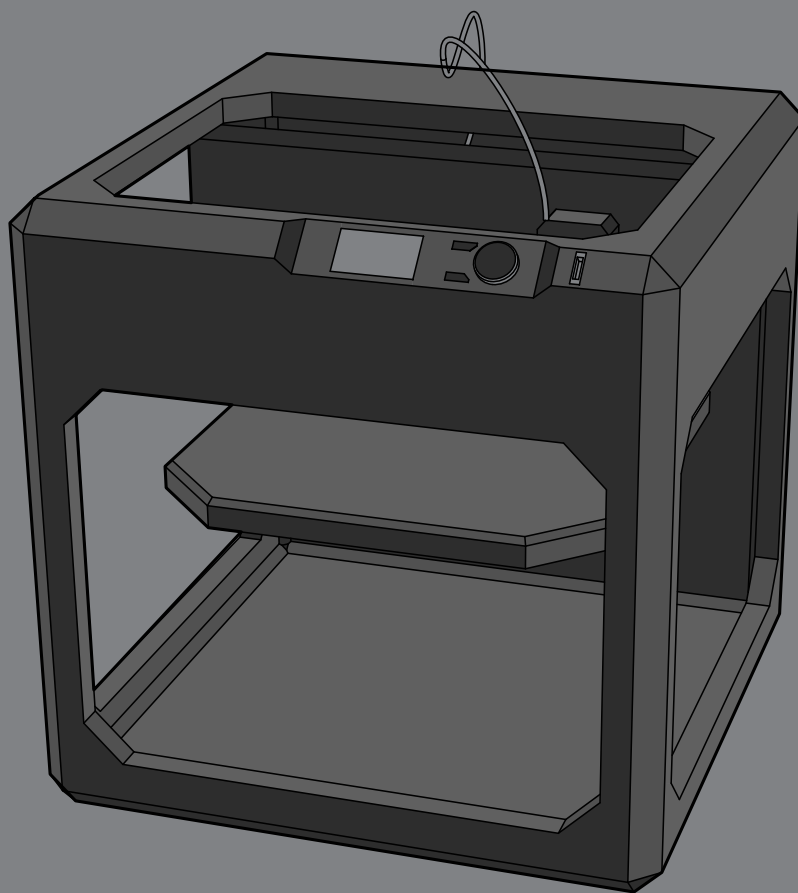# UMass Amherst Digital Fabrication Lab
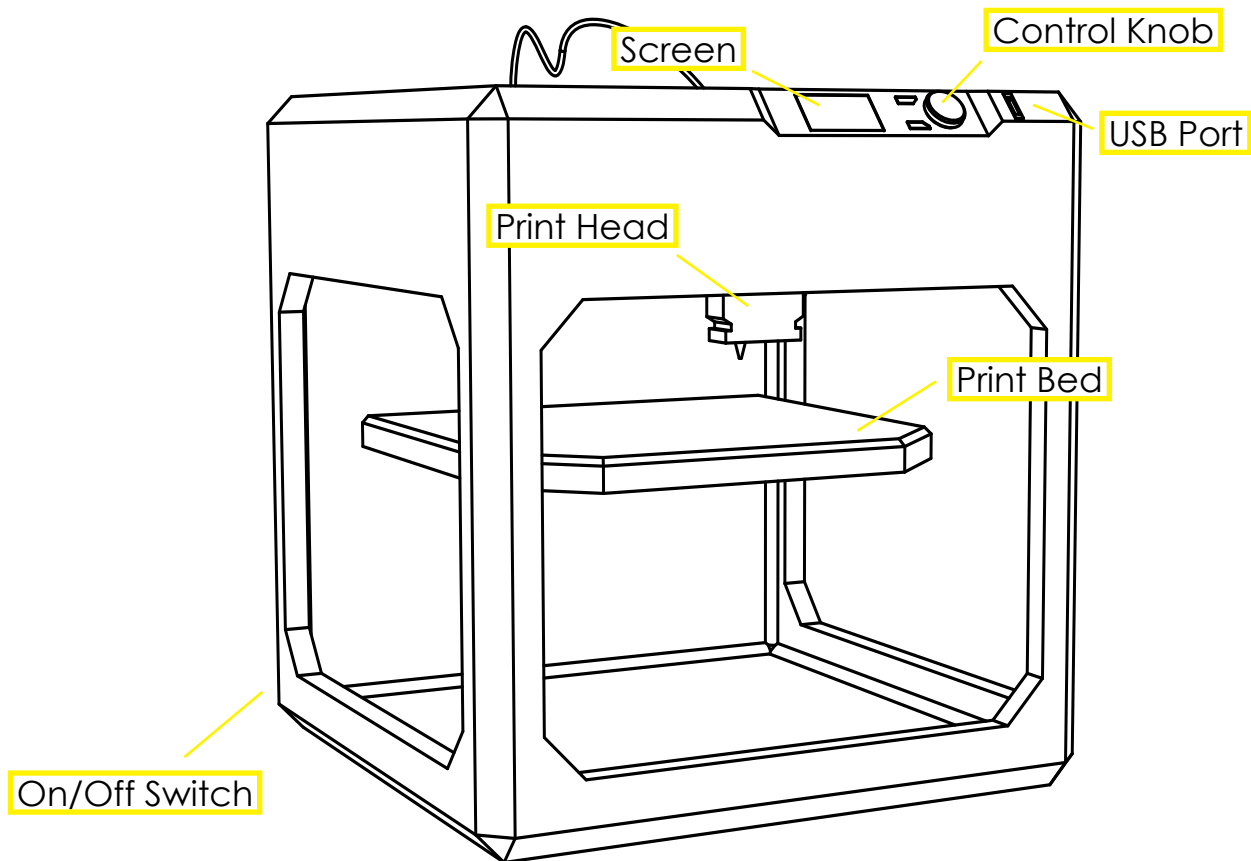## MakerBot Replicator+ 3D Printer

