FEED YOUR MACHINE

LulzBot® 3D printers are compatible with many different 3D printing filaments; not only can you 3D print with ABS and PLA, but also with metallic, stone-like, wooden, nylon, and even flexible materials. 3D print with confidence knowing that each filament option has been tested and integrated into Cura LulzBot Edition, the recommended software for your LulzBot® 3D printer.

LULZBOT.COM/FILAMENT

ISBN 978-0-9893784-8-2
90000
9 780989 378482

LULZBOT®

ALEPH OBJECTS®
INCORPORATED

LulzBot® TAZ 6.0 User Manual

Aleph Objects, Inc.

LULZBOT®

TAZ 6

User Manual

PROOF

7.44 x 9.69
246 mm x 189 mm

.186
4.7244mm

7.44 x 9.69
246 mm x 189 mm

Content Type: Standard Color
Paper Type: STCO19 (Standard 50 White)
Page Count: 90
File type: Internal

Perfect Bound Cover Template

Lightning Source

Document Size: 19" x 12"
305 x 483mm

9 780989 378482

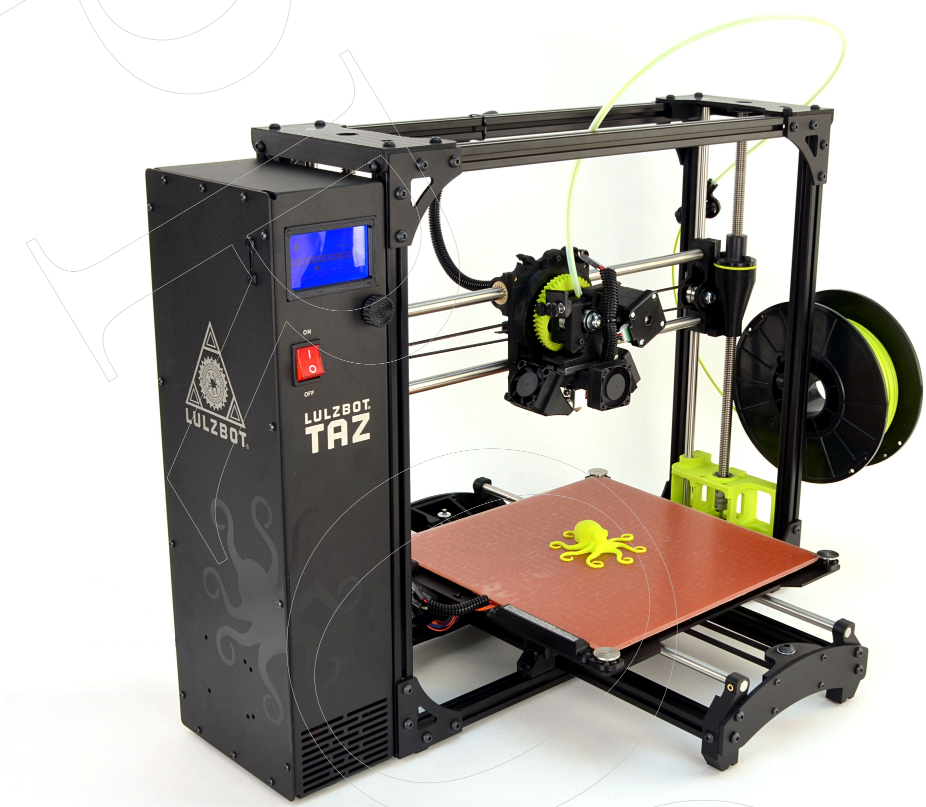
LULZBOT®

TAZ 6

User Manual

ON
OFF
LULZBOT
LULZBOT
TAZ

LULZBOT TAZ 6 USER MANUAL


ON
OFF
LULZBOT
LULZBOT TAZ

LULZBOT®

Aleph Objects, Inc.

DRAFT

LulzBot® TAZ 6 User Manual

by Aleph Objects, Inc.

Copyright © 2016 Aleph Objects, Inc.

Permission is granted to copy, distribute and/or modify this document under the terms of the Creative Commons Attribution 4.0 International Public License (CC BY-SA 4.0).

Published by Aleph Objects, Inc., 626 West 66th Street, Loveland, Colorado, 80538 USA.

For more information, call +1-970-377-1111 or go to LulzBot.com and AlephObjects.com.

ISBN: 978-0-9893784-8-2

6.0-20160322

PROOF

Contents

WARNINGS

Safety Information vii

Read Me First! viii

Hazards and Warnings viii

Regulatory Compliance Statement Class B ix

1 3D Printer Software 11

1.1 Software Overview 12

1.2 Software Types 12

1.3 Installing Drivers 13

1.4 CAD and 3D Modeling Software 13

1.5 Alternative Printer Host Software 14

2 Cura LulzBot Edition 17

2.1 Cura 18

2.2 Quick Print Settings 19

2.3 View Options 23

2.4 Starting Your First Print 27

2.5 Removing Your First Print 30

2.6 Full Settings 31

2.7 Basic Tab Options 32

2.8 Advanced Tab Options 36

2.9 Plugins 39

2.10 Start and End GCODE Settings 41

2.11 Expert Settings 42

2.12 Retraction 42

2.13 Skirt 43

2.14 Cool 44

2.15 Support 45

PROOF

2.16 Black Magic 46
2.17 Brim 46
2.18 Raft 46
2.19 Fix Horrible 48
2.20 Dual Extrusion 49

3 Printing with the Graphic LCD 55
3.1 Graphical LCD Controller, Cura or Other Host? . 56
3.2 Multiple Connections 56
3.3 Putting Print Files on the SD Card 57
3.4 Printing With the Graphical LCD Controller . . 57
3.5 Configuration Options 60
3.6 Advanced Settings 61

4 Maintaining Your 3D Printer 65
4.1 Overview 66
4.2 Smooth Rods 66
4.3 Lead Screw Drive Rods 66
4.4 PEI Print Surface 66
4.5 Hobbed Bolt 67
4.6 Belts 67
4.7 Hot End 67
4.8 Nozzle Wiping Pad 67
4.9 Bed Leveling Washers 68
4.10 Cooling Fans 68
4.11 Electronics 68

5 Advanced Usage 69
5.1 Intro 70
5.2 Changing nozzles 70
5.3 Bed Adhesion 70
5.4 Using 1.75mm filament 71

6 Hardware and Software Source Code 73

7 3D Printer Support 75
7.1 LulzBot 76
7.2 Support 76
7.3 Regional Phone Numbers 76
7.4 Community 76

