Within this module, you will be introduced to basic rules and functions commonly used while 3D Printing. This module will ensure your ability to assemble a functioning 3D print file, and geometry for use in the CAP Fabrication Lab or a personal printer, as well as handcraft to digital model conversion methods to further enhance your abilities to interface with CNC equipment. This module's objectives are to expand your workflow capabilities, introduce new methods of model making, and improve comfortability with 3D Printers.

A1- Intro to 3D Printing

With this assignment you will be introduced to terms and concepts that are used while 3D Printing. Understanding these terms will allow you to interface with 3D Printer settings, and slicing softwares later in this module. Terms such as : Watertight Geometry, STL, Infill %, Wall Count, Support Angle, and Layer Height, are all commonly used while preparing a file for printing. For this assignment please read and understand how each of these terms applies to a model you wish to 3D Print.

Watertight Geometry - This pertains to how your model is constructed, whether that be through Rhino, Revit, Fusion 360,Sketchup, etc. A watertight model must have all holes closed and edges joined. Think of it as a boat, you don't want any water to slip into your boat, therefore the boat must be well constructed and thoughtfully crafted to ensure there are no leaks.

.STL File Format - File formats are important, they're the language in which software such as slicers are able to interpret your information. In 3D Printing .STL files are the most commonly used, also known as "Standard Triangle Language" this file type has no units attached to it (inch, ft, cm .etc.) however it does contain a number associated to scale. It is important when saving a model as an STL to track which units you are using so you are able to make the appropriate conversion to millimeters.

Infill % - Infill is how 3D Printers turn soft plastic into a sturdy model. Within the slicer software a model will be hollowed out and replaced with a honeycomb like inner structure to give strength to the out walls of your model. Higher infill amounts will result in a sturdier model however at the cost of time and materials. Balancing this setting will help with time and material costs.

Wall Count - The wall count represents how many layers of plastic a 3D Printer will generate as the "shell" of your model. Going back to the boat analogy, the walls would represent the bottom of your boat, with more walls, the thicker your boat becomes. With fewer walls the boat may become weak or "squishy" as its shell is thinner. Balancing this setting will help with time and material costs.

Support Angle - Generating supports for a model is very common when preparing for 3D Printing. Printers are unable to print in mid air. Support angle is the angle in which a printer is forced to add supporting material to build onto. Think of a vase, as the vase becomes wider while building upwards, extra material at the base may be required in order to help the vase continue outwards and upwards.

Layer Height - A model is generated by stacking molten plastic layer by layer until a specified height and shape is met. Layer Height is the variable that allows us to control just how many layers are in our model, the lower the layer height, the more layers to print, the higher the fewer layers needed.

Within this assignment, you will be using your hands to generate a model. This assignment is left open-ended for you to fully explore what the material is capable of. Use any type of clay you wish but the model must stay within the dimensions of 6.5" x 6.5" x 1".

Assignment Objectives
- Generate a Physical Model to then be transferred to digital fabrication methods.

Final Deliverable:
-One 6.5" x 6.5" x 1" Clay model with a cardboard base.

Instructions

1. Base
Cut out a 6.5"x6.5" cardboard base. The thickness of cardboard is up to you.

2. Model Generation
For this assignment, your only goal is to have fun and use your hands to explore shapes and patterns. Use any clay at your disposal, as long as it stays within a 6.5" x 6.5" x 1" volume. You can generate anything to your heart's content. Adjacent is some examples.
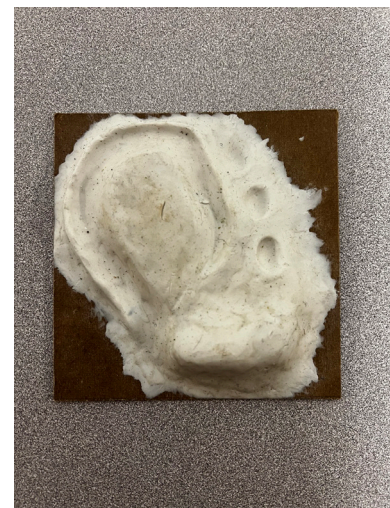
3. Labeling
Once finished please write your name and date on the back of the cardboard base.

