# Cura Tools & Navigation

### Open Cura

This guide and in general in the lab, we use the slicing program Cura. The rest of this guide won't make sense if youre not in Cura. So look for Cura's logo on the lab computer so that we can get to work!

### Starting a New Project

You first want to make sure there is nothing on our building surface. This can be done by pressing Ctrl+N or by going to File->New Project.

### Loading in a Model

Next course of action is to load in a model. This can be done by pressing Ctrl+O or by going to File -> Open File(s). Make sure that the file is saved as a .stl. Once you have loaded in the model, the tool bar below will pop up on the left

### Move (T)

Allows you to move the model within the scene.

### Scale (S)

Allows you to scale the model within the scene.

### Rotate (R)

Allows you to rotate the model within the scene.

### Mirror (M)

Allows you to mirror the model on all different axis.

### Per Model Settings

Allows you to change the mesh type and settings of models.

### Support Blocker (E)

Allows you to block an area so that supports will not be generated.

### Navigation
- Left Click- Select Models
- Right Click- Orbit around scene
- Middle Mouse Button- Pan the viewport
- Scroll in/out- Zoom the Viewport

## Table of Hotkeys

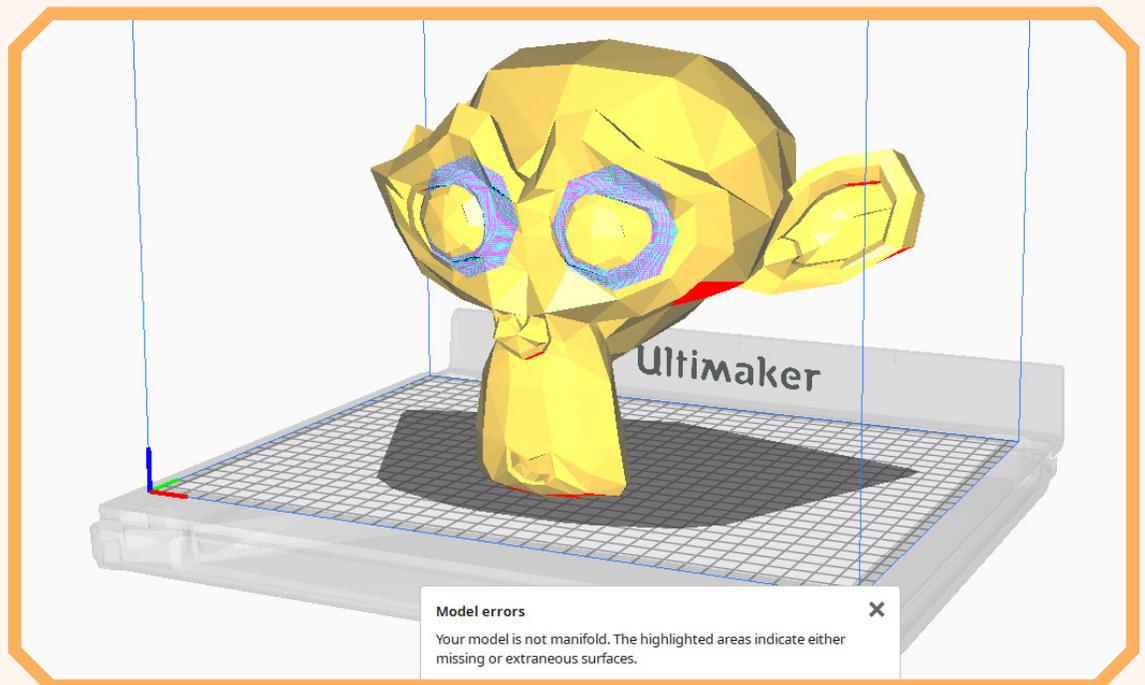| | | | | | |
|---|---|---|---|---|---|
| Move | T | Open File(s) | Ctrl+O | Undo | Ctrl+Z |
| Scale | S | Save File | Ctrl+S | Redo | Ctrl+Y |
| Rotate | R | Clear Build Plate | Ctrl+D | Select All | Ctrl+A |
| Mirror | M | Group Models | Ctrl+G | Arrange All | Ctrl+R |
| Support Blocker | E | Merge Models | Ctrl+Alt+G | | |
| New File(s) | Ctrl+N | Ungroup Models | Ctrl+Shift+G | | |

### Upload the Model(s) to Cura

The first thing that must be done is placing the model(s) (if you did not follow along in the Tools & Navigation). You can either drag and drop the .stl file or you can click in the upper left File->Open File(s). Once you do that, you should see a yellow version of the model you imported sitting on the build plate.