8 Warranty Information 77
8.1 `Warranty` 78
8.2 `Modification Warning` 78
9 Contact Information 79
9.1 `Support` 80
9.2 `Sales` 80
9.3 `Regional Phone Numbers` 80
9.4 `Websites` 80
Index 81
Glossary 85

PROOF

List of Figures

2.1 Quick Print Settings . . . 19
2.2 Options after selecting model . . . 21
2.3 Rotating your Model . . . 22
2.4 Scaling your Model . . . 23
2.5 View in Normal Mode . . . 24
2.6 View in Overhang . . . 24
2.7 View in Ghost . . . 25
2.8 View in Xray . . . 25
2.9 View in Layers . . . 26
2.10 Viewing Cumulative Layers . . . 26
2.11 Viewing Specific Layers . . . 27
2.12 Control Screen . . . 29
2.13 View in Full Settings . . . 31
2.14 Differences in Layer Height . . . 32
2.15 Support Types . . . 35
2.16 Cutoff Example . . . 38
2.17 View of Plugins . . . 40
2.18 View Expert Settings . . . 42
2.19 Before Merge . . . 52
2.20 After Merge . . . 52

3.1 GLCD Info Screen . . . 57
3.2 Main Menu . . . 59
3.3 Z Offset Screen . . . 62
3.4 GLCD Map . . . 64

PROOF

WARNINGS
Safety Information

PROOF

Read Me First!

READ THIS MANUAL COMPLETELY BEFORE UNPACKING AND POWERING UP YOUR PRINTER.

Hazards and Warnings

Your LulzBot® TAZ 3D printer has motorized and heated parts. Always be aware of possible hazards when the printer is operational.

Electric Shock Hazard

Never open the electronics case when the printer is powered on. Before removing the electronics case cover, always power down the printer and completely turn off and unplug the printer. Allow the printer to discharge for at least one minute.

Burn Hazard

Never touch the hot end nozzle or heater block without first turning off the hot end and allowing it to completely cool down. The hot end can take up to 20 minutes to completely cool. Never touch recently extruded plastic. The plastic can stick to your skin and cause burns. The print surface can reach high temperatures that are capable of causing burns.

Fire Hazard

Never place flammable materials or liquids on or near the printer when it is powered on or operational. Liquid acetone, alcohol, or other chemicals may release vapors that are extremely flammable.

Pinch Hazard

When the printer is operational take care to never put your fingers near any moving parts including belts, pulleys, or gears. Tie back long hair or clothing that can get caught in the moving parts of the printer.

PROOF

Age Warning

For users under the age of 18, adult supervision is recommended. Beware of choking hazards around small children.

Modifications and Repairs Warning

At Aleph Objects, Inc. we respect your freedom to modify your LulzBot® desktop 3D printer. However any modifications or attempted repairs that cause damage are not covered under the Warranty. Questions? Contact Technical Support by emailing `support@lulzbot.com`, or by calling +1-970-377-1111.

Regulatory Compliance Statement Class B

Federal Communications Commission Statement

CAUTION: Changes or modifications not approved by the party responsible for compliance could void the users authority to operate the equipment.
`NOTE:` This equipment has been tested and found to comply with the limits for a Class B digital device, pursuant to part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference in a residential installation. This equipment generates, uses and can radiate radio frequency energy and, if not installed and used in accordance with the instructions, may cause harmful interference to radio communications. However, there is no guarantee that interference will not occur in a particular installation. If this equipment does cause harmful interference to radio or television reception, which can be determined by turning the equipment off and on, the user is encouraged to try to correct the interference by one or more of the following measures:

1. Reorient or relocate the receiving antenna.
2. Increase the separation between the equipment and receiver.
3. Connect the equipment into an outlet on a circuit different from that to which the receiver is connected.
4. Consult the dealer or an experienced radio/TV technician for help.

Industry Canada Statement

Cet appareil numérique de la classe B est conforme à la norme ICES-003 du Canada. This device complies with Canadian ICES-003 Class B.

Australian Communications and Media Authority Statement

This device has been tested and found to comply with the limits for a Class B digital device, pursuant to the Australian/New Zealand standard AS/NZS CISPR 22:2009 + A1:2010.

PROOF

3D Printer Software

PROOF

1.1 Software Overview

To operate your desktop 3D printer you will need to install a few software packages onto your PC. You will need a 3D printer host, an `.STL` to `.GCODE` generator, and optional CAD or 3D modeling software.

`Cura LulzBot® Edition is the recommended software for your LulzBot 3D printer.` Download Cura LulzBot Edition by visiting `LulzBot.com/Cura`.

All of the following Free/Libre Software packages are available for GNU/Linux, Windows, and OS X. We highly recommend using these programs on GNU/Linux.

1.2 Software Types

Printer Hosts

Printer Host software is used to control the 3D printer. The program not only allows you to manually move the printer along all the axes, but set temperatures manually, send commands, and receive feedback/error messages from the onboard electronics. We recommend that new users start with Cura LulzBot® Edition as it includes a slicing engine as well.

`Slicers`

These programs take the 3-Dimensional model (typically STL/OBJ/etc) and determine the 3D printer toolpath based on the options selected. The slicing engine uses the nozzle diameter, movement speeds, layer height, and other variables to determine the coordinates where it needs to move, and the rates at which it will do so. This information is exported out of the program as a GCODE file. The GCODE file is a plain-text file with a series of text-based codes and a list of the complete X,Y, and Z-axis coordinates used for printing the 3D model. We recommend that new users start with Cura LulzBot Edition as it includes the printer host as well.

DRAFT

1.3 Installing Drivers

GNU/Linux and OS X users will not need to install a driver to communicate with the LulzBot TAZ 3D printer. Windows users will need to install the drivers. Using Cura LulzBot Edition as your printer host and slicing software is recommended, as the drivers will automatically be installed during the Cura installation process. Download Cura LulzBot Edition by visiting `LulzBot.com/Cura`. The drivers can also be downloaded from `LulzBot.com/downloads`. A visual guide showing the driver installation process can be found in our download section as well.

1.4 CAD and 3D Modeling Software

LulzBot is not distributing a CAD or 3D modeling software package. However, multiple Free/Libre Software packages are available. Other common non-free CAD and 3D modeling software are also capable of exporting the required `.STL` files.

