

3D Printer Certification Workshop

Department of Mathematics
CU Boulder

Supervisors: Faan Tone Liu, Carly Matson,
Cherry Ng, Athena Sparks

Contents

1	Introduction	2
2	Designing models	2
2.1	TinkerCAD	2
2.2	Blender	2
2.3	Mathematica	2
3	Procuring Filament	3
4	Using Cura for LulzBot	3
4.1	Loading Your Model	4
4.2	Moving and Viewing Your Model	4
4.3	Rotate/Scale/Mirror	4
4.4	Basic Quality Options	5
5	Printing an model	7
5.1	Connecting to the Printer	7
5.2	Loading Filament	8
5.3	Printing Process	9
5.4	Removing the Object	10
5.5	Shut Off Printer and Prepare for Next User	10
6	Post-processing	10
6.1	Sanding	10
6.2	Acetone Vapor Bath	10
7	Troubleshooting	10
8	Record Keeping and Sharing	10
9	Certification	11

1 Introduction

Our department's 3D printer program was funded by the math department and by ASSETT, which backs projects to improve undergraduate education at CU Boulder. The main purpose of this printer is to give instructors the opportunity to supplement their teaching with 3D models.

This workshop is designed to give a basic overview of how to start designing and printing your own 3D models. It is also the first step in receiving the certification required to use the printer. This document will cover software that you can use for designing models, how to use the 3D printer, and common troubleshooting tips. Additional information can be found in the LulzBot TAZ 6 Operation Guide, the LulzBot TAZ 6 User Manual, and the Cura LulzBot Edition Manual.

Our machine is the LulzBot TAZ 6. It is a self-leveling, self-cleaning machine that features a printable space with dimensions 280 mm x 280 mm x 250 mm. It can use many different types of filament, which will be discussed in more detail in section 3.

2 Designing models

In order to design a model for 3D printing, you will need to use CAD software. Most of these programs have online tutorials. Here are a few of the many software options.

2.1 TinkerCAD

TinkerCAD comes with a library of pre-made shapes that you can use to build models. The click-and-drag build environment is intuitive and simple, but its simplicity has its flaws. For example, many of its "round" shapes, like cylinders and paraboloids, have cross sections that are 24-gons instead of circles. Still, it is very easy to use and great for many applications.

Pros: Browser based, free, easy-to-use, no coding required

Cons: Not very powerful, entering parametrized surfaces requires Javascript

Source: www.tinkercad.com

2.2 Blender

Blender is a full 3D modeling suite, so it is much more complicated than TinkerCAD. Like TinkerCAD, it comes with a library of pre-made shapes that you can click and drag. But unlike TinkerCAD, Blender can thicken the wall of a model to make it suitable for 3D printing, smooth out rough surfaces, and create a mesh over your model that allows you to adjust vertices in great detail. Also notable is the fact that Blender allows you to input parametrized surfaces.

Pros: Powerful, free, cross-platform, good support for parametrized surfaces, no coding required

Cons: Harder to learn than TinkerCAD

Source: www.blender.org

2.3 Mathematica

The powerful math functions in Mathematica are great for creating mathematical models. For example, Mathematica will allow you to model and object that represents a graph of the gamma function. Then, you can export your model as an .STL file and print.

Pros: Excellent support for math functions
Cons: Free only on school computers, coding required
Source: www.wolfram.com/mathematica

3 Procuring Filament

The filament provided by the department is PLA plastic filament, which costs about \$30 for a 1 kg spool. This filament is *for classroom use*, which means you may only use to for projects intended to supplement your teaching.

You may purchase filament for personal use in the following ways:

- The Integrated Teaching and Learning Laboratory (ITLL) is a place where you can buy filament on campus.
- The LulzBot website sells filament and allows you to search for filament that is compatible with our machine, the LulzBot TAZ 6.
- Other online retailers also sell filament. Make sure your filament is compatible with our printer.