4. Proceed to M3A2

Clay Model Example

Clay Model Example

Within this assignment, you will be using your laser-cut model from Module 2. Ensure that the model is 6.5" x 6.5" x 1" and has at least two layers of thickness to it. If you wish to make any changes to the model you may do so by hand-cutting extra details or shapes into it.

Assignment Objectives
- Modify or Assemble a Physical Model to then be transferred to digital fabrication methods.

Final Deliverable:
-One 6.5" x 6.5" x 1" Laser cut Model.

Instructions

1. Material Thickness
Ensure your model consists of at least two layers while remaining in the less than 1" range.

2. Model Modification
For this assignment, you can adjust your cardboard model by hand if you wish. Carve shapes or details into the model to further enhance your vision (Optional).

3. Model Assembly
If you have not already done so from Module 2, please glue together your layers of cardboard.

4. Labeling
Once finished, please write your name and date on the back of the cardboard base.

5. Proceed to M3A2

Lasercut Model Example

Lasercut Model Example

Within this assignmen,t you will be using good old Google to find and download quality 3D Print ready files from websites. There are plenty of libraries online that contain user-made models that are free to download.

Assignment Objectives
- Find and obtain one .STL or .OBJ file that you want to print. Also, catalog websites found for later use.
Final Deliverable:
-One downloaded .STL or .OBJ file

Instructions

1. Explore
Find and download a model that interests you, and take note of any print settings that the modeler may recommend. Use websites such as (Thingiverse, Printables, Cults3D, MyMiniFactory, etc.)
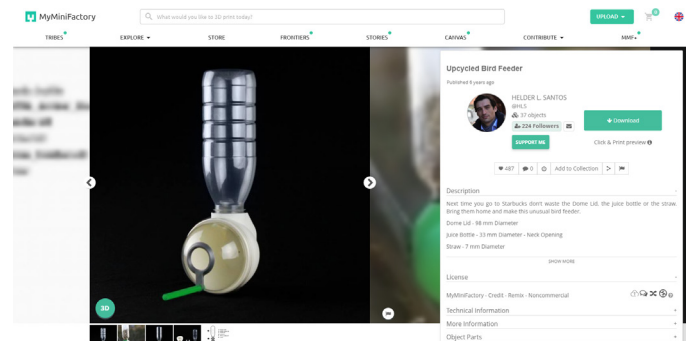
2. Model Modification
If you have "the know-how" and are up for a challenge, attempt modifying the model you downloaded...make it your own. Use software such as Mesh Mixer, Rhino, Fusion360 to engrave your name, or cut shapes out of your model (Optional).

3. Proceed to M3A3
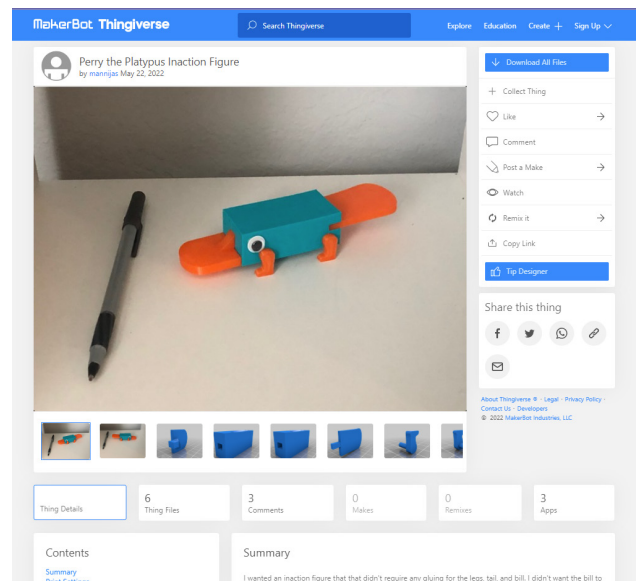
Thingiverse Model Example

MyMiniFactory Model Example

Within this assignment, you will be using 3D Modeling Software to generate your 3D Print Files. This process can be applied to any form of modeling whether it be Architectural or a simple desk organizer. For this assignment use Rhino/Sketchup/Fusion360/Revit to make your model.

Assignment Objectives
- To learn and understand modeling techniques that will allow you to bring your designs from a computer to reality.

Final Deliverable:
-One .STL or .OBJ
-One screenshot of a model in the software

Instructions

1. Select an appropriate 3D Modeling Software, Revit/Rhino/Sketchup/Fusion360 are all options. Sketchup and Fusion360 are free for at least 30 days. Also, Rhino is available for free in the Architecture Library with student login.