On some CAD and 3D modeling software you will need to select millimeters as the output unit. If possible it is best to build your 3D design in metric units rather than imperial units. Cura requires .STL files sized in millimeters. If an .STL with inches as units is loaded into Cura, the model will be scaled much smaller than expected. You can scale the model by `25.40` to compensate. The software listed below outputs millimeters as the unit by default.

FreeCAD

Website: `http://www.freecadweb.org/`

Although still in development, contains a full GUI for building CAD models. FreeCAD is capable of creating simple to complex designs. STL files can also easily be exported for use with 3D printing. FreeCAD is available for GNU/Linux, Windows, and OS X. The latest development version is recommended.

OpenSCAD

Website: `http://openscad.org`

OpenSCAD is different than FreeCAD in that it is script based. Rather than using a GUI to generate CAD designs, OpenSCAD CAD designs are created using script based renderings. Users with programming experience would find this useful. Also, OpenSCAD uses a simple script language that is easy for users with little or no programming experience to learn.

Blender

Website: `http://blender.org`

The most widely used Free/Libre Software 3D modeling software, Blender is well documented with tutorials available on the Blender.org website as well as found online.

Shapesmith

Website: `http://shapesmith.net`

Shapesmith is a web-based 3D modeling software. This means there is no required software to get started designing models. Shapesmith is also a great choice for anyone starting out in CAD/ 3D modeling.

1.5 Alternative Printer Host Software

OctoPrint

Website: `http://octoprint.org/`

Octoprint is a printer host that uses a web-based interface to access and control your 3D printer. Added web-cam functionality allows for time-lapse videos and a live stream. Octoprint will run on GNU/Linux, Windows, OS X based computers and can even run well on a Beagle Bone Black or a RaspberryPi (inexpensive business-card sized computers).

BotQueue

Website: `https://www.botqueue.com/`

BotQueue works well for those users wanting to have a web-based multiple 3D printer operation running off a queuing system.

MatterControl

Website: `http://www.mattercontrol.com/`

MatterControl is another printer host that currently runs on GNU/Linux, Windows, and OS X. It features 2D and 3D model viewing, a print queue, and print file organization and searching.

Source Files

Aleph Objects, Inc., the maker of the LulzBot® TAZ 3D printer, completely supports Free Software, Libre Innovation, and Open Source Hardware. Along with the LulzBot TAZ 3D printer being a Free Software and Open Source Hardware design, it has been tested to work with 100% Free/Libre Software. Our source code and design files are hosted on:

LulzBot Download Server `http://download.lulzbot.com`

LulzBot Development Server `http://devel.lulzbot.com`

Aleph Objects Code Repository `http://code.alephobjects.com`

Aleph Objects Download Server `http://download.alephobjects.com`

Aleph Objects Development Server `http://devel.alephobjects.com`

PROOF

PROOF

Cura LulzBot Edition

PROOF

2.1 Cura

Setup Cura

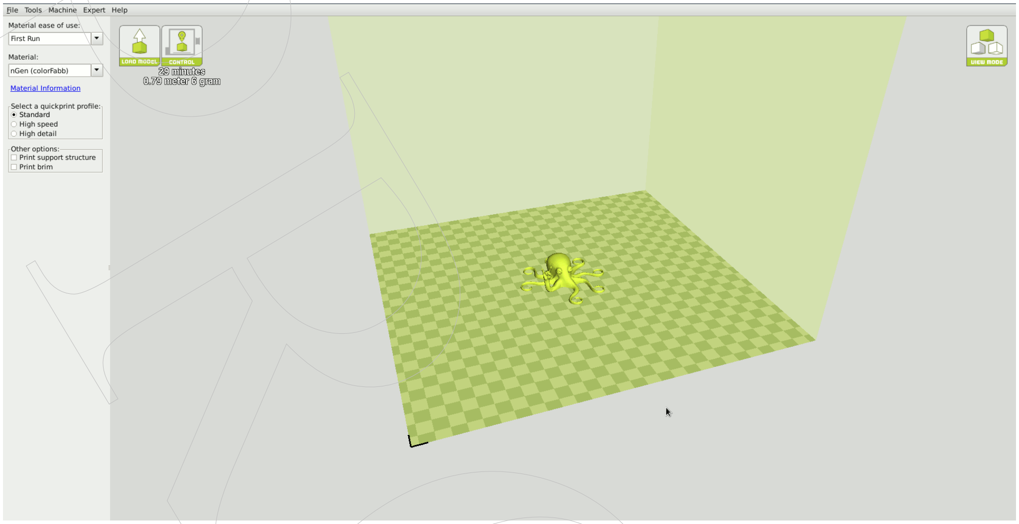
Cura LulzBot Edition is available for download on our website at `http://LulzBot.com/cura`. When installing, it is recommended to uninstall any previous versions of Cura you may have been using. Cura is designed for Fused Filament Fabrication (FFF) 3D printers. Fused Filament Fabrication is the term for the process of laying down successive layers of extruded filament to create a 3 dimensional object. As each layer of molten plastic is extruded into place, it fuses with the previous layer. When first opening Cura, you will be prompted to go through the `First run wizard`. This will consist of selecting your printer, hot end type, tool head type, and finally your nozzle diameter.

It is important to select the correct printer, hot end, tool head, and nozzle diameter as Cura uses custom profiles and machine settings based upon which printer, hot end, tool head, and nozzle you have.

- Download the appropriate installer for your computer operating system. Instructions on installation for each operating system are available at `http://LulzBot.com/cura`.
- Install Cura by double clicking on the installer.
- Click through the install wizard until it completes.
- Start Cura by launching it from your list of installed applications. If this is the first time that Cura has been used the “Configuration Wizard” window will open.
- Select `LulzBot TAZ 6`. Press `Next`.
- Select `Single Extruder v2.1`. Press `Next`
- Select finish.

Draft

2.2 Quick Print Settings



Material ease of use: First Run

Material: nGen (colorFabb)

Select a quickprint profile:
● Standard
○ High speed
○ High detail

Other options:
☐ Print support structure
☐ Print brim

29 minutes
0.79 meter 6 gram

Figure 2.1: Quick Print Settings

After setting up Cura for the first time, you will be shown the main interface screen. (Fig. 2.1, page 19)

Selecting a Quick Print Profile

The print quality settings can be found in the top left-hand corner of the window. For most filaments, there will be `Standard`, `High Speed`, and `High Detail` options. Some of the more exotic filaments may only have a single profile.

