3D Printing with the LulzBot TAZ 6 and Mini

Open the “Cura for LulzBot” Software
Click on the green “cura-lulzbot” icon
- *It takes a few minutes for the software to boot up. This is normal!*

Select the printer (LulzBot TAZ 6 or LulzBot Mini) in the right drop-down menu. This allows the software to load the appropriate bed plate size, etc.

Turning on the Printer
- Plug the 3D Printer power cable into a wall outlet, and then plug the 3D Printer USB cable into a computer.
- Flip the red power switch on the front left of the printer. The power switch’s light will glow when it is turned on, and the TAZ 6’s LCD screen will light up.
- *Note: Wait to turn the machine on until after Cura has fully launched! Occasionally, Cura will not recognize the 3D printer if the machine is turned on prior to the software being booted up. You will still need to make sure the printer is connected by following the steps in the Getting Ready to Print section of the handout.*

Loading a 3D Model (.stl)
- Either select File → Open, or click the folder icon to load an .stl file.
  - The model will import onto the simulated bed plate.
- Click the bottom icon in the left toolbar to change the model view mode (Solid, Layers, X-Ray).
- Right click and drag to rotate the build plate and see the model at different angles.
- Review the Moving and Editing a 3D Model (.stl) side of this handout for additional information on manipulating the 3D model.

Printing
Please review the other handout for information on Print Setup, Getting Ready to Print, and Temperature / Changing Filament.

When you are ready to start the print, click the Start Print button at the bottom of the Printing Monitor window. The printer will now go through the auto leveling and warm up process *which will take several minutes*. Then, it will start laying down material.

If the material does not extrude, or extrudes incorrectly (i.e., too close or too far away from the build plate), stop the print and start again. Watch the machine closely while printing to prevent tangles, etc.

Removing the Print
- When the print has finished, the bed will move back. This allows the build plate to cool down to the removal temperature.
- Wait until the bed has moved to the forward position and is at 50°C before removing the print with the removal tool. *This is crucial to prevent damage to the build plate!*
- After removing the print (and before you take off any supports), ask staff to weigh the print to accurately bill your account.
- Clean up your work area!
Moving and Editing a 3D Model (.stl)

NOTES:
- The model must be selected with a left click in order to use these modes.
- Depending on the selected mode, spheres or arrow lines will appear around the shape.
  - X axis is RED
  - Y axis is GREEN
  - Z axis is BLUE

Move - Adjust where the model will print on the plate
- Left click and drag on any of the axis arrows to move the object in a single direction.
- Left click and drag the object to move it around in any direction.
- Type in a numeric value (defaults to mm) to select an exact position.

Scale - Change the model’s size
- It defaults to 100% scaling, with Uniform Scaling checked. Left click and drag on any of the axis arrows to uniformly increase or decrease the object’s size, or type in a percentage value in the ratio window.
- To perform a non-uniform scale (stretching the object only along one axis) uncheck Uniform Scaling, then left click and drag on an axis arrow to stretch or shrink that axis only.

Rotate - Adjust the orientation of the model
- Left click and drag on any of the axis spheres to rotate the object in a single direction.
- Select the Reset icon to revert to the original position, or select the Lay Flat icon to set it down flat.
- Some files may print better at different rotations to minimize weak points. Adjust the rotation based on perceived stress points to ensure the longevity of your creation.

Mirror - Create a mirror image of the object, replacing the original object
- Left click on any of the axis arrows to mirror the object’s placement.

Multiply - Create additional copies on the build plate
- Selecting this mode will bring up a window to enter the number of desired copies to create.
Print Setup

**Category:** Select All.

**Material:** Please select ABS (IC3D), PLA (Verbatim), Magnetic (Proto-pasta), or Conductive (Proto-pasta) depending on your filament type.
- This selection is very important, as it automatically adjusts the hotend and build plate temperatures for printing.

**Profile:** Choose from Standard, High Quality, or High Speed.
- Profile selections change print time and quality of the print. High Speed will be lower quality, High Quality will be slower speed, and Standard is somewhere in the middle.

**Print Setup:** We encourage you to leave this at the Recommended setting.
- With Custom setup, you can manually adjust settings. Use caution when using custom settings!
- If you print with custom settings and it fails, we will still charge you for it.

**Infill:** This determines the internal density of the print. More infill will equal more weight and print time, while less infill is the reverse.
- We recommend 20%, 50%, or Gradual. Preview how the layers and supports will print with the View icon in the left toolbar.
- We do not recommend 0%.

**Generate Support:** Generally always leave this on. In some cases you don’t want any supports, so you can turn it off if needed.

**Build Plate Adhesion:** Choose from Skirt, Brim, or Raft.
- A Skirt (single outline pass) adds the least amount of materials and time to a print, and a Raft (solid base layer) adds the greatest amount of time and materials.
- If your print is curling, try adding a Brim or Raft.

Getting Ready to Print

When you have gone through the settings under Print Setup, you are ready to move to the printing interface!

On the Printing Monitor window, it should say USB Printing (Connected via USB).

If it doesn’t display USB Printing, check your connection to the printer (and make sure it is turned on). Then, press Connect.

If you are changing out filament colors or types, please review the other side of this handout, Temperature / Changing Filament.

You are now ready to start the print! Click the Start Print button.

**Tip:** To reduce wait time during the auto leveling and warm up stage of printing, pre-heat the build plate before you start the print!
Temperature / Changing Filament

If you are changing out filament, you will have to pre-heat the hotend (extruder) to the desired temperature.

Retracting / Withdrawing Filament
1) Enter the desired temperature and click the Pre-heat button.
2) On the tool head, compress the springs of the idler retainer and slide up, allowing the hinged idler to move freely. Let the hinged idler tilt left to the open position.
3) Once the hotend has reached the set temperature, pinch the filament and pull up to remove it from the hotend.
4) Trim the filament end, then carefully secure the loose end before putting the spool away so the filament does not get tangled.

Extruding / Inserting Filament
1) If you use a filament other than the type that was previously loaded in the printer, there may be a difference in the temperature required for purging the residual filament in the hotend and the temperature required for 3D printing with the new filament.
   a. Set the hotend to a temperature that splits the difference between the two filament temperatures. (i.e., if the temperature for one filament is 240°C and the other is 200°C, set the hotend to 220°C.)
2) When the hotend’s optimal temperature is reached, push the new filament down into the feed hole and through the hotend, until a small amount of filament comes out of the tip of the nozzle. This may require a firm push.
3) 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.
4) With the new filament now in, Extrude until you see the new color coming out of the hotend. We recommend setting the Extrusion amount to 10.
5) After the extrusion is finished, wait 10 seconds for the extruded filament to cool before removing it from the hotend (with tweezers).
6) You are now ready to print!