John Olver Design Building
Spring 2022

# Makerbot Replicator+: 3D Printer Summary

This machine uses a heating element to melt various types of plastic compounds and stack it to build up three-dimensional objects. The print head moves laterally, and the print bed moves vertically.

This machine can print using a few different material choices, however in the Fablab we only supoort its use with PLA (polyactic acid). PLA is easy to print with and is a default material with the machine which simplifies file setup and use, PLA is good for prototyping and so it is relatively fast and light but not very strong. For more functional printed parts, you would consider using the Stratysys printer, which prints with ABS plastic.

Screen

Control Knob

USB Port

Print Head

Print Bed

On/Off Switch

Makerbot Replicator+: 3D Printer Summary

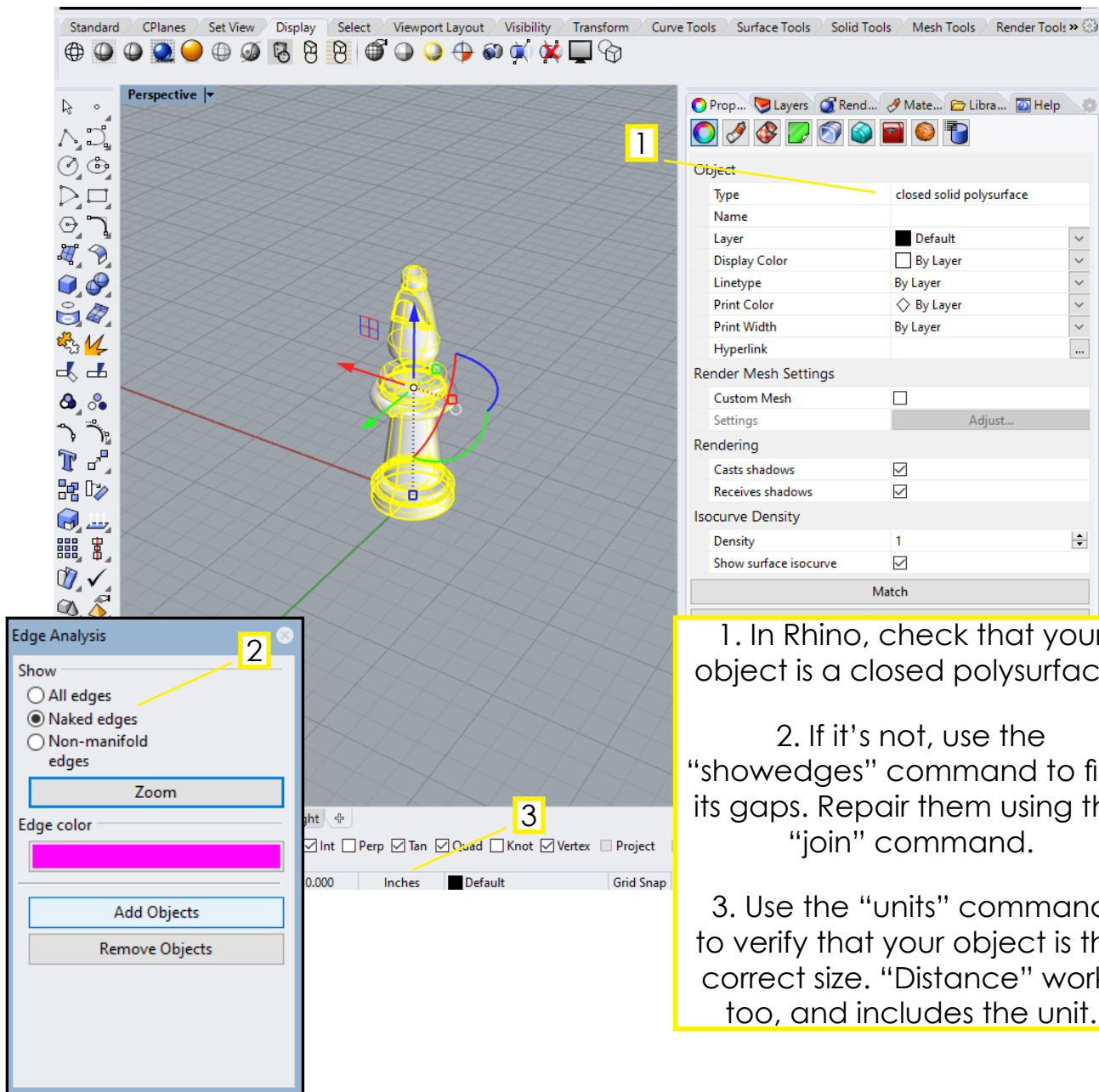Use the scroll wheel to navigate the Makerbot's menus. The power switch is on the back of the machine.