High Detail
Designed to give greater detail and finer objects. This will have a smaller layer height, which will make each layer thinner, so that curves seem more natural and walls seem less noticeable. This setting will also require more layers to be laid down, increasing overall print time.

Standard
Designed to give a balanced resolution by increasing the layer height

and print speeds. This will make the organic curves slightly more step-like than the fine setting, but it will reduce printing time.

`High Speed`
Designed for the fast prints, where the overall model finish is not of concern. Most commonly used for quick iteration of designs when rapid prototyping.

Material Selection

We have the different filaments separated by `Material ease of use.` From the `Material ease of use` drop down, select `"All"` to view all our pre-loaded filament slicing profiles. The TAZ ships with a filament sample for the first print. Refer to the included Quick Start Guide for the proper "First Run" settings.

Printing Support Material

The LulzBot TAZ 3D printer is able to print models that have angles and overhangs, even without support material. This will depend on the overhang distance and angle of your particular model file. Turn this option on if your model could benefit from support material.

Brim

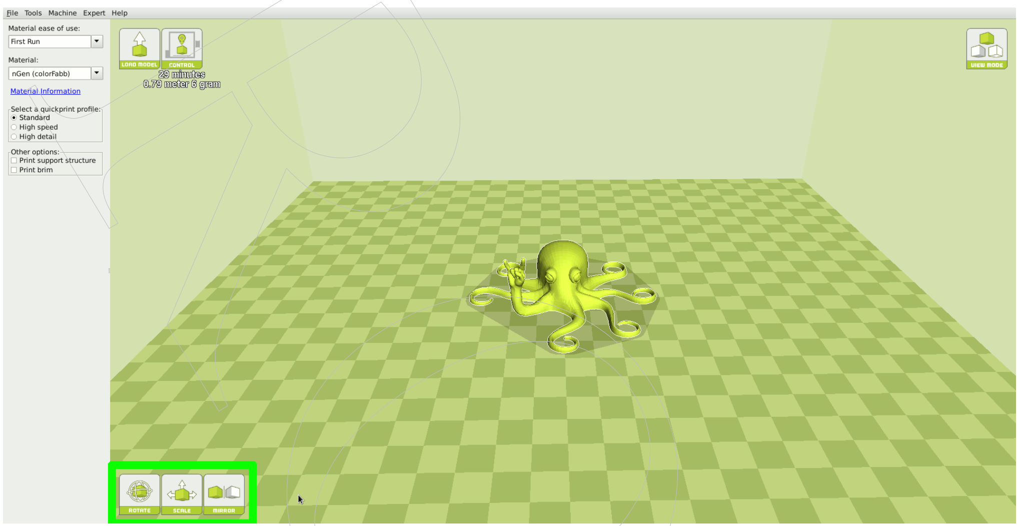
Brim is used to increase surface area of the part you're printing, thereby ensuring proper part adhesion. This will print a single layer high edge around the base of the part, helping first layer adhesion and minimizing warping.

Load Model File

Select the STL model you would like to print. Either use the `Load Model` button or select `File` > `Load Model`. Once the file has been loaded, you will see a 3D rendering of your object on the build platform. Select the model to see the various options.

Model Orientation

Move your model to change where it is printed on the build plate. Do this by left clicking on the model and dragging it to the desired location. The `black` outlined corner of the 3D print bed view represents the front left hand corner of the build plate on your printer. By holding down the right mouse button and dragging, you can view your model from different angles.



Material ease of use: First Run

Material: nGen (colorFabb)

Material Information

Select a quickprint profile:
● Standard
○ High speed
○ High detail

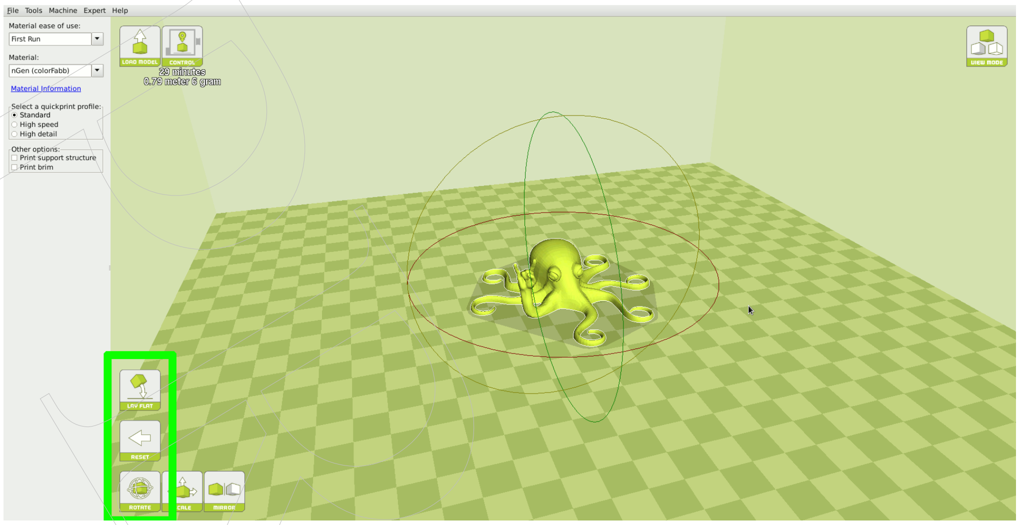
Other options:
☐ Print support structure
☐ Print brim

29 minutes
0.79 meter 6 gram

Figure 2.2: Options after selecting model

Rotate

The `Rotate` button will give you the ability to orient your model in along all three axes. Once you click the rotate button, three circles will surround your model. The red circle will allow you to rotate around the Z-axis. The Yellow circle will rotate around the Y-axis. The Green circle will rotate around the X-axis. Cura defaults to 15 degree increments. Hold `Shift` to rotate by `One Degree Increments`.



