# Slicing - Ultimaker Cura

Cura is a free program that converts .stl files into code that the printer is able to interpret. This process is called **slicing** because the model is "sliced" into many thin layers.

## Getting Started

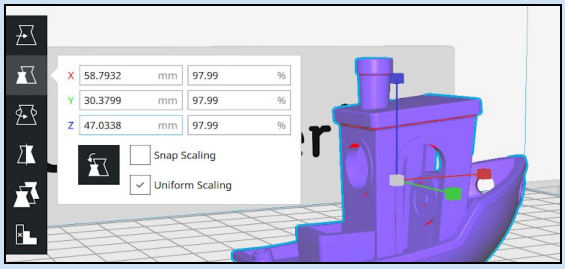
Open **Cura and** click the "**Open File**" icon to the left. You can also load a file using **File** > **Open**. Choose the .stl you wish to print.

# Manipulating your Model

**You now have the ability to orient and resize your model. Remember that Cura is a virtual representation of the printer's build plate, so you will not be able to manipulate your print outside of adjusting the size, position, and orientation on the build plate.**

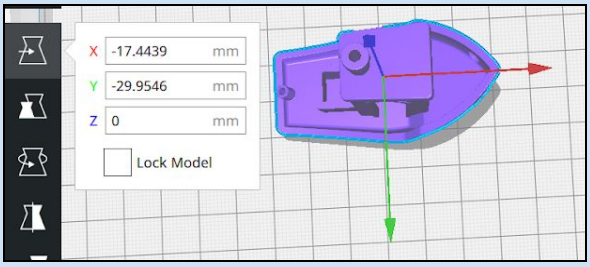
### Scale

When you click on the **Scale** icon, you will be able to resize using exact values in mm, percent of original size, or arrows.



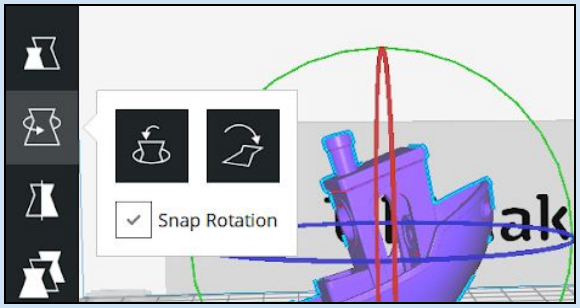

### Move

Click the **Move** icon and you can position your model either by clicking and dragging it, or by changing the exact coordinates.



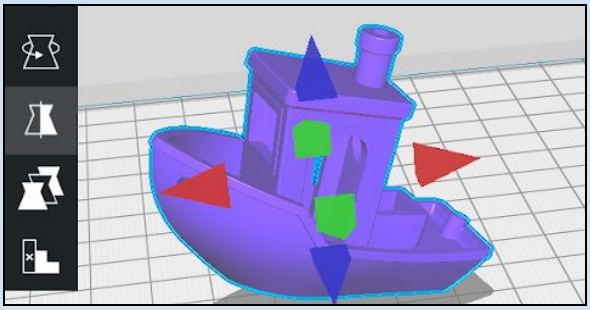

### Rotate

Click **Rotate** and you will be able to drag circular planes and adjust the orientation of your model.



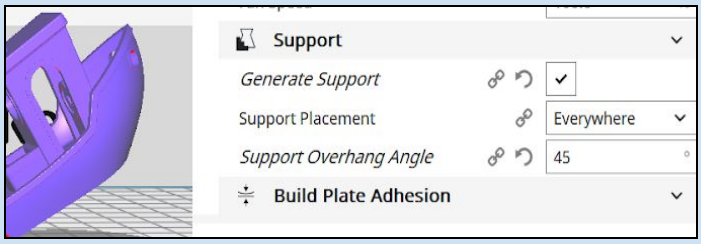

### Mirror

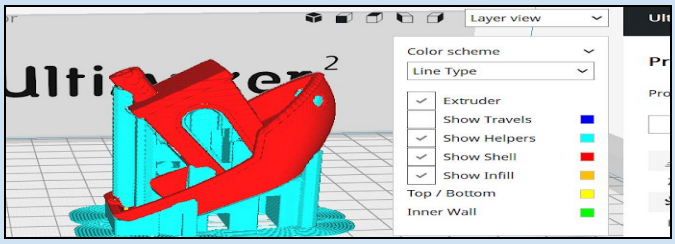
Click the **Mirror** icon and you will be able to reflect your model over the X, Y, or Z axis using arrows.



## Adding Supports

Cura highlights any overhangs in red. If your model has an overhang greater than about 45°, you may need to have it generate supports for you by checking the "**Generate Support**" box under **Recommended** settings, or the same box under "**Support**" in **Custom** settings. By going to **View** > **Layer View**, you will be able to see what your supports ("**Helpers**") will look like.

