BOULDER LABS LIBRARY
NIST | NOAA | NTIA

3D Printer Manual

Anna McDevitt
Anna.McDevitt@noaa.gov
(303) 497-3003

DSRC GB301

Contents

Introduction
The Boulder Labs Library self-service 3D printer is a **shared use resource for work-related projects in the research phase**. These include prototyping, data visualization, and creating research apparatus. It is required to attend a Library 3D printer Training prior to using the 3D printer.

First Come First Served Model
1. Once you have completed the 3D Printing Orientation/Safety Training, go to GB301 to print. If no one is using you, you are free to use the 3D printer and/or computer kiosk.
2. DSRC GB301 is locked. You will need to go to the office of Stacy Bruss (DSRC GB134) or Diann Cullen (DSRC GB304A). One of them will check that you have attended a training to ensure you are authorized to use the 3D printer in order to unlock the room. Please keep in mind that the door automatically locks; so if you need to run to the bathroom and need to return to the 3D printer, you will need to notify either Stacy or Diann to let you back into DSRC GB301. Do NOT leave the door ajar at ANY TIME.
3. Enter your name, phone #, department/division, date, time started, time ended, and purpose/ description of item printed to the Log.
4. Remain with your print for AT LEAST the first layer of its printing.

Finding/Creating Printable Models
Use **thingiverse.com** to find free 3D printable models that have been created by others or **tinkercad.com** to create your own. Cura Lulzbot Edition is used to ensure your print is compatible with the specific printer the Library offers.

Compatible File Types

The preferred file type is .stl (Stereolithography), but you can also use an .obj file if necessary.

To reduce time spent in the 3D printing room (to allow other patrons to use the printer), prepare a 3D model before coming to the Library to print. Save the 3D model as a .stl file, exported in mm units. We suggest emailing the file to yourself so that you can access it at the 3D printer computer kiosk.

Safety First

Follow the safety protocols when working in and around the 3D printer. Failure to comply with these safety protocols may revoke your authorization to use these printers.

3D Printers Have Moving Parts—Avoid Hazards!

Do not place objects or body parts within the printer. Do not reach into a printer in operation. Secure long hair, scarves, and lanyards to prevent these from being caught in moving parts. The printer uses **stepper motors** that moves in discrete steps and allows for precise positioning and/or speed control.

Extruder and Build Platform are Hot!

Never touch a hot extruder or the heated build platform after the print job has begun. Wait until the print job is over and the platform has moved forward before removing your printed object. If you feel heat from a printer component, do not approach any closer. **The 0.50 mm nozzle** can reach **300+°C.** The 11" x 11" Borosilicate Glass/Polyetherimide (PEI) bed reaches about **120°C** when printing.

Tools are Sharp—Use Gloves!

If you need to use the putty knife to remove an object from the build platform, always use the provided gloves and direct the sharp edge of the tool away from body parts. Use proper tools to remove excess plastic from a printed object.

Beware Static Electricity

Touch the frame of the printer before operating any buttons. In our dry climate with carpet, we can build up static electricity that can short out electrical components.

Creating a Print File

Cura Lulzbot Edition (Cura)

Cura (https://www.lulzbot.com/Cura) creates 3D print file, which contains the instructions for the 3D printer to recreate the printable model as an object. Cura is easy to use and is installed on the computer kiosk that is also in GB301.

Other software products can generate .gcode 3D print files, just make sure it is compatible with the Taz6 model (https://www.lulzbot.com/store/printers/lulzbot-taz-6).

1. Open Your File

Make sure that the platform on the screen is cleared. From the top File tab or the left side Open File tab, select the file you want. Access your file via webmail, SD card, or FAT32 formatted USB. Make sure that the **Material** (right side menu) is set to **PLA Verbatim** (or to the filament type and brand you are using).
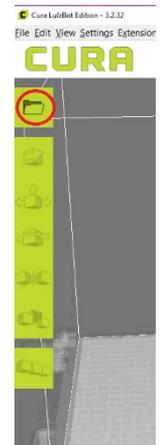
2. Manipulating Objects on Build Platform

Scale

To change the size of the object, click on the object and use the **Scale** button. If you type in the desired size (or percentage change) for one axis and the whole object uniformly scales. Note: Models usually import the correct sizing in millimeters.