Material ease of use: First Run

Material: nGen (colorFabb)

Material Information

Select a quickprint profile:
● Standard
○ High speed
○ High detail

Other options:
☐ Print support structure
☐ Print brim

29 minutes
0.79 meter 6 gram

Figure 2.3: Rotating your Model

`Lay Flat`

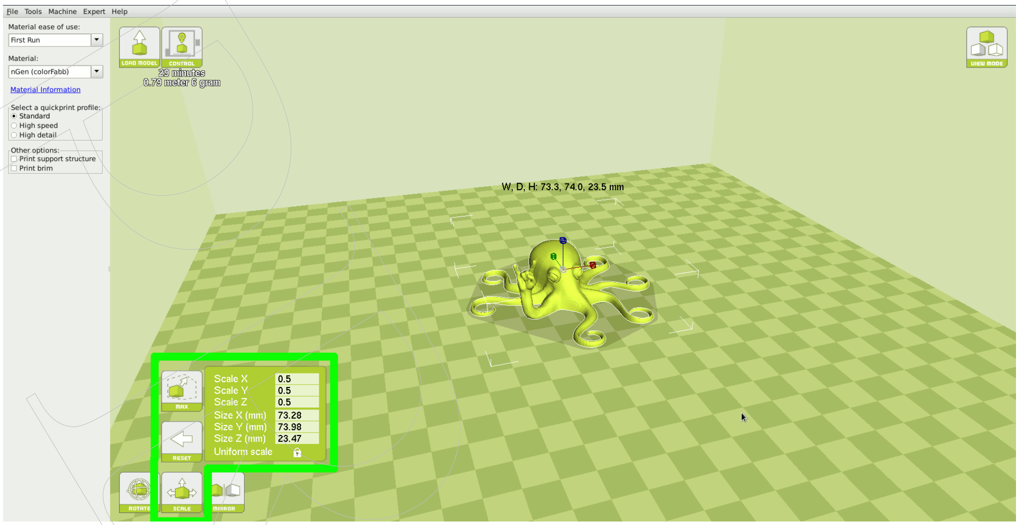
The `Lay Flat` button will ensure that the flat portion of your print is securely attached to the bed. It is highly recommended to use this option after rotating your model in the Z direction, as it will help prevent potential adhesion issues during the print.

`Reset`

The `Reset` button will return your model to the original orientation as defined by the CAD program used to create the model.

`Scale`

The `Scale` button displays the model dimensions, along with the ability to scale along the X Y or Z axes. Anything `below` the number `1.0` will reduce the objects size, while anything `above` the number `1.0` will increase the objects size. As a default, it will be set to uniform scaling. This will cause the X Y and Z axes to be scaled by the same amount when you make a change to any of them. To disable this, select the lock in the lower section of the scaling window.


File Tools Machine Expert Help
LOAD MODEL
CONTROL
VIEW MODE
MAX
RESET
ROTATE
SCALE
MIRROR
W, D, H: 73.3, 74.0, 23.5 mm

Material ease of use: First Run

Material: nGen (colorFabb)

Material Information

Select a quickprint profile:
● Standard
○ High speed
○ High detail

Other options:
☐ Print support structure
☐ Print brim

29 minutes
0.79 meter 6 gram

Scale X	0.5
Scale Y	0.5
Scale Z	0.5
Size X (mm)	73.28
Size Y (mm)	73.98
Size Z (mm)	23.47
Uniform scale	

Figure 2.4: Scaling your Model

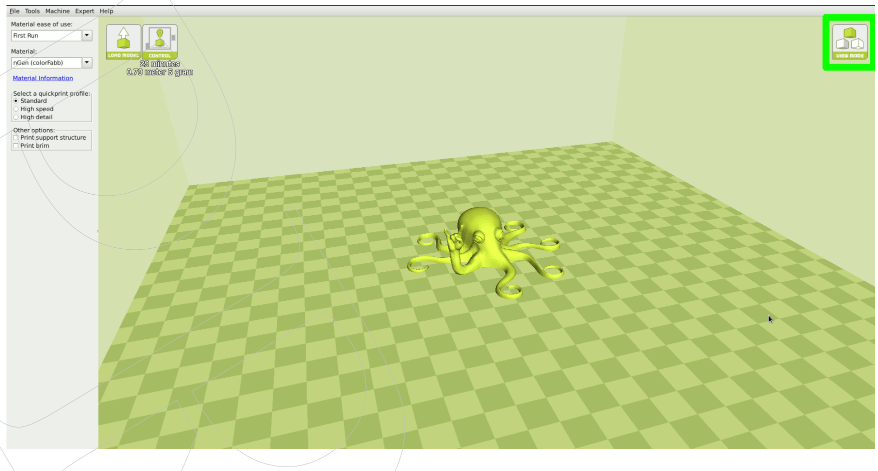
2.3 View Options

Different modes allow you to view your model in a variety of ways. This can be helpful for spotting issues before the print even starts.

Normal

This is the standard view and shows the solid outer surfaces of the model. (Fig. 2.5, page 24):

PROOF


File Tools Machine Expert Help
LOAD MODEL
CONTROL
VIEW MODE
Material Information

Material ease of use: First Run

Material: nGen (colorFabb)

Select a quickprint profile:
● Standard
○ High speed
○ High detail

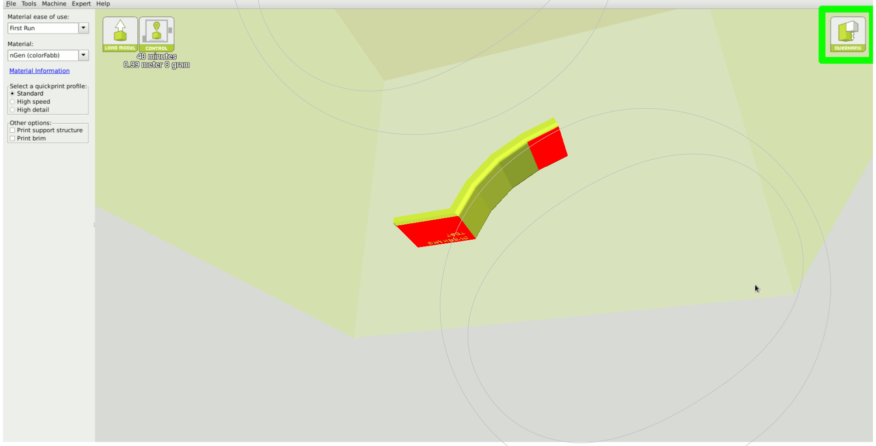
Other options:
☐ Print support structure
☐ Print brim

29 minutes
0.79 meter 6 gram

Figure 2.5: View in Normal Mode

Overhang

Overhang mode shows where your model may need support material. In Fig. 2.6, page 24 the red highlighted areas show overhangs and more severe angles and areas where support material is recommended. The overhang threshold can be defined in Expert Settings.