2. If in Rhino, download our CAP Fab Lab scaling file from our website, this will be useful for testing scale in the future. Please size your model to fit within a 4" x 4" x 4" cube regardless of the software chosen.

3. When generating your model. Please keep in mind, the CAP Fab Lab guidelines require that your model should be
   -Water Tight (Closed)
    -Avoid thin walls or pipes
    -Avoid large 90-degree overhangs (45 degrees or less can be printed without supports)
    -Scaled to fit within a 4" cube
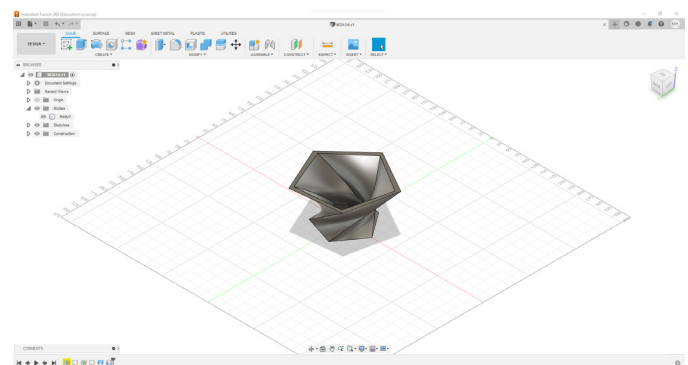If you have any other concerns, please refer to the 3D Printing Guidelines on the CAP Fab Lab website.

4. Export as an .STL or .OBJ and then select Binary (if asked). Once completed, take a screenshot of your model in the software you selected.

Link to Scaling File/Guidelines
https://sites.bsu.edu/capfablab/fdm-3d-print-request/
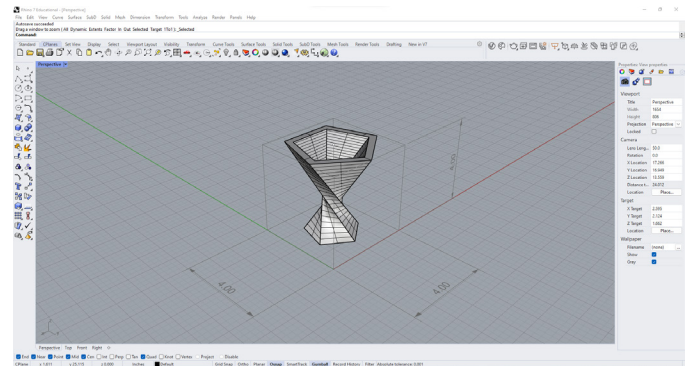
Rhino Model Example

Fusion 360 Example

Within this module, you will be introduced to a new form of analog-to-digital conversion. 3D Scanning, using a dedicated 3D scanner allows you to translate a physical object into an editable 3D model. Using a 3D scanner also allows us to translate texture data with our model, allowing for hand-drawn or other textures to be translated without the use of Photoshop or modeling software. In this assignment, we will be using the scanner to translate either a laser cut or clay model into a digital format.

Final Deliverables:
One .STL or .OBJ file

Instructions

1. Installing EX-Scan
Download Exscan. Click next until the icon appears on your desktop. Then plug Scanner USB into your computer. Review an SOP developed by Jim Flowers on our website: 3D Printing > A2 > attachments.

2. Setup
Open the Scanning software and select EinScan-SE, if prompted to activate the online license, do so. Follow the steps to calibrate the machine using the dotted grid and white paper, follow the steps on the screen. If the progress bar gets stuck at 80% for more than a few minutes, remove the dotted grid from the scanner view. It should proceed after that.

3. Scanning
Select a new project, and place your clay or laser-cut model roughly in the center of the turntable. Adjust brightness to where there is little to no red in the background, then select "begin scan."

4. Full Scan
Once one scan is complete you may notice it has not scanned your entire object (usually top or bottom of it), flip the model onto its side or upside down. Then select begin scan.