Scroll Wheel

Button A

Display

PRINTING

22%

00:04:08
ELAPSED

00:59:35
REMAINING

Button B

G 2
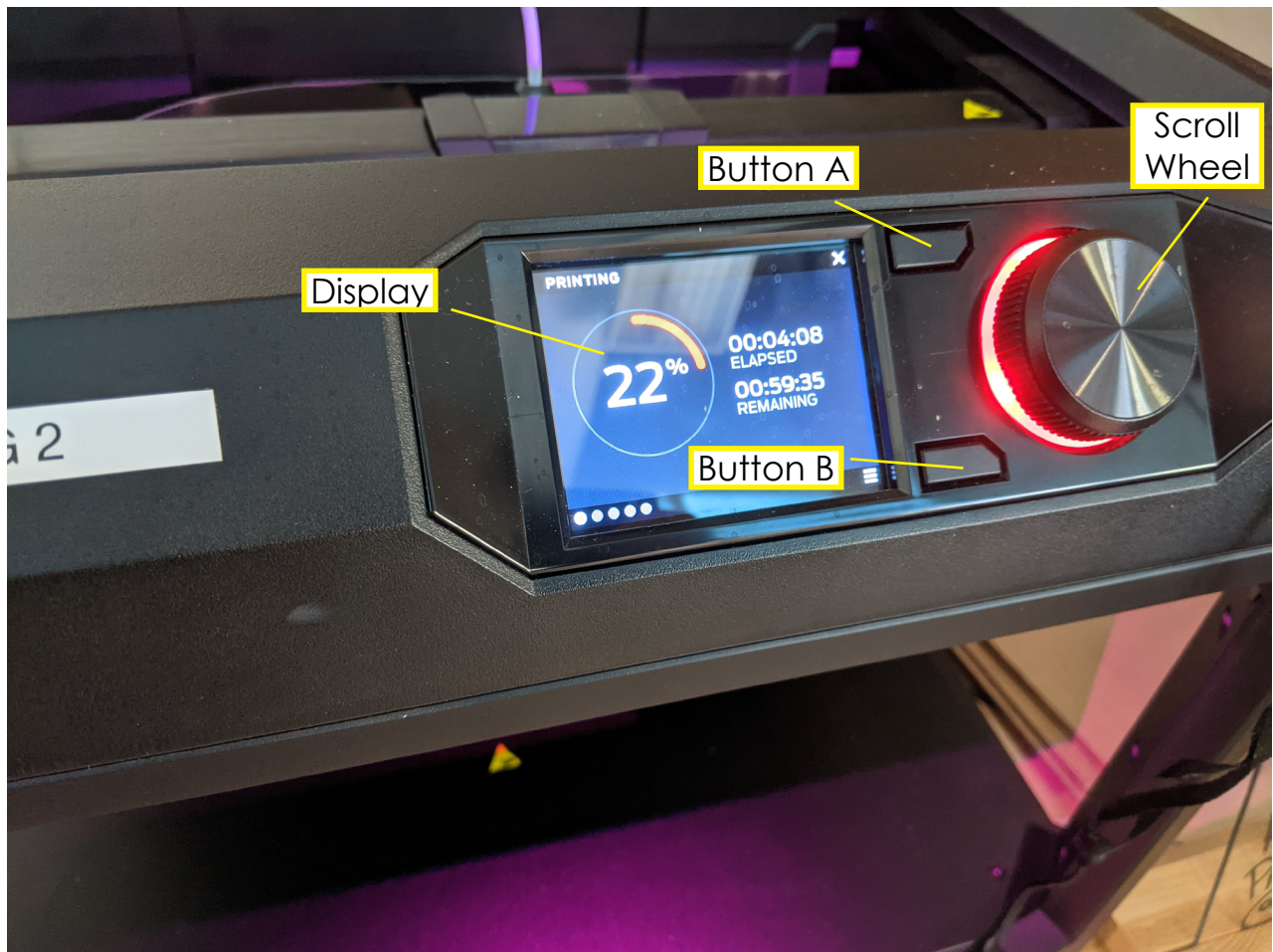
Button A and B are used for opening up, closing and navigating to different menus available on the Makerbot.