File Tools Machine Expert Help
LOAD MODEL
CONTROL
OVERHANG
Material Information

Material ease of use: First Run

Material: nGen (colorFabb)

Select a quickprint profile:
● Standard
○ High speed
○ High detail

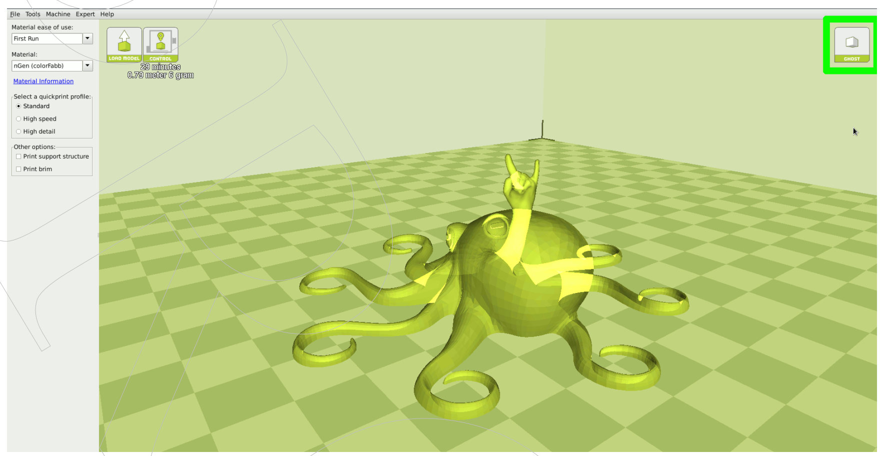
Other options:
☐ Print support structure
☐ Print brim

48 minutes
0.99 meter 8 gram

Figure 2.6: View in Overhang

Ghost

Ghost view mode makes the model translucent to allow you to see what is behind it.

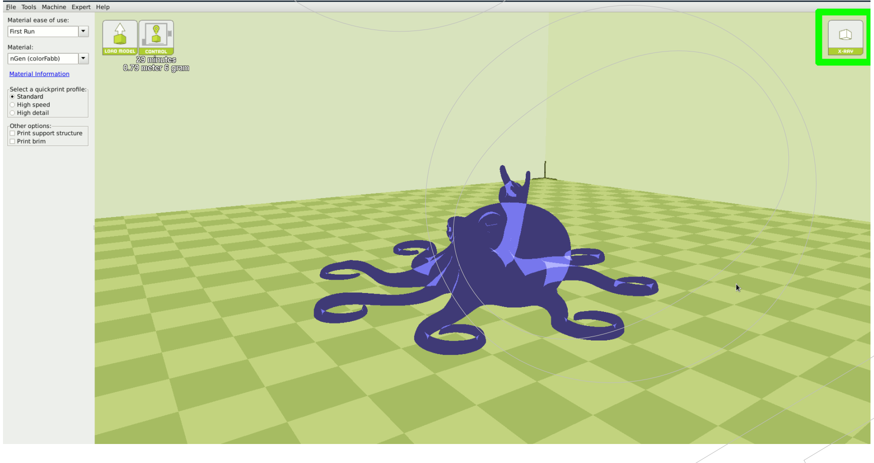

File Tools Machine Expert Help
LOAD MODEL
CONTROL
GHOST
Material Information

Material ease of use:
First Run
Material:
nGen (colorFabb)
Select a quickprint profile:
● Standard
○ High speed
○ High detail
Other options:
☐ Print support structure
☐ Print brim
29 minutes
0.79 meter 6 gram

Figure 2.7: View in Ghost

Xray

Xray is similar to Ghost mode. It will allow you to see into objects, ensuring that inner details are correct.

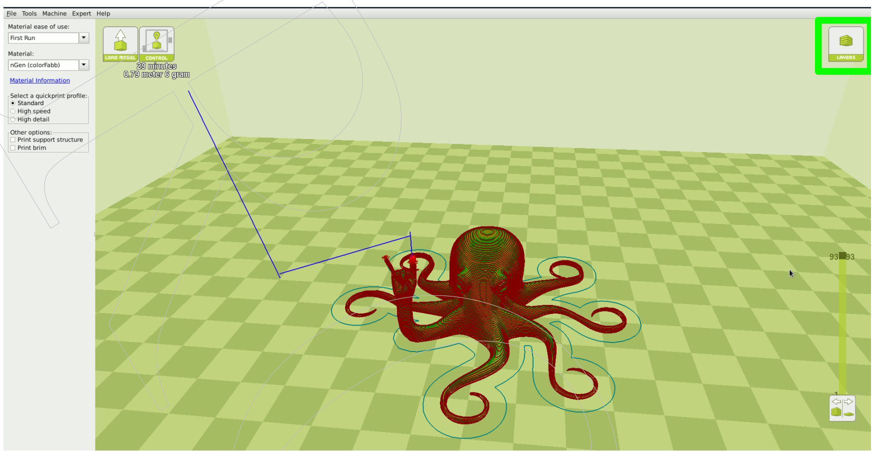

File Tools Machine Expert Help
LOAD MODEL
CONTROL
X-RAY
Material Information

Material ease of use:
First Run
Material:
nGen (colorFabb)
Select a quickprint profile:
● Standard
○ High speed
○ High detail
Other options:
☐ Print support structure
☐ Print brim
29 minutes
0.79 meter 6 gram

Figure 2.8: View in Xray

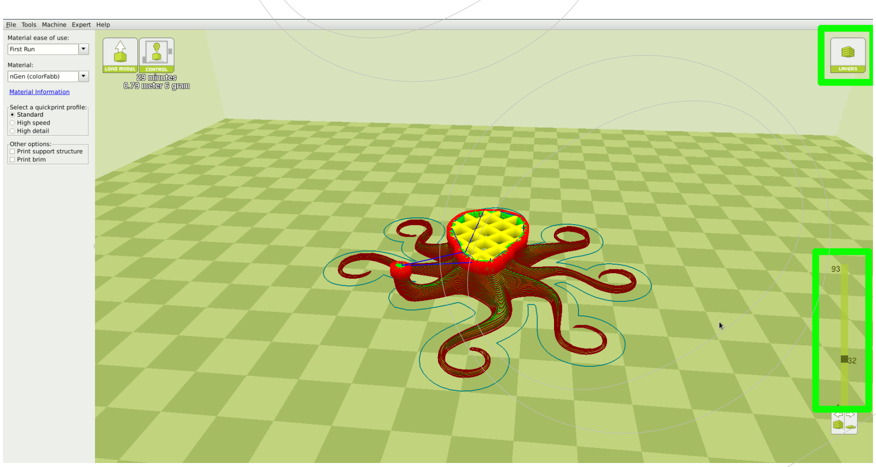
Layers

To view the tool path of your print head and to ensure no skipped layers or gaps use this option. Use the slide bar on the right hand side of the window to move up and down through the tool path layers. Click the icon below it to view an individual layer at a time. If the `Print support structure` option is activated in Cura, it will be shown in blue.


