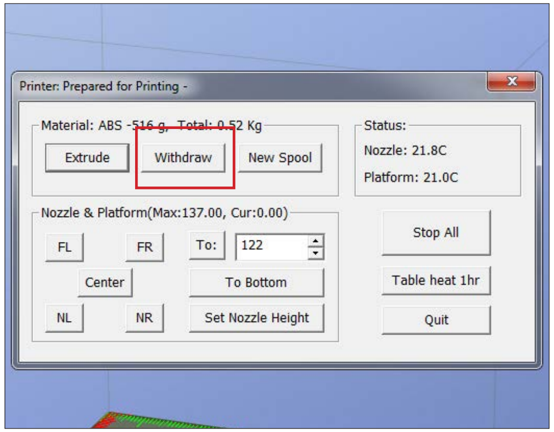
Once you have changed the filament on the printer, follow the appropriate filament replacement steps which can be accessed on the printer software (or on the printers physical user interface). This will usually involve preheating of the extruder and then manually feeding the filament further into the extruder in order for the extruder motor to catch the filament. You can stop the process once the extruder has reached the appropriate temperature and melted plastic is being extruded from the nozzle.

Guideline for filament change in Up!

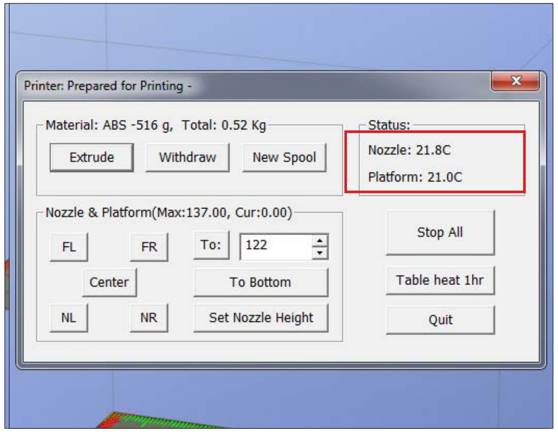
1. Click on “Maintenance”.



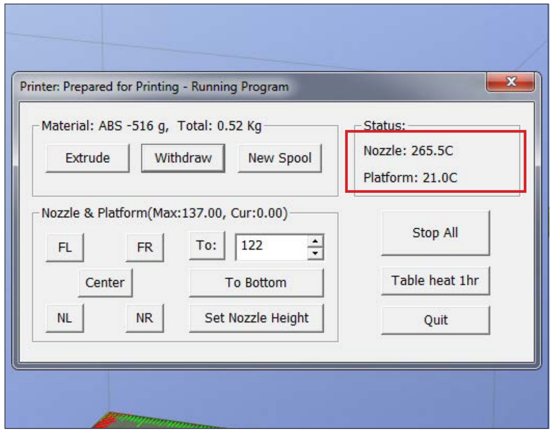
2. Click on “ Withdraw”



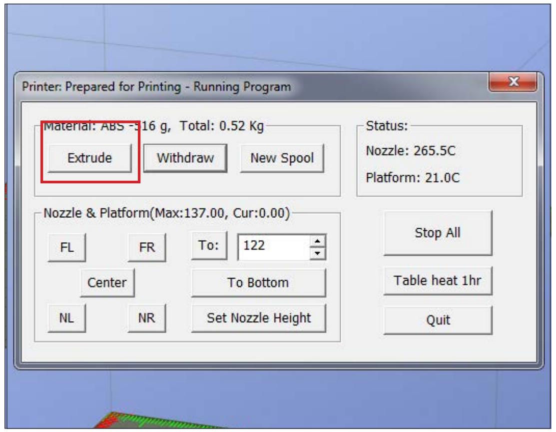
3.Pay attention to the nozzle temperature.



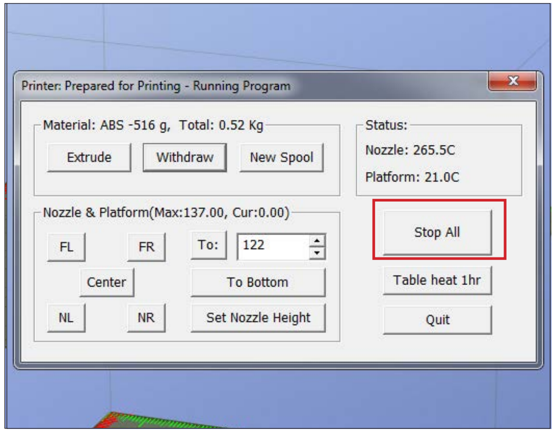
4. Once it reaches 265 °C, you can withdraw the left over filament from the machine.



5.Load the new filament and click extrude.