Rotate

You may need to use the Rotate button to ensure the best print outcome. The colored rotation lines correspond to the following axis: red=X, green=Y, blue=Z. It is recommended to type in the rotation amount in the rotate box, and to use 90-degree units.
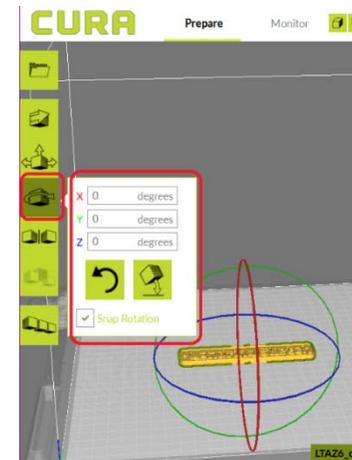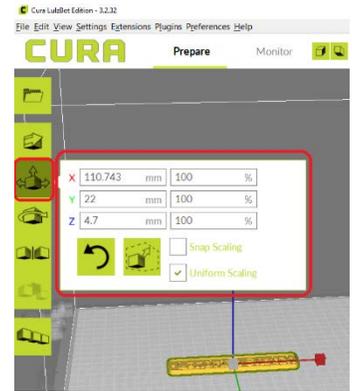
Quick Tips:

- Minimize curves in the Z (vertical) axis as much as possible. Curves in the horizontal axis are much smoother.
- Avoid the largest, flattest side in the Z axis. This will produce the highest quality and avoid potential printing issues.
- Avoid tall, thin prints with no wide base. These objects are liable to tip over during printing.
- The default of the **High Speed** profile is usually sufficient. This helps determines the thickness of each layer. However, if the sides of the object are very curvy, you probably want to use **High Detail**. You can change this setting on the right side menu.
- You can create removable supports for the object (under the right side menu options) by selecting **Brim** under **Build Plate Adhesion** or checking off **Generate Support** to create supportable scaffolding. The latter is especially desirable if there is overhang, which is highlighted in red.
- You can choose **Solid or Layer View** to preview how your design will print.
- To see how long the print is estimated to take, review the time in the lower right hand corner of the Cura dashboard.

3. Save Your File

Use the Library provided SD card or your personal SD card or USB (formatted to FAT32) drive to save the **.gcode file** of your print. The bright green **Save to File** (or Save to Removable Disk) button is located in the lower right corner of the Cura dashboard. The SD card read is on the right side of the computer. To access it, click on the folder labelled LIBRARY_3DP in the upper right corner of the computer screen.

If you want to save your file for **future use**, make sure you either email it to yourself or save it on a personal USB or SD drive. Your files will automatically be removed after you logoff or the computer is shutdown. The computer automatically logs off after some time.

Note: Add about 5 extra minutes to Cura estimate for a better estimate of the total print time.

Checking & Turning on the Printer
Check that the correct filament color is loaded in the printer. If not, change it or ask for it to be changed if you are uncomfortable. Check the build platform is clean; if not, use the microfiber cloth to remove any residue, such as oil, excess filament or dust. Turn on the printer with the **power button**.

Inserting SD Card & Starting Print Job
Ground yourself by touching the metal frame of the printer.

Insert the SD card into its slot with the **label facing away from you.** Confirm that printer monitor says **Card inserted**.

Push the **control knob** to enter the main menu. Turn the knob clockwise until you are at the last menu item, **Print from SD** then push the knob to select. A list of the files on the SD card will appear; the file most recently added to the SD card should be listed first. Turn the knob as needed to highlight the desired print file, then push the knob to select.

At the beginning of each print job, the printer will conduct a check of the build platform. It cleans the nozzle then tests the distance between the nozzle and the platform at each corner. Then, the printer will automatically start heating the nozzle and build platform. This process can take up to 6 minutes. **Keep out of the build area from the start of the calibration through the completion of the print.**

Watching the Beginning of a Print Job
The Cura software should have the correct temperature, but check to make sure the nozzle temperature is in the range of 190-200 degrees C (indicated on the monitor). Otherwise, the filament can crystallize if it does not properly cool down after extruding.

Print failures usually occur in the first layer. Therefore, stay at the printer until it finishes printing the first layer. Look for filament extruding evenly along its entire path and adhering to the platform. Also listen for loud (thumping or grinding) noises.

Please stay at the Library for short print jobs (less than 30 minutes), or return once an hour during business hours to check on a longer print job. Long print jobs (>6 hours) should be started late in the day (after 2:30 pm) so they run overnight; please return first thing the next morning to remove the object.