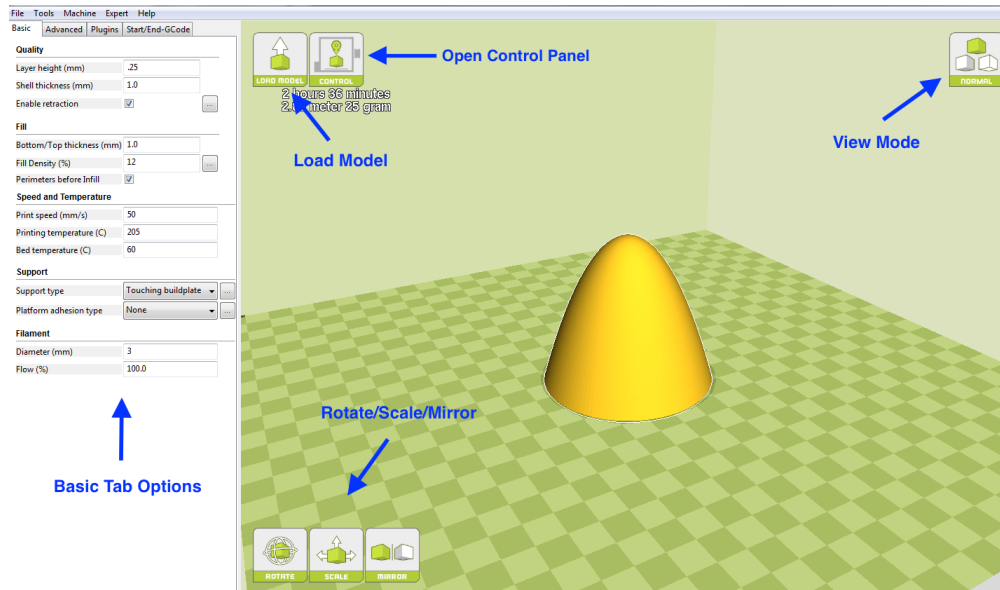
Our machine is capable of using more exotic filaments like wood, bronze, polycarbonate, and conductive material, too. Come talk to the supervisors about using exotic filaments!

4 Using Cura for LulzBot

In this section, we will cover the basics of how to get your model ready to print using the free software program Cura LulzBot Edition. This software has been installed on the computer connected to the printer. You may download the software onto your personal computer by following this link <https://www.lulzbot.com/cura>. Instructions for installation and set up can be found in the LulzBot TAZ 6 User Manual.

Cura prepares an stl file for printing by converting your model into GCODE. This software also allows you to control the operation of the printer. The sections below cover some of the common controls you may use in editing your model. More information can be found in the Cura Lulzbot Edition User Manual.

Below is a screen shot of a Cura window with loaded model. The arrows highlight the options discussed in this section.



4.1 Loading Your Model

Upon opening the application Cura, a window will appear showing a list of setting options on the left and a graphic representing the print bed on the right. To load your model into Cura, select *File/Load model file* or select the *LOAD MODEL* button located in the top left of the print bed graphic. An image of the model will appear on the screen.

4.2 Moving and Viewing Your Model

To re-position your model, click and drag. You can center the model on the platform by right clicking on the model and selecting *Center on platform*. By right clicking on the model you can also delete, make a copy, and reset the model.

By scrolling in or out, you can zoom in or out on your model. To change the angle that you view the model, right click anywhere on the print bed screen and drag. Clicking the top right button that says *NORMAL* will reveal a drop down menu of buttons which change the viewing mode.

4.3 Rotate/Scale/Mirror

When you click on the model, three buttons will appear at the bottom left of the print bed screen labeled *ROTATE*, *SCALE*, and *MIRROR*.

The *ROTATE* button allows you to rotate you model allow three specified axes. To rotate you model, click the *ROTATE* button then click and drag along one of the axes. To rotate by a single degree, hold down the *shift* key while dragging. To make you oject lay flat on the print bed, select the *ROTATE* button followed by the *LAY FLAT* button which will appear.

The *SCALE* button allows you to rescale your model before printing. Select your model then click the *SCALE* button. A menu will appear allowing you to change the scale by a constant multiple or by entering the desired dimensions. You may also select the *MAX* button to scale the model to the maximum size the printer can print.

The *MIRROR* button allows you to reflect your model about the *x*, *y*, or *z*-axis. To do this, select the model then click the *MIRROR* button followed by the *MIRROR X*, *MIRROR Y*, or *MIRROR Z* button.

4.4 Basic Quality Options

The sections below are copied directly from the LultzBot TAZ 6 User Manual. It covers the quality control options located on the left of the window under the *Basic* tab. For the advanced setting, click on the *Advanced* tab and consult the LultzBot TAZ 6 User Manual. It is advisable to reset to the default profile settings when you open Cura. This is done by selecting *File/Reset Profile to default*.

Layer Height

The thickness of each printed layer is known as the “Layer Height”. The smaller the layer height, the smoother curves will appear. Larger layer heights are better for bridging and overhangs. Smaller layer heights will also increase print time, as it will take more layers to complete the model.

Shell Thickness

This defines the number of vertical walls that comprise the outside of your model. We recommend keeping this set to multiples of your nozzle width. Your TAZ 3D printer is equipped with a 0.5mm nozzle.