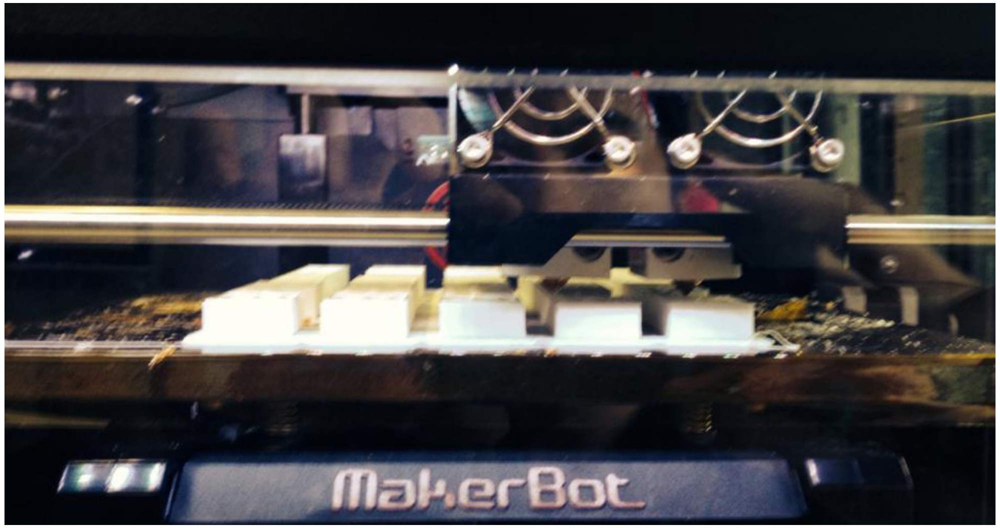


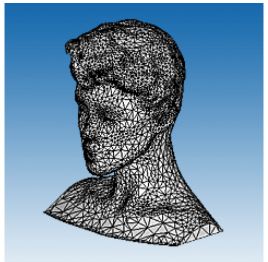
5. Onece the filament begins extruding from the nozzle click “Stop All” to complete the filament changing process.



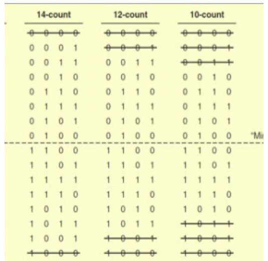
WACTH OUT FOR YOUR MODEL WHILE IT'S BEING PRINTED!

Well done! You finally got your model to print. But you need to check the printer frequently to make sure it is printing correctly. The printer's nozzle can become blocked sometimes which will need to be cleaned. You will then have to reset your print in this case. If your print fails and there are no detectable problems with the printer, remove the failed print and try reprinting. If this does not work, there may be a problem with your file which will need fixing.



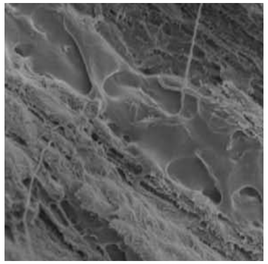


STL
An STL is a widely used type of 3D model file. It consists of surfaces made up of triangles. Each triangle has an inner side and an outer side. The outer side is called the normal. In a well-formed STL, all the normals face outwards and the surface is continuous, with no holes. When a model meets these standards, it is referred to as manifold. STLs with normals that face inwards (inverted normals) may be printable, but a manifold model is considered a prerequisite for 3D printing.
STLs are compatible with many different 3D modeling programs and have become the standard file type for 3D printable models. SolidWorks®, Rhinoceros®, and most Autodesk® programs will export STL files, and there are free plugins available that will allow you to export STLs from SketchUp®.



GCode
GCode is a computer language for controlling CNC machines, including many 3D printers. When your slicing software turns your 3D model into a set of instructions for your 3D printert, those instructions are written in GCode. The instructions consist of commands that tell the extruders how hot to get, where to move and when to start extruding plastic, commands that control the build platform and commands for peripheral components.
Before being sent to the 3D printer, the 3D printing software converts the human-readable GCode to the more compact, computer readable X3G. You can't edit X3G files, but you can edit GCode to do things like change extrusion temperatures and insert extra commands.

ABS
ABS is short for Acrylonitrile Butadiene Styrene. It is made of a combination of those three plastics. The three plastics can be mixed in different proportions to formulate ABS intended for different uses. ABS is tough and somewhat flexible. ABS becomes softer with increased temperatures, but at the extrusion temperatures used in 3D printers remains fairly viscous. That means that ABS melts quickly inside the extruder but doesn't drip during travel moves. ABS can also withstand heat well enough that we use it to make the plastic components of the printers extruders.



PLA
PLA, or Polylactic Acid, is a biodegradable plastic with features that make it great for 3D printing -- it doesn't give off bad-smelling fumes and it has a low rate of thermal expansion, so it doesn't warp very much. PLA is harder and slightly brittle. It is more likely to snap than bend, but that does not mean that PLA is easily breakable. PLA also stays flexible for a short while as it cools.