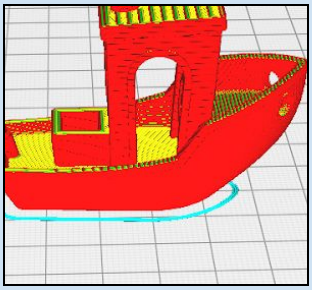



## Build Plate Adhesion

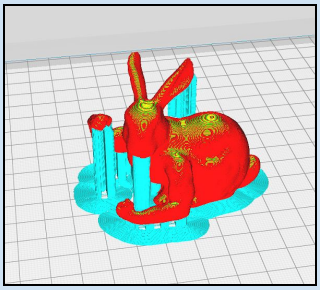
There are three types of build plate adhesion: **Skirt, Brim,** and **Raft.** Each of these adhesion types has a different way of helping your model stick to the build plate.

**Skirt**



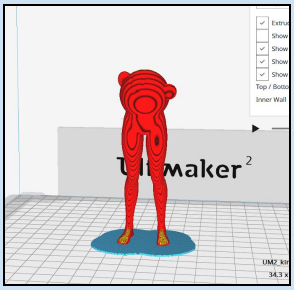
Skirts are most commonly used with models which don't need any extra help adhering. The skirt just allows you to visually make sure the material is sticking.

**Brim**



A brim is useful when you are using supports connected to the build plate. It essentially places a cm or two of filament around the edges of the base of the model, which helps to hold it up.

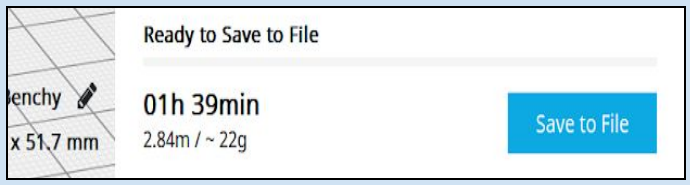
**Raft**



Rafts are generally only used when an object will not be able to stand up on its own. A raft is a full layer of filament underneath your model. This can sometimes damage the bottom of the model.

## Preparing to Print and Checking Time & Material

On the lower right corner of your screen you will see a button that says "**Prepare**". When you click that button, Cura will slice the print and display the amount of time that it will take to print, as well as the length and weight of filament that it will require.

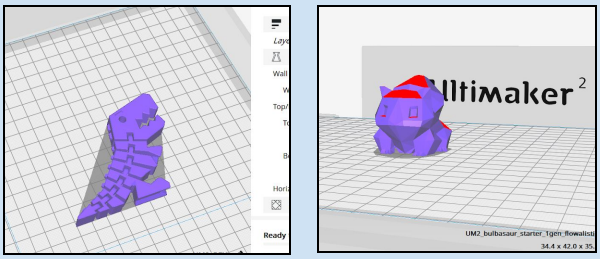



Once the print is prepared, you can click on "**Save to File**" and download your .gcode file. Add this file to an SD card and you are ready to print!

## Which models won't work for printing?

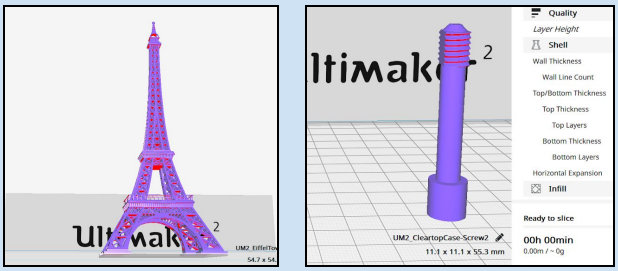
Any model with a lot of steep (greater than 50°) overhangs will be difficult for most any 3D printer to handle. This is because the machine cannot print without building off of a surface. Models that have tall and skinny extrusions also don't tend to print well. This is because they are prone to being nudged by the nozzle and slanted to the side or even tipped over, which will cause the printer to attempt to print into the air.

**Good**



Less fine detail, few overhangs, flat to bed

**Bad**



Too many overhangs, too tall/skinny

## How do I…?

**Increase Print Quality**

- Decrease the print speed.
- Set the layer height to .1mm to have smoother transitions between layers.
- Decrease the nozzle size to .4 or .25mm in **Settings > Printers > Manage Printers* > Machine Settings** to achieve finer detail.
- Increase the amount of walls in the shell for a smoother finish.

**Decrease Print Time**

- Increase the nozzle size to .6 or .8mm to allow for faster extrusion.
- Increase speed (**Warning:** this could impact the quality of your model).
- Set layer height to .2mm to decrease the amount of layers needed for the model.

**Decrease Material Used**

- Decrease the infill in recommended settings to make your build more hollow. (**Warning**: This could weaken your build.)
- Decrease the scale of your model to cut down on the amount of filament needed.

*Some versions of Cura have an **Extruder** menu. If this is the case, the setting for nozzle size will be located under that menu.