Makerbot Replicator+: Printing

STEP ONE: Formatting your 3D Model

These instructions will describe exporting from Rhino into Makerbot's software. Other softwares are also capable of modeling and exporting. Your objects should be appropriately sized, solid, and closed. The maximum size for a print on the Makerbot is 11.6 x 7.7 x 6.5 - your print probably shouldn't come anywhere close to this in volume. There can be no open surfaces with zero thickness. Closed meshes will also be acceptable.


1. In Rhino, check that your object is a closed polysurface.

2. If it's not, use the "showedges" command to find its gaps. Repair them using the "join" command.

3. Use the "units" command to verify that your object is the correct size. "Distance" works too, and includes the unit.

Makerbot Replicator+: Printing

STEP TWO: Exporting your 3D Model

Consider the orientation of your object and how the 3D printer works, there should be as little overhanging material as possible. The 3D printer can navigate overhangs and arches by inserting breakable support material, but your print will go faster if you minimize the amount of support material. The printer works by building up layers while moving vertically, picture the printer trying to lay down a bead of melting plastic in mid-air without any support below it.
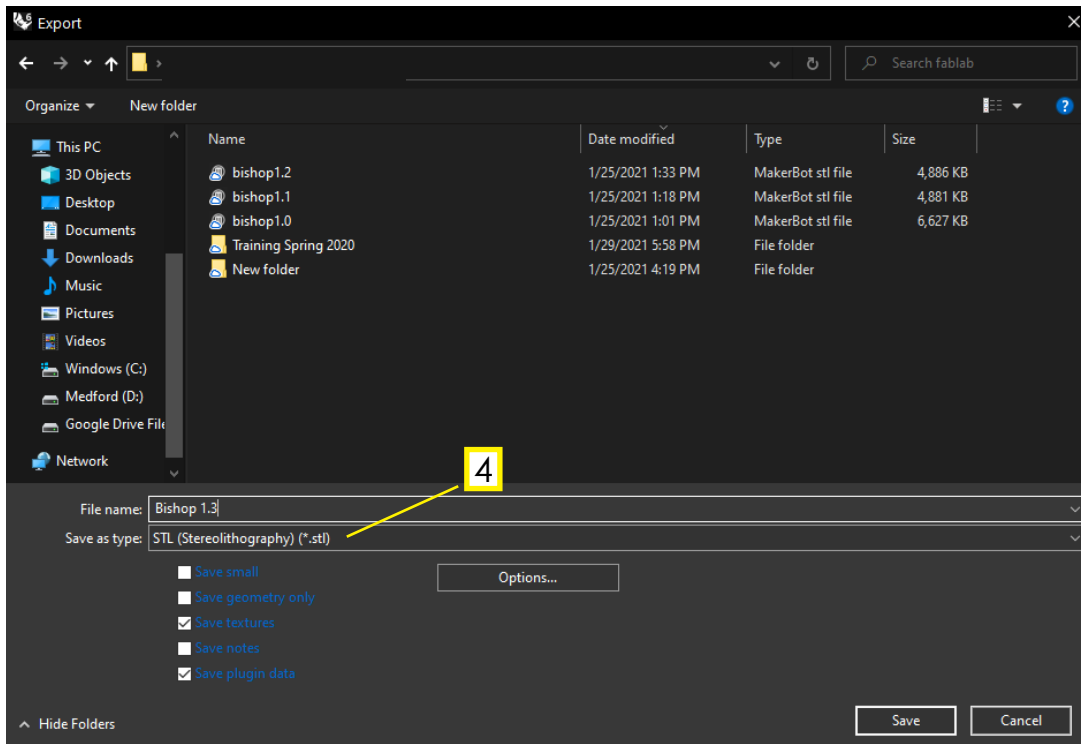
4. Export your object as a .STL (stereolithography) file.
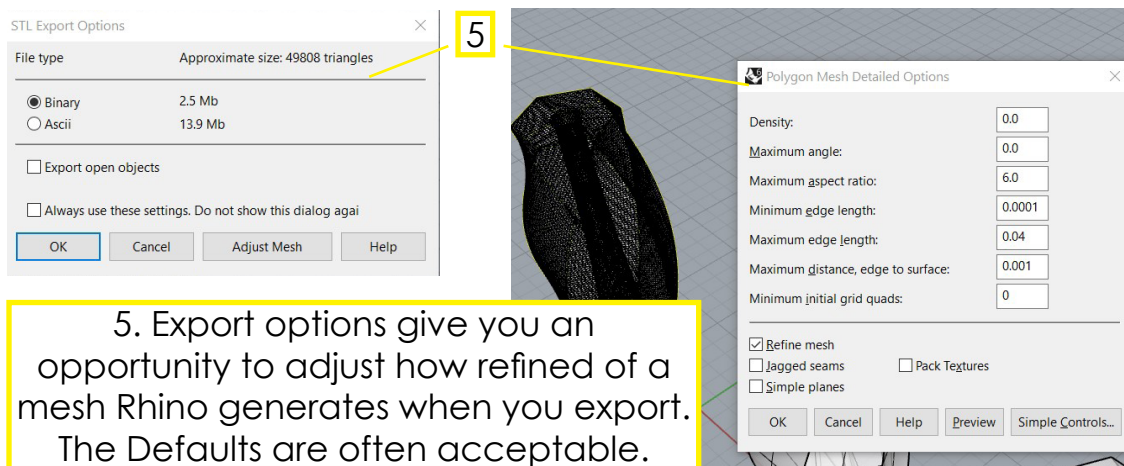
5. Export options give you an opportunity to adjust how refined of a mesh Rhino generates when you export. The Defaults are often acceptable.

Makerbot Replicator+: File Setup

STEP THREE: Defining Print Settings

Makerbot's "MakerBot Print" software will take you through formatting a print and exporting it to the printer. It allows you to modify your settings for higher and lower quality, and faster or slower print times. Sometimes you'll realize at this step that your model won't print well, and you should go back to Rhino.
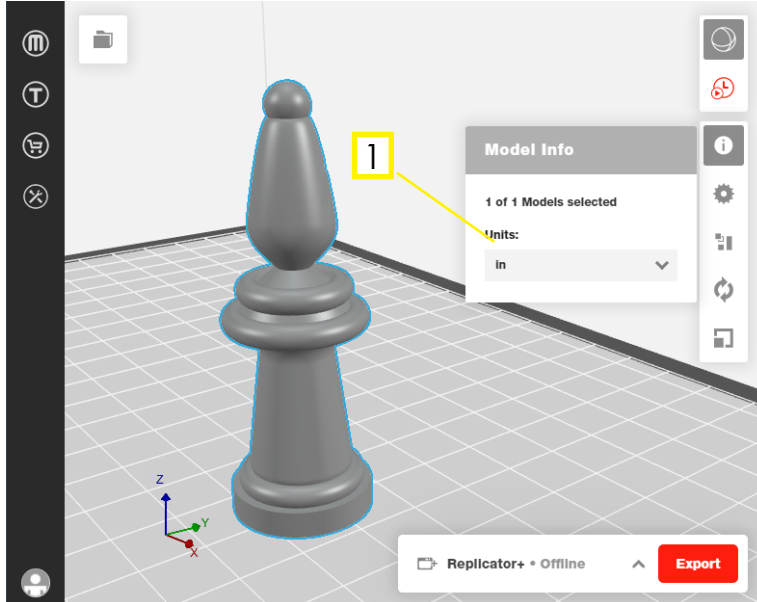
Open the Makerbot Print software. Go to File>Insert File, and select your .stl export.