Stopping Print Jobs
If you notice an issue with the print job or with the object as it is printing, you can stop the job. Push the **control knob** and turn it clockwise until **Stop Print** is highlighted, then push the knob to select. The printer will automatically move the nozzle up and the build platform forward. Remember that the build platform will be hot; let it cool before removing any objects.

If a problem occurs in the printing, cancel the print job. To cancel a running print job, press the control knob on the printer to access the printing menu. Rotate the knob clockwise until the Stop Print menu item is highlighted, then press in the knob to select. If the damaging is quickly impending and there is not enough time to do this, turn off the printer. Please note that the nozzle costs ~$200-$500 to replace, and the printer as a whole costs $2,500.

Please make sure Library staff are aware of the failure.

Removing the 3D Print
When the print is complete, the bed will move to the front of the printer. At this point, you can remove the print.

Most prints should be easy to remove from the bed using your bare hands. Gently twist the object to pop the seal between the object and the platform. **This is the goal: to break the seal.**

If you are experiencing difficulty, break the vacuum seal between the object by using the **putty knife** to get under a section of the 3D object and twisting the knife. **Do NOT scrape**, simply getting under and lifting a corner of the object.

When you remove the print using the putty knife, you **MUST**:
1. Wear gloves – the Library offers both small- and large-sized pairs
2. Hold the bed with the your other hand while using the edge of the putty knife.
3. Direct the putty knife away from your body and keep it flat against the build platform to avoid damaging the platform.

Post-Processing & Wrap-Up
There may be excess plastic on your object from the printing process. Use your fingers as much as possible. **Do not** use the putty knife as a finishing tool. If necessary, use the Library supplied needle nose pliers and tweezers. To smooth out the sides of PLA, you can use Future Floor Finish. Please place excess filament material into the compost bin.

Filament
Make sure the correct filament (plastic) brand in the **Material** drop-down in the Cura software.

The Library offers select colors of **PLA (polylactide)** filament. PLA is corn starch based, biodegradable, is made from renewable resources so few greenhouse gases are released while printing,

The Library offers the following PLA colors: clear, translucent blue, translucent red, green, silver, white, and black. For most projects, the filament will be **Verbatim PLA**. The special clear filaments are **PolyLite PLA**.

You are free to use other compatible material, such as Nylon or ABS. However, if you do, you accept responsibility for the material and ensuring any troubleshooting issues are known to staff. You must contact Anna McDevitt before you print, if you plan to use a material besides PLA.

Place the filament in the ziplock bag and place in the rolling filing cabinet once you are done. If the filament is not put away, it will absorb the moisture and ruin the filament.

Change (Remove and then Add) the Filament
If you need to use another color of filament or if you need to shut off the computer if you are the last user for the day, select **Change filament** from the monitor.

You will be prompted to remove the filament once the nozzle is heated up. **To remove**, pull the filament upward, hold onto the spool holder, and hold onto the end of the filament simultaneously.

Pull the spool through the top bars of the metal frame, use some "elbow grease", and pull out quickly.

Place the filament you no longer wish to use is in a 2-gallon ziplock bag in the rolling filing cabinet. Place your desired filament onto the spool holder and **add it** to the metal frame, ensuring that the end of the filament is facing the nozzle.

Push in the filament into nozzle head and press the knob on for the printer monitor when the monitor notifies you that it is ready to be changed out. Lightly push until you feel the filament catch. You do not need to use as much force putting the filament in as you may have when you took out a filament.

You will have 2 options:
1. **Purge more** – Select this UNTIL you see the color of the filament come out of the nozzle properly. You generally have to select this 2-3 times.
2. **End filament change** – Select once you can see the proper colored filament come out of the nozzle.

Appendix A: Tips for Orienting Objects for Optimum 3D Printing

Facing Delicate Features Upward
To protect delicate features or critical surfaces, rotate the object so these surfaces face upward. Removing brims or support structures from the bottom surface may be difficult if it is intricate.

Placing Curves on XY Plane
Curves reproduce best in the XY plane. If there are any important curved surfaces, such as a cylinder, the curves reproduce most accurately if the cylinder is upright rather than lying on its side. Curves in the XZ or YZ planes will have a stair-step effect because of the layer-by-layer process the printer uses to approximate a curve.