File
Tools
Machine
Expert
Help
Material Information
LOAD MODEL
CONTROL
LAYERS
93
93

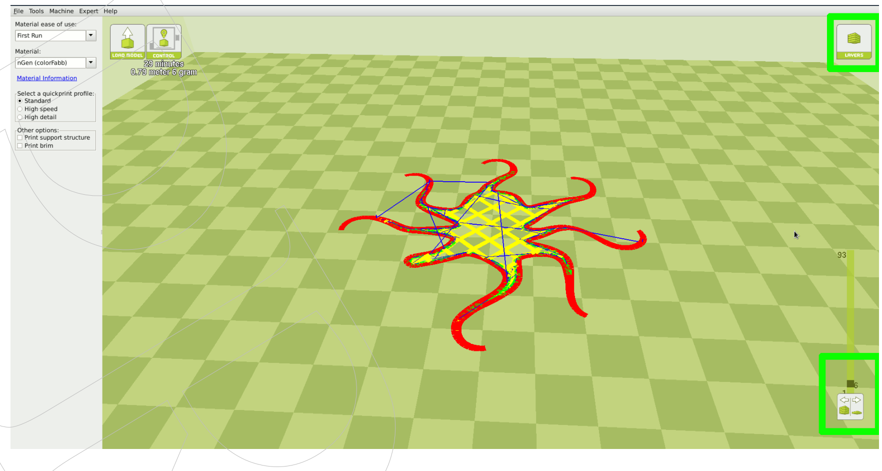
Material ease of use: First Run
Material: nGen (colorFabb)
Select a quickprint profile:
● Standard
○ High speed
○ High detail
Other options:
☐ Print support structure
☐ Print brim
29 minutes
0.79 meter 6 gram

Figure 2.9: View in Layers


File
Tools
Machine
Expert
Help
Material Information
LOAD MODEL
CONTROL
LAYERS
93
32

Material ease of use: First Run
Material: nGen (colorFabb)
Select a quickprint profile:
● Standard
○ High speed
○ High detail
Other options:
☐ Print support structure
☐ Print brim
29 minutes
0.79 meter 6 gram

Figure 2.10: Viewing Cumulative Layers



Material ease of use: First Run

Material: nGen (colorFabb)

Select a quickprint profile:
● Standard
○ High speed
○ High detail

Other options:
☐ Print support structure
☐ Print brim

29 minutes
0.79 meter 6 gram

Figure 2.11: Viewing Specific Layers

2.4 Starting Your First Print

Once you have your model, profile, and filament loaded, it is time for your first print! Refer to the `Quick Start Guide` included with your 3D printer. A PDF version is available at `LulzBot.com/downloads`.

Automatic Bed Leveling

Before each print your, TAZ will go through a wiping and a probing procedure in order to determine the slope/tilt of your bed. If your nozzle is dirty or not cleaned properly, this will prevent the printer from being able to create an electrical contact with the corners. Your TAZ will automatically retry the wiping and probing procedure if this happens, up to a maximum of three times. If three consecutive wipes and probes have failed, the printer will stop and a warning will sound. If this is happening consistently, you should replace the wiping pad and/or adjust your wiping and probing temps for that specific filament. See section 2.10 on page 41 for details on changing these temps.

Printing from SD Card

Load your model into Cura with the proper filament and profile selected. Next, insert the included SD card into your computer. Select `File` > `Save`

`GCODE` and choose the SD card location to save your GCODE file to your SD card. Safely eject the card from the computer, and insert it into the Graphical LCD Controller.

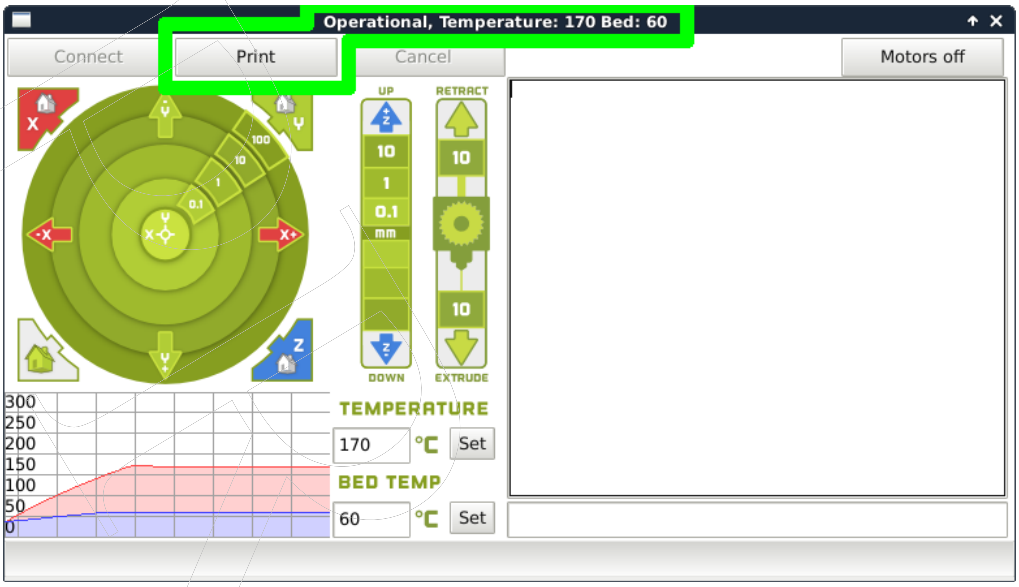
Start Print

Thanks to the TAZ automatic bed leveling system, it is now time to kick off the print! Through the LCD screen go to: `Print From SD` > `Desired File`. Your TAZ will then go through it's automatic cleaning and wiping process before starting the print. This should take approximately 5 minutes.

Printing from USB Cable

Connect your 3D printer to a computer using a USB cable, power it on and select the `Control` button at the top of the 3D viewing window. If the control button is not present, be sure you have a model file loaded into Cura. If a model is loaded but the Control button is still not present use a different USB cable, reinstall Cura, or contact our Support Team. This will bring up your Printer Interface. You will not be able to send any commands until the window title changes to `Operational`. Once operational, you can start your first print. Your TAZ will then go through it's automatic cleaning and wiping process before starting the print. This should take approximately 5 minutes.