1. First, verify that the unit of the makerbot file matches that of your Rhino file: Makerbot's default unit is millimeters so you may need to adjust this to correctly scale your object.
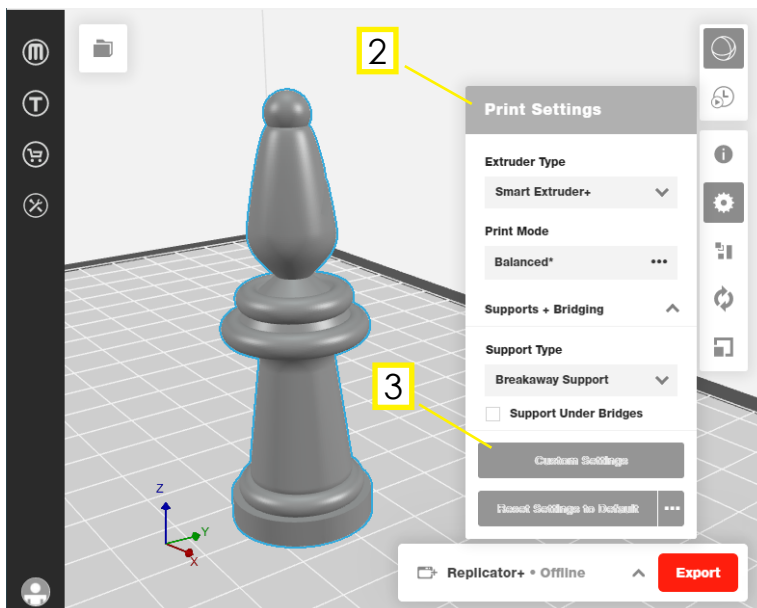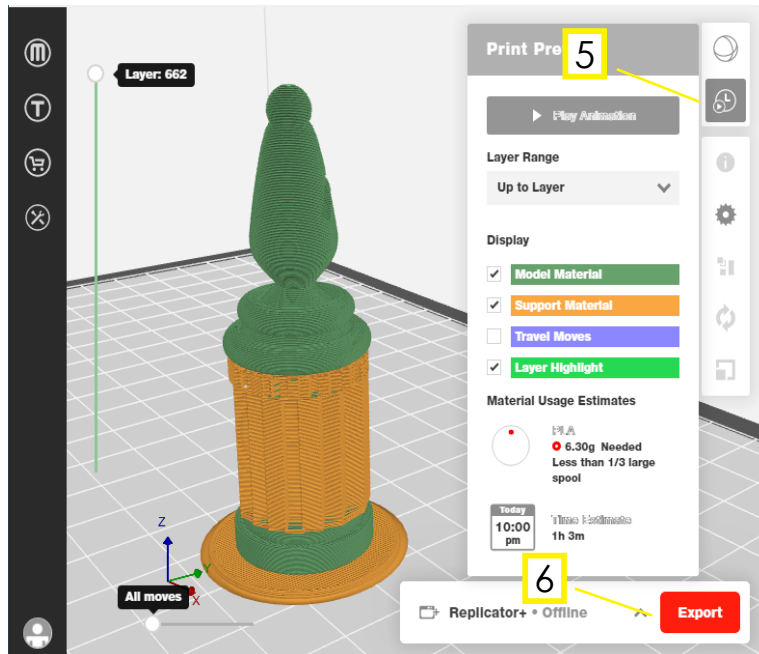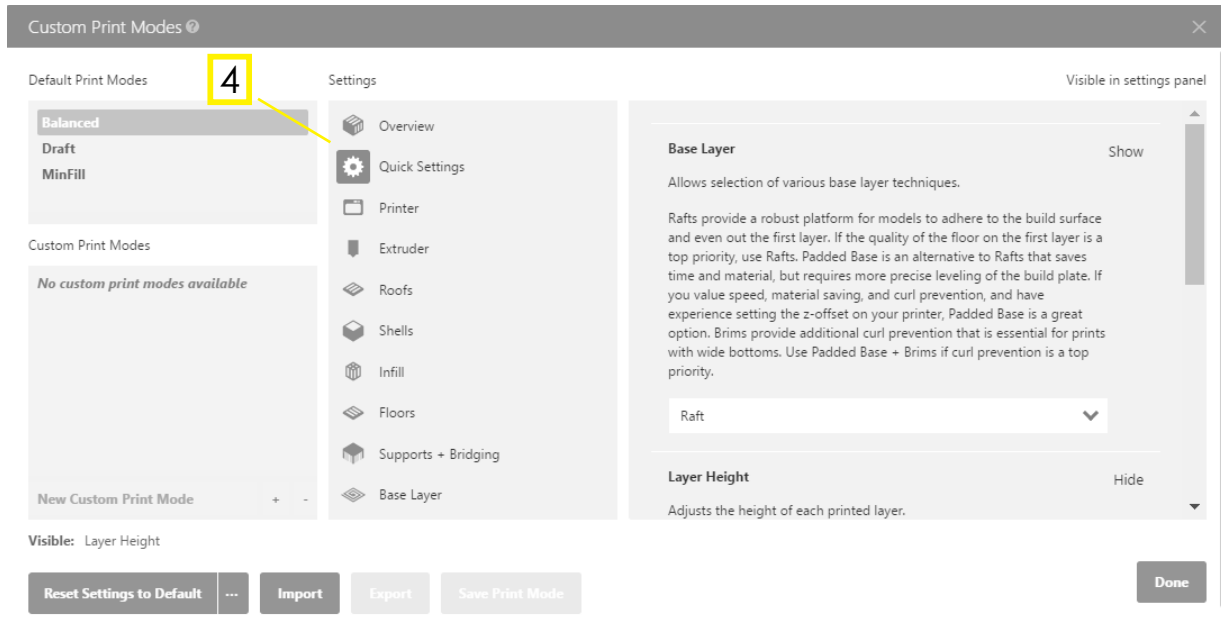