# So, where do I get a model?

If you've decided TinkerCAD isn't your thing, no worries! 3D modeling isn't for everyone and there are tons of resources out there to find fun, cool, and even useful models to print. Some makers charge for their files, but a majority of them are completely free. Here are a few sites we recommend:

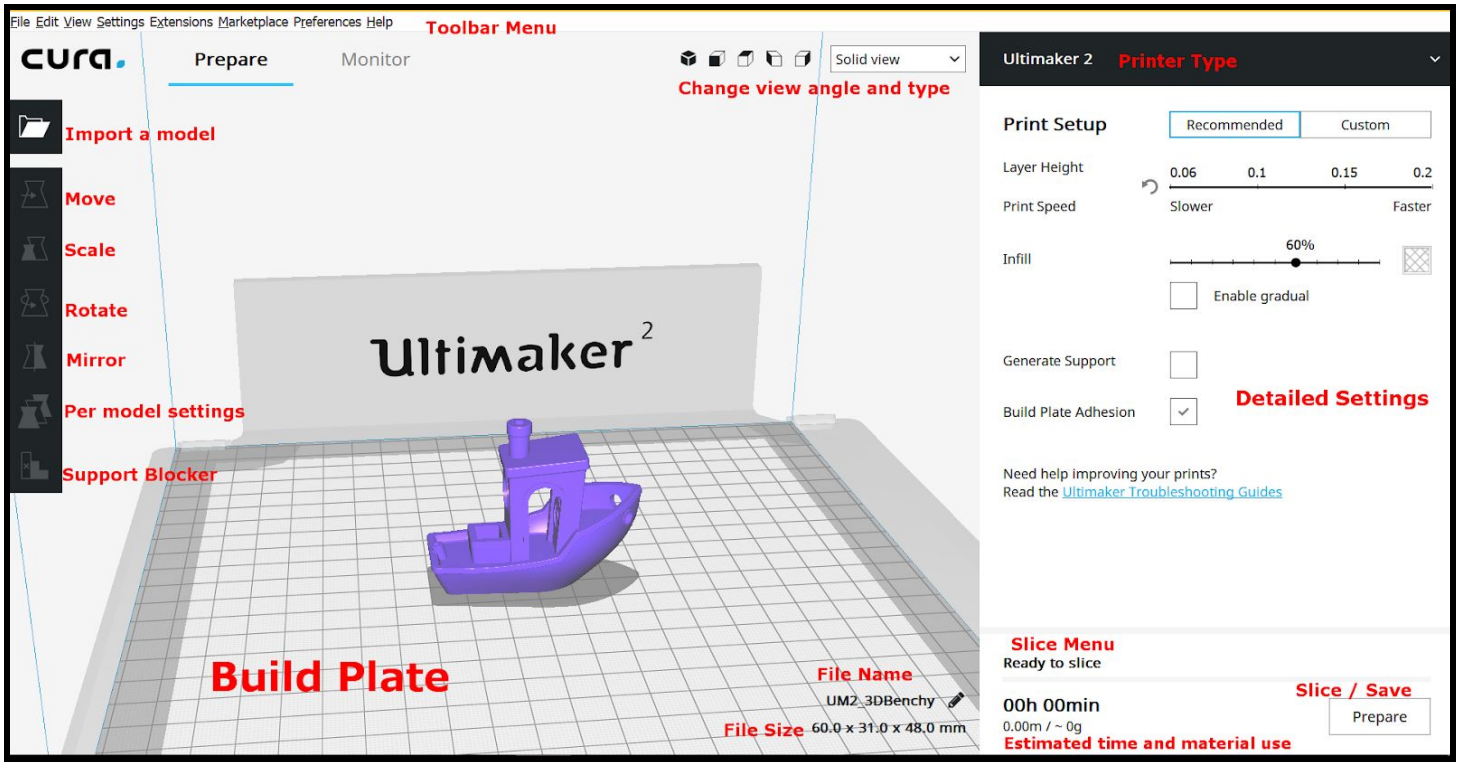
www.thingiverse.com (recommended)
www.pinshape.com (contains some paid files)
www.cults3D.com
https://3d.si.edu/download-browser (Smithsonian)

# Workspace Overview

## Basic Commands

| Rotate/zoom your view | Select Object | Move view |
|---|---|---|
| Scroll wheel | Left mouse click | Right mouse click and drag |

## Screen Layout




Hotkeys

**3D Viewer controls**

- Arrow keys: Rotate the camera in the 3D viewer.
- '+' and '-' keys: Zoom the camera in the 3D viewer.
- CTRL + UP or DOWN arrow keys: Jump layers in layer view mode. *Unfortunately, this does not currently work on Mac OSX.*

**Snap adjustment**

- Hold SHIFT: Disable snap rotation or snap scaling, when the tool is active.

**Uniform adjustment**

- Hold CMD / CTRL: Disable uniform scaling, when the tool is active.

**Undo / Redo**

- CMD / CTRL + Z: Undo.
- SHIFT + CMD / CTRL + 'Z': Redo.

**Other**

- Hold SHIFT + click: Select multiple objects in the 3D viewer.
- Hold CMD / CTRL + click: Select an object in a group.
- CMD / CTRL + D : Clear all objects from the build plate.
- FN / CTRL + DELETE: Delete selected object.
- CMD / CTRL + O : Open file.

Student Technology Center
Haggard Hall 2nd Floor
(360) 650-4300
www.stc.wwu.edu
Workshop guide updated Fall 2018
by Allie Pinnock