Enable Retraction

Retraction tells your printer to pull filament out of the hot end upon travel moves. Travel moves are when your print head moves from one area of the print, to another without laying down filament. We recommend keeping this on for all filament types, and adjusting the retraction length and speed for the specific filament.

Bottom/Top Thickness

Also known as “Surface Layer” this will determine how thick the top and bottom layers are. A larger number here will create a thicker top and bottom which can be helpful for strength, bridging, and quality purposes. We recommend keeping this number as a multiple of your layer height.

Fill Density

This number is expressed as a percentage. 0% will give a completely hollow print, while 100% will give you a completely solid model. We have found that 20% to 40% fill density is functional for most prints.

Perimeters Before Infill

This option will toggle the order in which the in ll and perimeters will be printed. We recommend leaving this on.

Print Speed

Your overall printing speed can be adjusted here. If no other speeds are determined in the later sections your printer will automatically default to this speed. This speed will be different, depending on what type of filament you are using.

Printing Temperature

This is where you will set the hot end temperature for your specific material. It will be preset when using the recommended quickprint profiles. This can be adjusted for fine tuning of different filament manufacturers to help layer adhesion or stringing. See Recommended Temperatures Table on page 30 of the LulzBot TAZ 6 User Manual. Your hot end is capable of reaching a maximum temperature of 300°C.

Bed Temperature

This is where you will set the print surface temperature for your specific material. These will be preset when using the recommended quickprint profiles. This can be adjusted for fine tuning of different filament manufacturers to help adhesion or release. See Recommended Temperatures Table on page 30 of the LulzBot TAZ 6 User Manual. Your print surface is capable of reaching a maximum temperature of 125° C.

Support Type

Some models will require support material in order to print properly. This will usually occur when an object has an angle in relation to the build plate between 0 to 45 degrees. It is highly recommended to orient your object so that it minimizes or eliminates the need for support.

Selecting *Touching Buildplate* causes the support material to build up between the heated bed and the object.

Selecting *Everywhere* prints support material between the heated bed and object as well as between the object and itself.

Platform Adhesion Type

Some models have a small surface area contacting the plate. This can create adhesion issues causing your part to pop off at some point during the print. To fix this, use either *Brim* or *Raft*. *Raft* is better used when a model has small heated bed contact points and overhangs.

Brim will create a single layer of filament, contacting and surrounding your model. This will increase the surface area of the part contacting the build platform thereby preventing it from popping off the heated bed. *Brim* will also help in situations where you are seeing corner lift. *Brim* settings can be adjusted in the Expert Settings options.

Raft will generate a layer of material underneath your object. *Raft* was more often used before the addition of heated plates to increase surface area. *Raft* settings can be adjusted in the Expert Settings options.

Filament Diameter

The filament diameter setting is one of the more important settings. Make sure that you update this value periodically with your average filament diameter. While your filament may be referred to as 3mm, it is more likely going to be near 2.9mm +/- 0.1mm. You will want this to be an accurate average, as it will allow your printer to correctly calculate how much filament it is pulling into the hot end. The default value should be set to 2.85mm.

Filament Flow

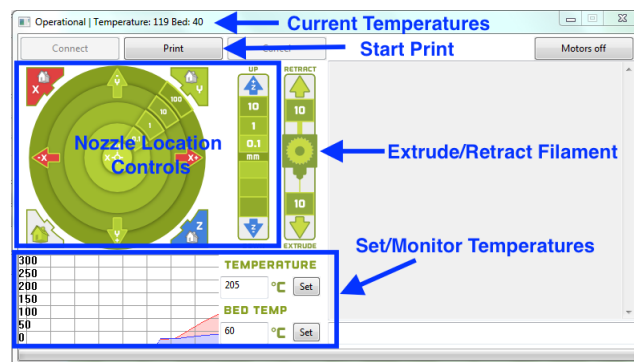
This controls how much filament your printer is extruding in relation to speed. This setting is mainly used to adjust for filament density variations. This may need to be adjusted when switching between different manufacturers or types of filament to adjust for die swell. It is recommended to make small adjustments between prints for fine tuning.

5 Printing an model

The TAZ 6 is has a Graphical LCD controller that allows you to print off of a SD card. The advantages to this is that you do not need to use Cura to run your print. However, Cura is the preferred printer host software, as the inclusion of quick print profiles, the combination of slicing engine, and printer host communications allows for easy all-in-one use. If you would like to print drectly form an SD card, consult the LulzBot TAZ 6 User Manual. This section will discuss how to print from Cura.