2. Next, check the print settings. Generally, Makerbot's presets work adequately. Check that "breakaway support" is selected under "Support Type".

3. If your print isn't working correctly, you'll have to adjust settings under "Custom Settings".

Makerbot Replicator+: File Setup

STEP FOUR: Exporting your Model

4. The Custom Settings Menu is where you can adjust support, density, infill and more.
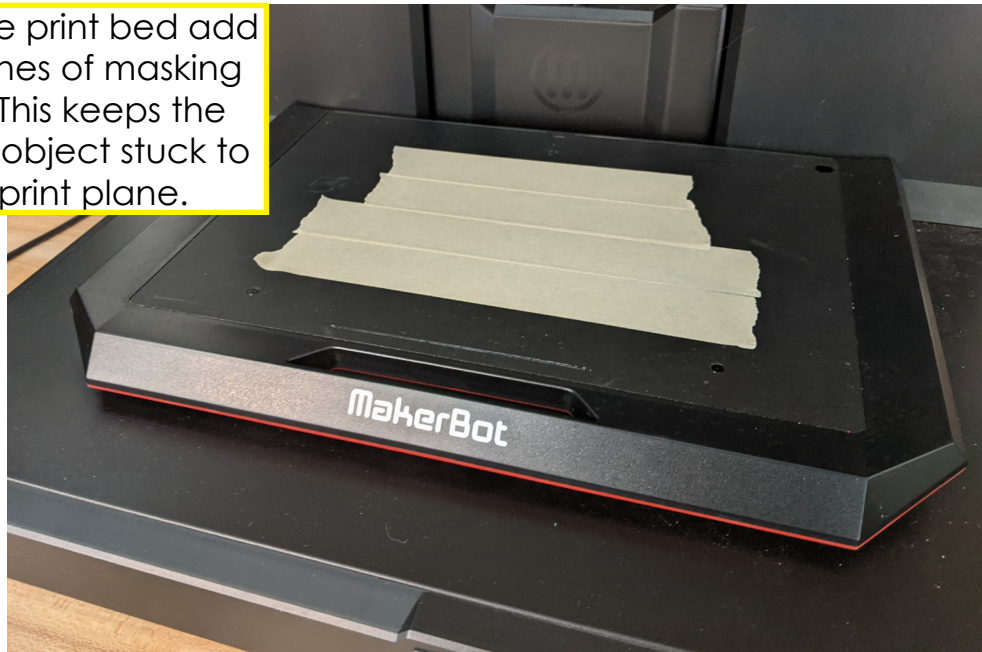
5. Click the preview icon to make the software generate an estimate of your file's geometry, weight, and machine time.