### Look for Problems

Some times the model will have some problems that will need to be fixed in order to get it to print. Look at the bottom of the screen to see if any errors popped up. If theres no errors and you do not see any highlighted areas on your model, thats a good sign! If there are highlighted areas, don't worry, because we will cover how to begin to fix it on the next page.

### Check the Size

Finally, when doing your preliminary checks, look at the size of your model. If it looks really small, that's common. Many times when you import into Cura, your model is 10x smaller than in whatever 3D modelling program you were using. This is because often what the modelling program thinks is 1m, Cura reads it as 1cm.



**Model errors** ✕
Your model is not manifold. The highlighted areas indicate either missing or extraneous surfaces.

### Model Error Codes

Reading any error codes is the first thing that should be a red flag. Luckily, they tell you exactly what the error is. Which often makes it a simple fix. Exactly what the codes mean and what to do to fix them is on the next page.
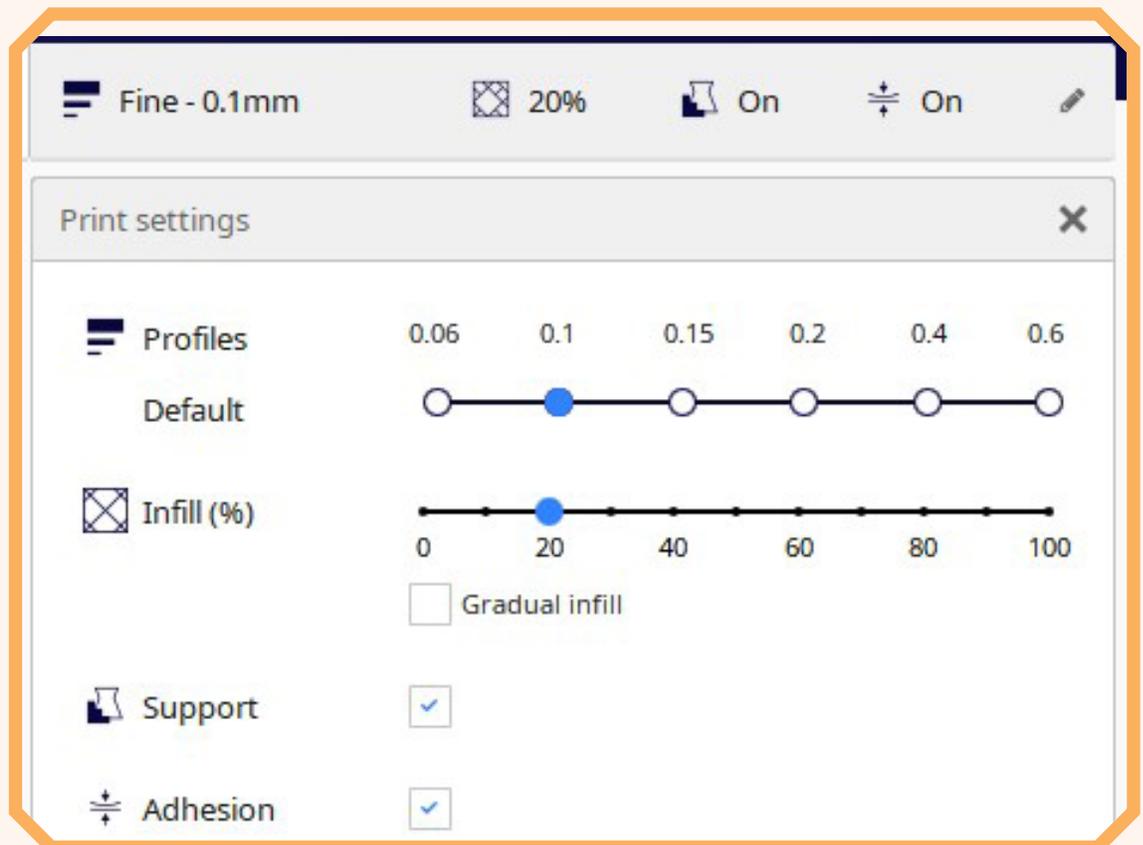
### Overhangs

While its not exactly a "problem" it is something to be aware of. Overhangs are shown by the red faces that can be seen on some bottom facing faces. This means that your model will need supports to help aid in the printing process. These supports can be easily broken off after printing. You can find the drop down menu that has the supports check box in the top right of the UI.

