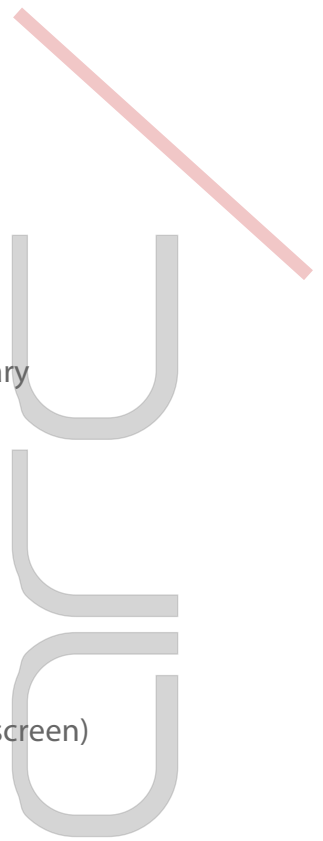


# Printing with Makerbot

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



# 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 significant amount of time to your print, but crucial for overhanging ledges of 45 degrees or more).

## Default Advanced Options:

Low Setting

Infill: 10%

Number of Shells: 2

Layer Height: 0.30mm

Standard Setting

Infill: 10%

Number of Shells: 2

Layer Height: 0.20mm

Advanced Setting

Infill: 15%

Number of Shells: 2

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")**