5.1 Connecting to the Printer

The campus computer next to the printer is already plugged into the printer. To connect to the printer, turn the printer on by flipping on the red on/off switch in the front left corner of the printer and then clicking the *CONTROL* button right of the *LOAD MODEL* button on your Cura window. A new window should automatically open with the control panel. Below is a picture of the control panel with some of the key features highlighted.



Moving the Nozzle Head

The *X*, *Y*, *Z*, *UP*, and *DOWN* buttons control the location of the nozzle head. You will rarely need to use these buttons.

Retract and Extrude

The *RETRACT* and *EXRTUDE* buttons allow you to retract and extrude the filament from the nozzle head. The *EXTRUDE* button will be used in the process of loading filament. We recommend clicking *RETRACT 10* if the nozzle head is hot, but you are not ready to start printing. This will retract the filament a small amount and avoid wasting filament that would melt out of the nozzle head.

Setting Temperatures

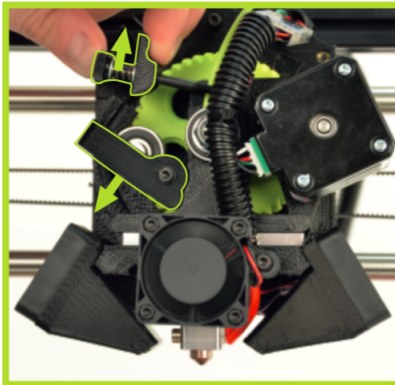
The nozzle temperature is set by entering the desired degrees (Celsius) into the box underneath *TEMPERATURE* and then clicking *Set*. The temperature of the print bed is set by entering the desired degrees (Celsius) into the box underneath *BED TEMP* and then clicking *Set*. Both the nozzle and bed temperatures are determined by the filament being used. Purchased filament will provide a range for these temperatures.

Once the nozzle and bed temperatures are set, you can read the temperature of the nozzle and bed at the top of the window. You can also monitor the temperatures by the timeseries graph shown left of where you set the temperatures. When both the nozzle and the bed are up to temperature you are ready to load the filament and begin your print.

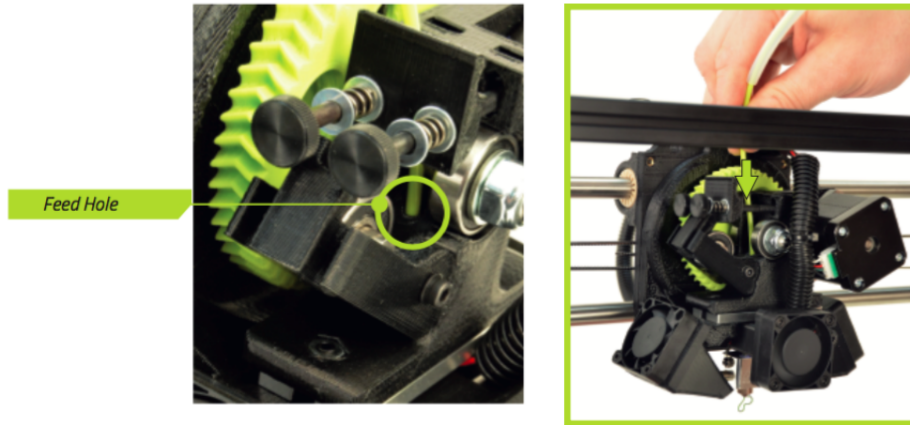
5.2 Loading Filament

While the nozzle and bed are getting up to temperature, you can begin to load the filament. Below gives a quick overview of how to load the filament. For more detailed instructions, see the LulzBot TAZ 6 Operation Guide.

First, place the filament on the filament holder on the right hand side of the machine, then pass the filament through the filament guide tube so it feeds down into the tool head. Using the idler retainer, compress the springs and slide up, allowing the hinged idler to move freely. Then, allow the hinged idler to rotate into the open position (see the picture below). You may need to loosen the two screws on the idler retainer slightly.



When the nozzle head is up to temperature you can insert the filament into the feed hole (see the pictures below). If there is filament that you wish to remove, you simply pull the filament out of the feed hole and then insert your desired filament. Note, filament should not be removed unless the nozzle head is hot. Push the filament into the feed hole until filament begins to extrude out of the nozzle head. This may require a firm push.



To lock the filament in place, compress the springs with the idler retainer and rotate the hinged idler clockwise into an upright position. Lock the hinged idler into place by sliding the idler retainer back down into the locked position. Then tighten the retainer screws if you loosen them previously.