5. Repeat until happy with results, then select global optimization, then apply the edit. Finally save your model as a watertight, high-quality .STL or .OBJ

6. Proceed to M3A3
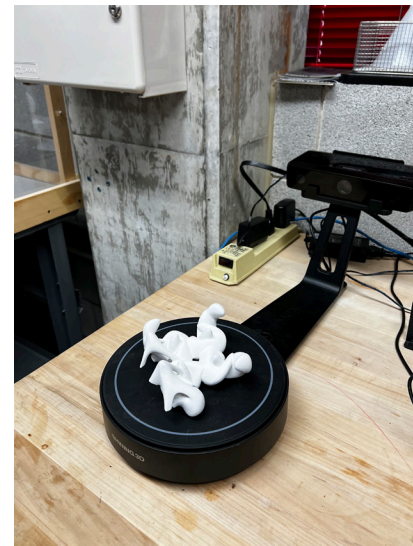
3D Scanner

3D Printed Replica Model

Within this module, you will be introduced to 3D Print File Setup, also known as Slicing. Slicing is the process of applying settings and orienting your model so that the software can generate code to communicate with your printer. Setting up your slicer settings is crucial to a successful print, and to bringing your digital model to life. There are several popular slicers out there, but for this assignment, we will be focusing on two, Prusa Slicer and Ultimaker Cura. Whichever one you decide to use will work for this assignment and each has its own unique benefits. For future use, we recommend you get used to both.

Final Deliverable
One .GCODE file sliced for CR-10

Instructions

1. Installation
Download Cura: https://ultimaker.com/software/ultimaker-cura

Download PrusaSlicer: https://www.prusa3d.com/page/prusaslicer_424/

Follow the install instructions and open the selected software.

Drag in your 4"x 4" x 4" model from Assignment M3A1 or M3A2

2. Orientation
Following the THEY Rules, orient your object to have the HIGHEST amount of surface area touching the build plate while also REDUCING the amount of support required

3. Settings
Infill - The amount of structure within your walls
20% is standard but can go up or down depending on your requirements.
Speed - Typical printers can handle 60mm/s but can go up or down depending on the material or required finish quality
Layer Height- Determines finish quality typical is .2mm(Most important time saver setting)
Adhesion- Enable whenever possible
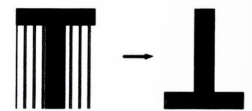Support - Reduce usage as much as possible but enable to ensure print quality and stability

THEY Rule

The 'THEY' rule is a set of basic rules that explains how to orient the parts to print according to their geometry.

The rule says we can only rotate the pieces in the XZ plane.

T needs supports. The problem is solved if we rotate it 180°.

H needs supports. It has a bridge. Some slicers have a 'bridge' option that increases the printing speed in that area avoiding the supports.

E needs intermediate supports. The problem is solved if we rotate it 90°.

Y is the maximum angled shape to 3D print without supports. It could be 3D printed as it is, or rotated 180°, improving the contact with the build plate.

Advanced 3D printing with Grasshopper®   23

Source :Advanced 3D Printing with Grasshopper Clay and FDM- Diego Garcia Cuevas , Gianluca Pugliese

Instructions

1. Import
Either drag in a file from the desktop or elsewhere, or go to File>Open Files and select your model.

2. Scaling + Printer Selection
Ensure the model is within 4x4x4 dimensions, also make sure you are on a Prusa i3 Mk3s model print bed. If not go to (Preferences -->Printers-->Add New-->Add a Non-Networked Printer-->Prusa3D-->Prusa i3 Mk3/Mk3s-->Add) If prompted click next until you return to the build plate menu.