Laying Long, Thin Structures on Their Side
Avoid orienting high-aspect ratio structures (i.e., long, thin ribs) vertically by laying these structures on their side. These may fall over during the build due to the lateral force exerted by the 3D printer's moving nozzle.

Aligning Tensile Forces with XY Plane
If strength is important, be sure to rotate the object so that applied tensile forces align with the object's horizontal placement (along the XY plane). Printed objects are weakest in the Z-direction as layers may delaminate when tensile forces are applied.

Minimizing Overhangs
Overhanging features (i.e., hanging over 70 degrees from previous layer) will often require support structures during printing. Rotating the object to minimize overhangs will reduce the amount of support structures used.

Appendix B: Tips for Selecting Print Quality Settings

Layer Height (mm)

The vertical resolution of the printed object. Changing the vertical resolution is often necessary only when there are fine vertical features or slanted/rounded features in the Z axis, such as threaded objects, data visualizations, or molecular models. Layer height can be changed by changing print profiles, or can be changed through custom print setup.

0.38 mm is the default layer height and is appropriate for most objects. 0.075 mm is the minimum size available on the printer, but 0.1 mm is the minimum recommended layer height. 0.4 mm is the maximum recommended layer height, for draft versions of an object or for fast printing; delamination during printing can occur with layer heights greater than 0.4 mm. Note: straight vertical surface quality is not materially affected by layer height. Decreasing the layer height proportionally increases the print time.

Infill Density (%)

The percentage of plastic that fills the hollow interior of the object. The software will generate a vertical grid structure during the slicing process with a size relative to the selected density. Default value is typically 20; the density should usually not be less than 20 for successful printing. The maximum recommended density for strength is 50. However, objects used in vacuum applications, thin-walled objects, or those that will have holes machined in them or cut should be printed at 100 (solid object). An increase in fill density will increase the print time of an object.

Infill pattern: The default Cura infill pattern is grid. Depending on the geometry or forces to be applied to your part, you may wish to change the infill pattern. The options available include:

Shell wall thickness (mm): The thickness of vertical walls. Default value is 1. Use 0.5 for objects with fine inset features on the top surface. Use a higher shell thickness value (1.5-2.5) to make solid small pegs or other fine features that will not be filled (the printed size will not exceed the modeled size of a feature even if the thickness of the shells is greater). Use a shell value of 2 in conjunction with a higher fill density to create a stronger object. Shell thickness values typically do not affect print time.

Detail Level

There are three detail levels available, adjusted using the Profile drop-down box. The detail level adjusts the height (in the Z axis) of each layer. The thicker the level selected, the shorter the print time will be. The thickness of layers are shown next to each detail level in the drop-down box.

Most objects can be printed with the High Speed detail level, giving the fastest print time. Objects with curves in the Z axis or spirals will look smoother with the High Detail detail level, but will take twice the time to print than High Speed.

Do not make the size smaller than 0.50mm, since the nozzle is this size.

Appendix C: Tips for Using Support Structures & Brims

Support Structures

Support structures are needed in cases where overhangs cannot be printed without support. These structures are a lattice of plastic automatically added by the software during the slicing process where needed if selected in the print setup with a checkbox in Generate Supports.

If support structures may be needed, first attempt to print the object without support structures to determine the print quality. If the structure is small and/or rounded, support structures will be easy to use and remove. However, support structures are difficult to remove from interior cavities and from large, flat surfaces. For these types of objects, consider support structure alternatives.

Support Structure Alternatives

Instead of using the 3D printing software to add support structures, try designing them as part of the object:

- chamfer edges to not exceed 70 degrees
- build in a light scaffold for overhangs; narrow the interface between the scaffold structure and the object to be 0.5 mm wide (the minimum dimension that can be printed)

Brims

Because of the unique surface of the TAZ build platforms, most print jobs will not require a brim. A brim is a layer of plastic printed adjacent to the footprint of an object for enhancing platform adhesion without leaving any marks on the print or for keeping objects with a small footprint from tipping over.

Brims are selected in print setup under Build plate adhesion. A brim may be required for:

- Small parts with little surface area on the print platform
- Any noted warping Brim Alternative

Instead of a brim for large or warping parts, try adding thin (2-3 layers thick) 10mm diameter tabs to the corners of the object. These are effective, easy to remove, and take less time to print than a brim.