6. Press the "Export" button, and save to an external flash drive.

Makerbot Replicator+: Printing
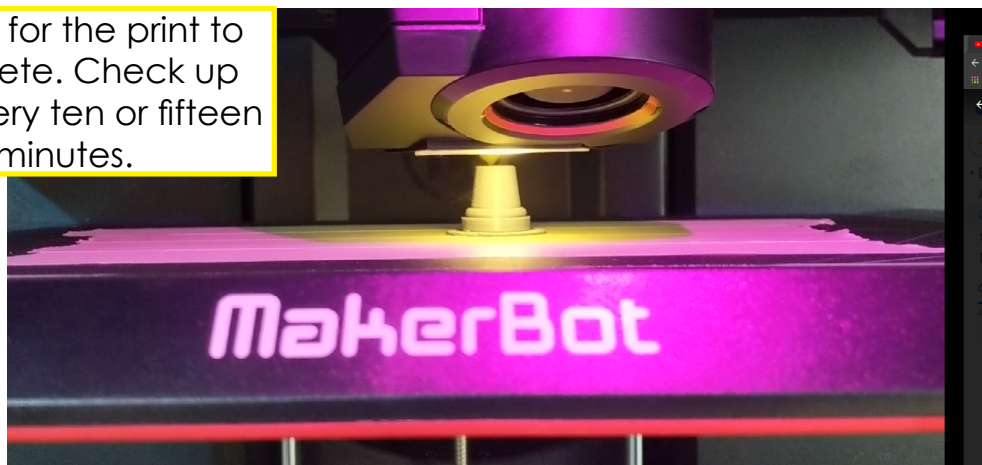
STEP FIVE: Printing the Object

1. On the print bed add some lines of masking tape. This keeps the printed object stuck to the print plane.

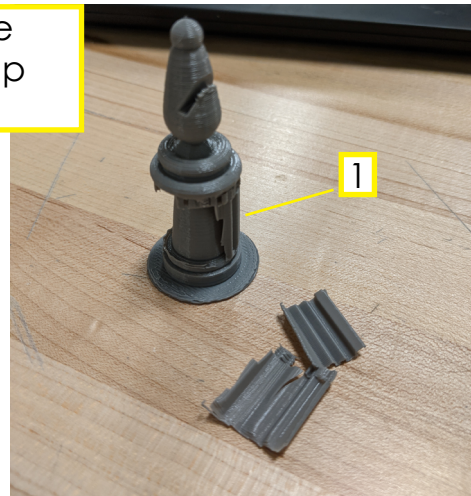2. Insert the USB drive with your file, and press "go". Navigate with the scroll wheel.

3. Wait for the print to complete. Check up on it every ten or fifteen minutes.

Makerbot Replicator+: Printing

STEP SIX: Remove Support Material

1. Remove the print from the Makerbot Print bed, and snap away the support material.

2. Snap away the "raft" underneath the object. Sometimes a thin putty knife can help.

Your print is now finished and ready to take. Be sure to properly dispose of the support and turn of the machine

Makerbot Replicator+: Troubleshooting

Problem:	Cause:
In Makerbot Print software, all of the menus are greyed out.	Object isn't selected- the software isn't sure which object to apply changes to.
Print fails. Strings of filament fall to the print bed.	Object doesn't have adequate support material.
Print Fails. Object becomes dislodged from the print bed and stuck to the print nozzle.	Object became dislodged from the print bed and stuck to the nozzle.
Print fails for unclear reasons.	

Makerbot Replicator+: Troubleshooting

Solution:

Select your object by clicking on it before editing menu items. It should become outlined in blue.

Check that "breakaway support" is enabled under the Gear icon in MakerBot Print. Specify more support material under Print settings.

Speficy new "raft settings" under Print settings. Ensure that there is a bed of masking tape on the Makerbot print bed. Try moving the object to another area of the print bed in the Makerbot Print software. Consider resetting the "z" zero.

Try these raft settings out:
  Filament cooling, First Model Layer, %80
  Print speed, First Model Later, 20 mm/s
  Filament cooling, Raft Base, %40
  Print Speed, Raft Base, 15mm/s
  Raft to model vertical offset, 0.25mm
  Raft size, 5mm

Adjust settings under "custom settings". Be scientific- only change one or two factors per print, so that you can see how your changes affect printing. Talk to a lab monitor or TA.

Some to try:

Changing resolution by adjusting layer height. No smaller than 0.1mm, or larger than 0.3 mm. Smaller layers will be heigher resolution, but will take longer.

Changing infill density or pattern. Higher than %50 density can cause heat buildup and warming.

Number of shells- adjust the thickness of the outer layer, bounding the infill. More shells makes a tougher part.