3. THEY Rule
Follow THEY Orientation Rules from the previous page, and reduce the amount of filament used by supports when possible (Use the rotation button on the left side of your screen to orient your object).
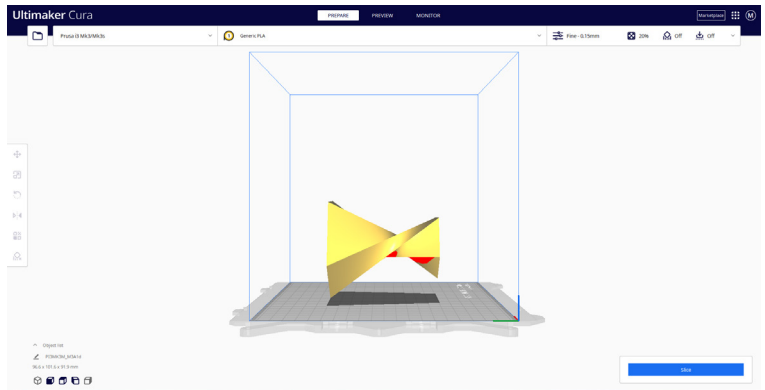
4. Settings
Adjust layer height to .2mm
Infill to 20%
Enable Adhesion in the form of a Brim or Raft
Enable supports for any overhangs or orientation issues that couldn't be solved by the THEY Rules
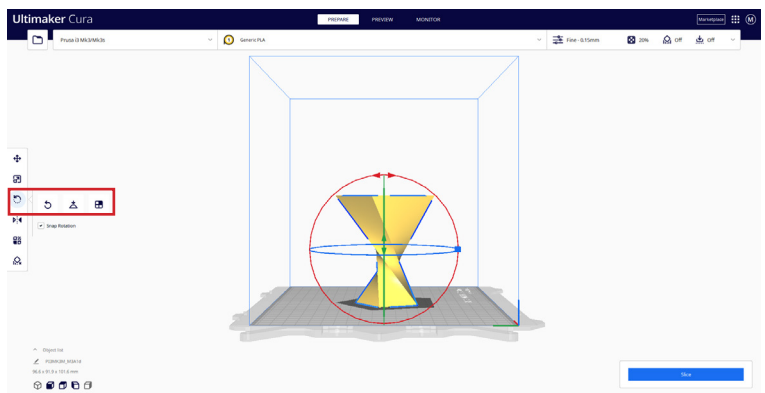
5. Slice
Finally, slice your file with all the applied settings, and use the sidebar to scroll through the layers to check for gaps or areas where extra support may be needed.
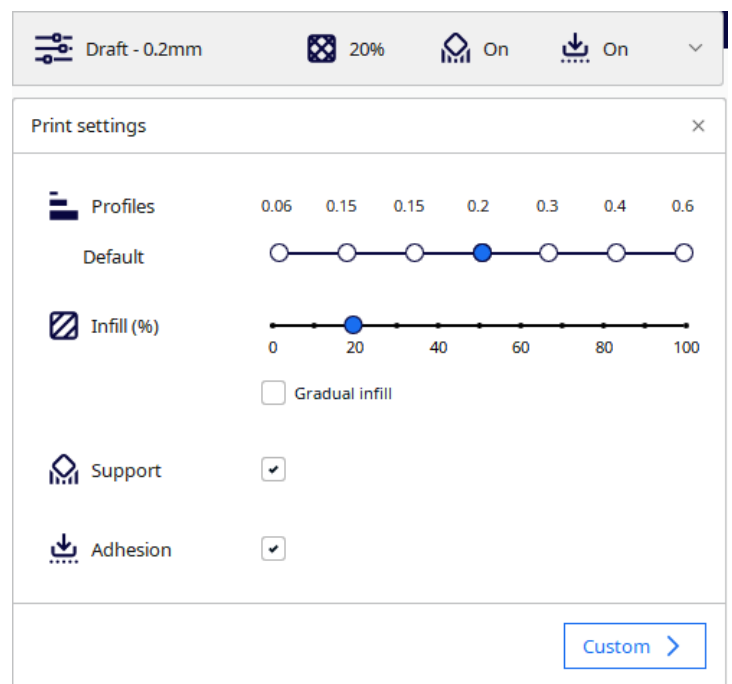
6. Save .GCODE

Proceed to M3A4

Bad Example of Orientation

Good Example of Orientation

Recommended Settings

Instructions

1. Wizard
When you open PrusaSlicer for the first time it will run you through a setup wizard. On the bottom left press (Select all Standard Printers). Then press finish.

2. Import
Then go to file Import, STL/OBJ then open your file.

3. Setup
First, select the Prusa i3 Mk3s/Mk3s+ from the Printers drop-down on the right menu. Then use the THEY Rule to orient your model in its best possible printing orientation (Use the rotate menu on the left to orient your model).

4. Settings
Ensure on the top right the mode is set to Simple for this assignment otherwise your menus will not look the same as the screenshots.