To verify that the printer is ready to print, click the *Extrude 10* button on the Cura control panel. The printer will then extrude a small amount of filament out of the nozzle head. If no filament extrudes, try clicking *Extrude 10* again until you see consistent and repeatable extrusion. Once the nozzle head has successfully extruded filament, wait ten seconds for the filament to cool, then remove the extruded filament with the tweezers.

5.3 Printing Process

Once the nozzle head and bed are up to temperature and the filament has been loaded, you are ready to print your object. To initiate the print process, select the *Print* button on the Cura control panel. The printer will then go through its automatic cleaning and wiping process before starting the print. This should take approximately 5 minutes. The printer will then create an outline of your object before starting the object itself. Do not worry if the beginning of this outline is extruded poorly.

If your print is going to fail, it is likely to fail during the first stages of the print. The most common failure we've encountered is that the initial layers do not stick to the print bed. Therefore, we ask that you wait for the first few layers of your object to be printed before leaving the printer. If no failure has occurred in the first layers, you may leave the printer and check in on its status later. It is a good idea to check your print every hour. You must return to the printer by the end of the estimated print time. The printer will not automatically turn off. It is your responsibility to be available to turn off the machine when your object is finished.

If at any point you need to pause or cancel your print, select the *Pause* or *Cancel* button. You may pause your print and then resume printing. However this is discouraged because while the print is paused, the filament is cooling and when you resume the print, the new warm filament may not adhere well to the cooler filament. If a print begins to fail, cancel the print job. You may need to adjust the profile settings in Cura and try your print again. Consult the LulzBot TAZ 6 User Manual for fixes to common failures.

5.4 Removing the Object

When your object is done being printed the print bed will move all the way to the back for cooling. Then, when the object is done cooling, the printer will automatically move the print bed forward allowing you to remove your object. To remove your object from the print bed, start freeing the object from the outside edges and work your way inward. You may need to use the tools under the desk to help pry the object off of the print bed. Be sure you have removed all filament from the print bed.

5.5 Shut Off Printer and Prepare for Next User

After you are done printing, follow the steps below to shut off the printer and get it ready for the next user.

1. While the nozzle head is still hot, remove the filament and place it in the storage bin.
2. Switch the red on/off switch to the off position to turn off the printer.
3. With dry paper towel, wipe down the print bed. This will remove the oils from your skin that may have come into contact with the print bed.
4. Log your print. See the section 8 for more information.
5. Before closing your Cura window, reset the profile setting to the default settings. To do this select *File/Reset Profile to default*.
6. Close Cura and turn off the computer.

6 Post-processing

6.1 Sanding

6.2 Acetone Vapor Bath

7 Troubleshooting

8 Record Keeping and Sharing

All persons certified to use the printer will be given access to a shared Google Drive folder called *Print 3D Math*. This folder will include this document along with the other manuals mentioned. A shortcut to this folder can be found on the desktop of the provided computer.

As we learn more about our 3D printer it will be important to keep track of what we create. To this end, we will require all prints (personal or not) to be logged using a Google Form called *3D*

Printer Log. A shortcut link to this survey can be found in the shared folder. This short form will provide the supervisors information about the prints that are being made.

We would also like people to share their creations. The form will include a way to upload stl files. All stl files shared via the form will be uploaded to the folder *Shared stl Files* in the shared folder. Sharing your file is optional.

9 Certification

To gain access to the 3D printer, you must first be certified by the department. To get certified you must complete the following.

1. Attend the certification workshop.
2. Agree to the terms and conditions given on the last page of this document.
3. Load and unload the filament under supervision.

Once you completed these tasks, fill out the form on the last page and submit it to one of the supervisors. Upon approval, you will be added to the 3D printer email list, be given access to the shared Google drive folder, and be given access to the printer log (a Google Form).

3D Printer Certification Form

Name: _____

Terms and Conditions:

The filament procured by the department is for models to be use in the classroom. All objects used for personal use may not be made with department purchased filament. You must log every print using the provided Google Form. To use the 3D printer, you must first reserve the printer for the estimated print time using the online reservation form. If you encounter any problems with the machine, you must stop your print and contact one of the supervisors immediately.

Certification Criteria:

- I attend the 3D Printer Certification Workshop on _____.
- I agree to the terms and conditions stated above.

Signature: _____

- I changed the filament with a supervisor.

Supervisor's Signature: _____

For Supervisors' Use Only

The person named above is certified to use the Math Department's 3D printer and has been given access to

- the email list,
- the shared folder, and
- the printer log.

Supervisor's Signature: _____ Date: _____