### Placement

Finally, make sure that your model is properly placed, especially on the Z-axis. It is easy to accidentally move the model too low and it will be cut off by the build plate. You are also able to use Ctrl+R or right click on your model to arrange the model(s) on the build plate.

| Fine - 0.1mm | 20% | On | On | ✏ |

**Print settings** ✕

| Profiles | 0.06 | 0.1 | 0.15 | 0.2 | 0.4 | 0.6 |

Default

Infill (%)
0    20    40    60    80    100

☐ Gradual infill

Support ✓

Adhesion ✓

# *Fixing Simple Things*

## Model Errors and their Solutions

Often times, models will either have a "Reversed Normals" or "Manifold" errors. Reversed normals, while it might look normal in the Prepare tab, you will see the layers in the Preview tab are not correct. So you will need to open the model in a 3D modelling program and have the normals recalculated.

A manifold error means that there is a hole and your model is not solid. You should be able to see the highlighted area where the hole exists. This as well needs fixed in a modelling program such as Blender.

## Rotation and Scaling

After using the tool bar on the left to scale to the desired size, try rotating your model to a flat side (if you have one) or to the area that will require the least amount of support. This will allow for a faster print time that uses less grams of filament. As you can see in the lower image, printing the monkey upside-down, will use much less supports than if it was printing rightside-up.



Model errors
Your model is not manifold. The highlighted areas indicate either missing or extraneous surfaces.
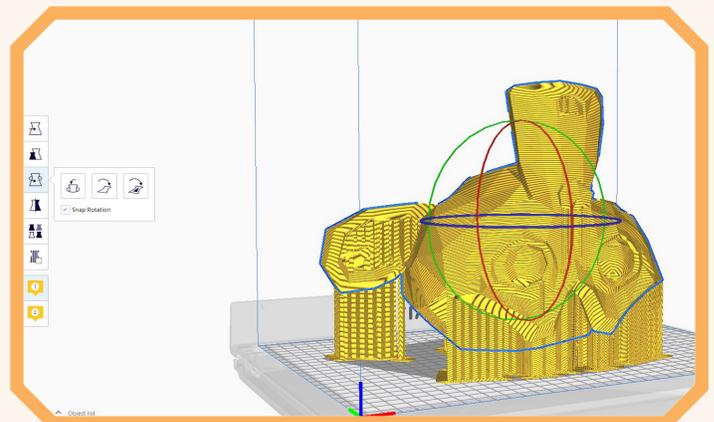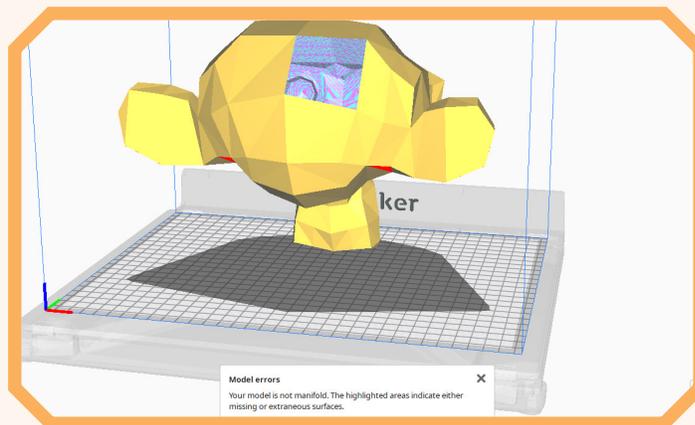
# *Submitting files for printing*

### Slicing

Once your model is ready to go press the "Slice" button in the bottom right. Cura will take a minute to figure out the best way to print the model. It will allow you to go to the preview tab to look at all of the layers using the bar on the right of the screen.

### Print Information

After slicing, check the box in the bottom right of the screen. There is the estimated time as well as the grams of filament. Both of these need to be inputed into the excel sheet. Once you have filled out all of the boxes. Save the .stl file from Cura into the "For Print" folder on the desktop.

### Double Check

Now that you're finished, double check that everything was filled out and the model is correct before leaving. This makes it much less likely that something was done incorrectly.



1 day 0 hours 36 minutes

474g · 59.94m

Save to Removable Drive