Select .20 mm quality
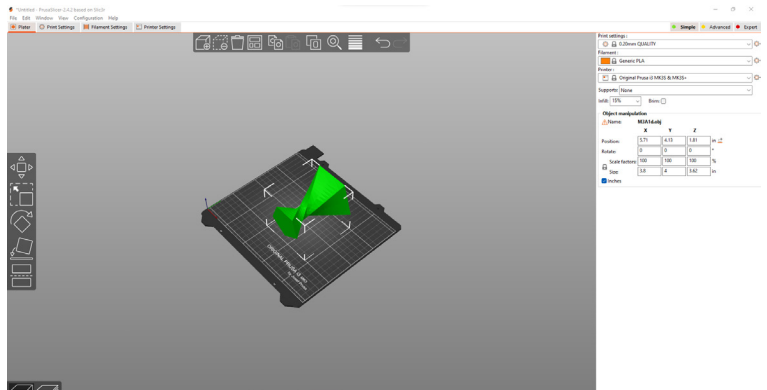Generic PLA
Supports Everywhere
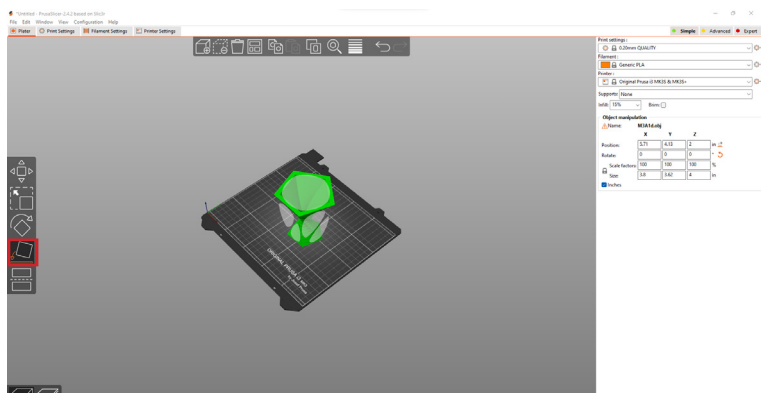Infill 20%
Brim Enabled

5. Slice
Select slice and begin scrolling through the layers generated by the software. This will show you how the printer will build up your model, try to use the vocab presented at the beginning of this module to understand what each color means.
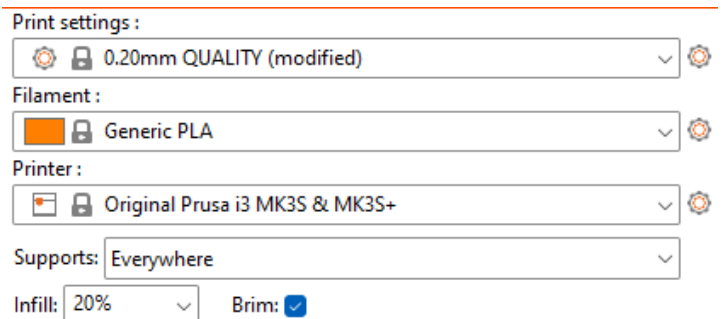
6. Save .GCODE

Proceed to M3A4

Bad Example of Orientation

Good Example of Orientation

Recommended Settings

In this assignment, you will be introduced to the mechanics of 3D Printing. Up until now, you have completed all the necessary digital steps required to 3D Print. Now we will apply that work to setting up and operating a 3D Printer.

What to bring with you

-Flash drive containing your .GCODE from M3A3 and your .STL or .OBJ from M3A2

Final Deliverable:
One .JPG or .PNG of the finished 3D Printed Model

Instructions

1. Sign up
To complete this module you must sign up for a time to operate a 3D Printer. Waiting time may vary depending on Fab Lab loads. Sign up on our website under the Consultation tab. Look for M3: 3D Printer. One of our staff will get back to you after sign up to schedule a time.

2. Printing
During your assigned time, we will provide you with filament for your run a "first layer" test print. A lab faculty member will walk you through how to heat up and load filament into the machine. After that we will use your sliced GCODE to upload and begin printing your model. If we find errors in your .GCODE we will determine the severity of the error and either recommend you resolve these errors and sign up for another time slot or we will use your .STL to generate a new .GCODE. (This will be left up to the LAB Staff)

3. Observations
The golden rule is to always watch the first layer go down, most errors in 3D printing happen at the first layer if it is not fixed then it can compound into machine failure or damage. Once your first layer has been printed, you have successfully completed the Module.

4. Optional
If want your model printed in its entirety, you will be required to submit the STL file to our request management system. After the model is printed, you will be contacted by Fab Lab staff for pickup.