

Important steps:

- 1 Get your file checked by staff
- 2 Complete 3D Print Request Form
- 3 Take completed form to Cashier's Office
- 4 Pay, and request **RECEIPT**
- 5 Bring Receipt back to 3D Printing Lab

Before you print

Pay-per-print:	Submit your file for pricing
BYOF:	Free if using your own <u>Makerbot</u> filament
Have in hand:	Exported STL file Original model file (.3DM etc...)
Bring your own:	SD card (2GB) 2" Painter's tape

Cashier's Office:
72 5th Avenue 2nd Floor
Pay cash or credit

You can also purchase 2GB SD card at 3D printing Lab and pay cashier's office

Recommended Settings

Export Guide:

File -> Export to X3G

Material: Makerbot PLA (1.75mm) -> **Do not use third party filament!**

Resolution: The three presets are a trade-off between quality and print time
Ninjabflex Semi-flex can be brought in by appointment only

Raft & Support:

Raft : lays a crosshatch down below model for better attachment to build platform.

Support : Generates one shell thick scaffolding under your overhangs (can add a significant amount of time to your print, but crucial for overhanging ledges of 45 degrees or more).

Default Advanced Options:

Low Setting	Standard Setting	Advanced Setting
Infill: 10%	Infill: 10%	Infill: 15%
Number of Shells: 2	Number of Shells: 2	Number of Shells: 2
Layer Height: 0.30mm	Layer Height: 0.20mm	Layer Height: 0.10mm

Specifications:

Build Volume: 11.2" X 6.0" X 6.1"
(28.5cm L X 15.3cm W X 15.5cm)

Layer Resolution: 100 Microns (0.0039")

Printing with Makerbot

Before using a Makerbot, please sign in at the front desk.
This helps us keep track of their health. Healthy bots are happy bots.

- 1 **Open** your STL file in MakerWare software
- 2 **Move** object to platform
- 3 **Rotate** your objects to minimize overhanging ledges. Repeat step 2 if necessary
- 4 **Check** that the printer selected is Replicator 2. If it isn't, go to **Devices** -> **Select Type of Device** -> **Replicator 2**
- 5 **File** -> **Export X3G** (refer to "Export Settings" for guidance)
- 6 **Copy** the file to your SD Card (2GB is max card size)
- 7 **Power on** Makerbot (switch on the rear) and **insert card** into slot (above the screen)
- 8 **Change Filament**
Utilities -> Change Filament -> Unload
*Wait for extruder to heat up
*Remove old PLA spool
*Place new PLA spool on holder in rear (make sure it moves well)
*Cut a clean tip on the end of the plastic, feed up through clear guide tube, while keeping tension on the spool so it doesn't uncoil
Utilities -> Change Filament -> Load
*Gently feed PLA into top hole of extruder, until you feel the motor feed in by itself
*Make sure the plastic feeds out at a steady rate and all the old plastic has fed out
- 9 **Level Build Plate**
Utilities -> Level Build Plate
*Ask a staff member for a levelling card
*Follow the instructions on the makerbot screen and loosen/tighten the 3 thumbscrews under the platform
*Test the height by sliding the card between the nozzle and the platform. There should be just enough friction to feel the nozzle just slightly slow down the card movement. Too much friction means the nozzle is too close and you must tighten the screw (twist to the right)
- 10 **Print your file**
Select "Build from SD" from the main menu
Select your file and press the 'M' button
- 11 **Watch** your print to see that the first few layers print well
Most print failures happen in the beginning of the print
- 12 **Remove** your part from the platform
Clean up after yourself- leave area nicer than you found it
- 13 **Report** - If you notice any problems, report them to staff and record on your sign-in sheet

Printing with Ultimaker

Before using an Ultimaker, please sign in and ask for assistance at the front desk. This helps us keep track of their health. Healthy bots are happy bots.

1 Open your STL file in Cura software (<http://software.ultimaker.com/>)
Make sure the printer model is set to "Ultimaker 2"

2 Refrain from applying blue tape to the build plate since it is a heated bed

3 Rotate your objects to minimize overhanging ledges. Make sure the object remains on the bed

4 Save to Disk -> Export .gcode (refer to "Export Settings" for guidance)

5 Copy the file to your SD Card (**any size**)

6 Power on Ultimaker (switch on the rear) and **insert card** into card slot

7 Change Filament

Remove old filament

*Access the material change menu by going to Material -> Change

*Wait for print head to heat up

*Ultimaker will automatically start turning the feeder wheel (through which the filament will rewind completely. If the filament doesn't come out of the feeder you can pull it out by hand.

Load new filament

*Make sure the old spool of filament is removed from the spool holder and the new one is placed

*Cut a clean tip on the end of the new filament.

*Take the end of the new filament and insert it into the bottom of the material feeder. Push it until the filament is grabbed by the "knurled wheel" (this may require some force). Once the knurled wheel has grabbed the filament, it will slowly feed up the guide tube. Wait for the filament to reach the first black guide tube clip and then press the button to continue.

*The Ultimaker will automatically load the filament through to the print head.

*Filament should begin to extrude through the nozzle.

8 Level Build Plate

Ask a staff member to demonstrate BEFORE attempting yourself.

Maintenance -> Build Plate -> Continue

*Roughly level the build plate by rotating the button at the front of the Ultimaker until there is approximately 1 mm distance between the nozzle and the build plate (make sure the nozzle is close to the build plate but not touching it). Press "continue".

*Perform the same adjustment for the front, left, and right side by turning the build plate screws. Turning the build plate screws to the left raises the build plate closer to the nozzle. To fine-tune the build plate calibration get a regular sheet of A4 paper. Slide the sheet between the nozzle and build plate at each test point, adjusting the build plate screws until you feel slight friction when moving the piece of paper. After each adjustment, press "continue". When complete, press "done".



- 9 Print your file** by selecting "Print" from the main menu. Select your file using the scroll wheel, then pushing in the wheel to run.
- 10 Watch** your print to see that the first few layers print well. Most print failures happen in the beginning of the print.
- 11 Remove** your part from the platform
Clean up after yourself. Leave area nicer than you found it!
- 12 Report** - If you notice any problems, report them to staff and record on your sign-in sheet

Recommended Settings

Export Guide:

Save to Disk -> .gcode

Export for Ultimaker 2

Material: PLA & ABS plastic 2.85 mm

Resolution: The three presets are a trade-off between quality and speed

Raft & Support:

Brim : lays a single-layer-thick flat area around model for better attachment to build platform. It is easy to cut off after printing and is the recommended option.

Raft : lays a crosshatch down below model for better attachment to build platform.

Support : Generates one shell thick scaffolding under your overhangs (can add a significant amount of time to your print, but crucial for overhanging ledges of 45 degrees or more).

Default Advanced Options:

Low Setting

Infill: TBD based on model

Shell Thickness: .8mm

Layer Height: 0.20mm

Standard Setting

Infill: TBD based on model

Shell Thickness: .8mm

Layer Height: 0.10mm

Advanced Setting

Infill: TBD based on model

Shell Thickness: .8mm

Layer Height: 0.06mm

Specifications:

Build Volume: 12" X 8" X 8"

Layer Resolution: Up to 20 micron (.001mm)