PROOF



Operational, Temperature: 170 Bed: 60

TEMPERATURE 170 °C

BED TEMP 60 °C

Connect
Print
Cancel
Motors off
UP
RETRACT
10
1
0.1
mm
10
10
DOWN
EXTRUDE
100
10
1
0.1
X
Y
Z
300
250
200
150
100
50
0
Set
Set

Figure 2.12: Control Screen

Pausing Mid-Print

You will notice after you click the print button through Cura, it will change to a pause button. When activated, it will pause the print and automatically move your print head away from your object. This will allow color changes or material changes mid print.

Recommended Temperatures and Bed Preperation

When generating GCODE, it may be beneficial to adjust temperatures depending on your specific model file or filament. We have also found that for certain materials applying unscented glue stick to the print surface is required for successful bed adhesion and/or part release. Glue stick can also be added to help first layer adhesion on any material, and may be helpful for objects with a larger surface area. You can find our recommended temperature ranges and adhesion techniques below.

Filament Type	Bed Preparation	Nozzle Temp	Bed Temp	Removal Temp
ABS	Clean PEI	230-250	110	50
PLA	Clean PEI	195-215	60	45
HIPS	Clean PEI	230-250	110	50
Laywoo-D3	Clean PEI	175-195	60	45
bambooFill	Clean PEI	185-195	60	50
woodFill	Clean PEI	185-195	60	50
Laybrick	Clean PEI	175-195	60	45
bronzeFill	Clean PEI	225-235	60	50
copperFill	Clean PEI	225-235	60	50
Magnetic Iron PLA	Clean PEI	220-230	60	50
Stainless Steel PLA	Clean PEI	220-230	60	50
Conductive PLA	Clean PEI	215-230	60	50
Coffee PLA	Clean PEI	220-230	60	40
nGen	Clean PEI	220-240	85	60
t-glase	Glue stick	240-260	60	45
Flexible Filaments	Glue stick	215-230	50	35
Nylons	Glue stick	220-270	110	50
Polycarbonate	Glue stick	260-300	110	50
Polycarbonate + ABS	Glue stick	260-280	110	50
INOVA 1800	Glue stick	235-255	75	50
n-vent	Glue stick	225-245	60	50

Table 2.1: Recommended Temperatures

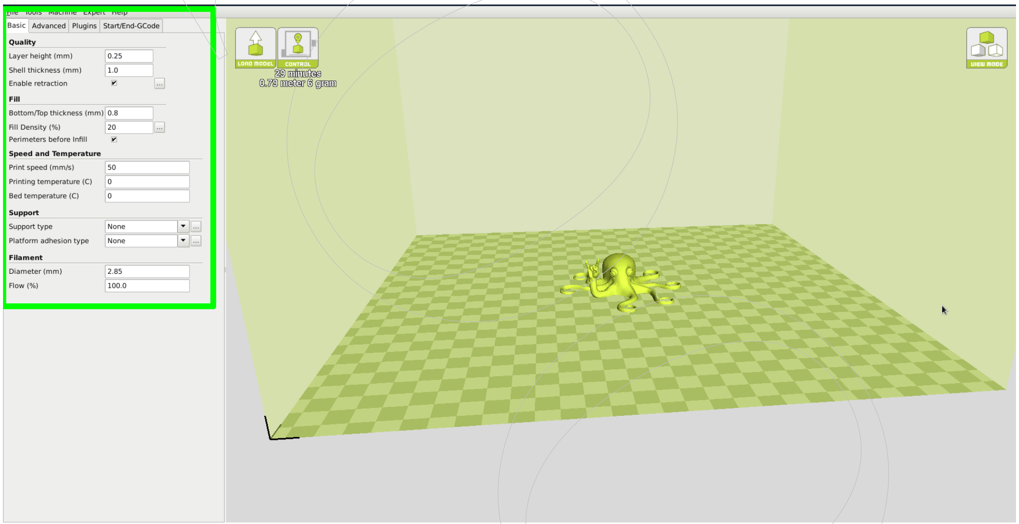
2.5 Removing Your First Print

After your first print has finished, wait for the print surface to cool to the recommended print removal temperature. Your parts will be easier to remove if you allow your heated bed to cool down to optimal temperature. This will allow the plastic to contract, making it easier to remove.

Once your heated bed has cooled, use the blue handled knife that was included with your toolkit to remove the item. Carefully insert the blade of the knife between your print and heated bed. Once underneath the part rotate the blade lifting with the sharp edge into the part, to gently pop the piece off your plate.

2.6 Full Settings

Full settings should not be used until enough experience with 3D printing has been gained to feel comfortable with all aspects of the printer and its operation. Quickprint Mode will provide good results for most models. The first time Cura is launched it will default to the `Quick Print` interface. In order to have more control of your slicing and GCODE generation, switch to `Full Settings`. Choose your desired Quick Print profile, as you can copy over the current settings when changing to `Full Settings` mode. This will automatically update the settings for that filament. Select `Expert` > `Switch to Full Settings`. The following tabs will now be available: Basic, Advanced, Plugins, and Start/End-GCODE. You will also have access to the Expert Settings menu entry.



Quality
Layer height (mm) 0.25
Shell thickness (mm) 1.0
Enable retraction ☑

Fill
Bottom/Top thickness (mm) 0.8
Fill Density (%) 20
Perimeters before Infill ☑

Speed and Temperature
Print speed (mm/s) 50
Printing temperature (C) 0
Bed temperature (C) 0

Support
Support type None
Platform adhesion type None

Filament
Diameter (mm) 2.85
Flow (%) 100.0

29 minutes
0.79 meter 6 gram

Figure 2.13: View in Full Settings

View in Full Settings

When you first switch to Full Settings, Cura will need to know what filament and quality you wish to use. It will automatically transfer our quickprint settings over to allow adjustments if one is selected. IF A QUICKPRINT PROFILE IS NOT AVAILABLE, YOU WILL NEED TO MANUALLY LOAD ONE IN. We recommend using our tested profiles that are available here: `https://www.lulzbot.com/cura`. You will want to choose the profile that matches your filament and quality needs. Once downloaded, you can load the file into Cura by selecting `File > Open Profile`. This will automatically update all of your Cura settings for use with your specific filament.

2.7 Basic Tab Options

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 object.

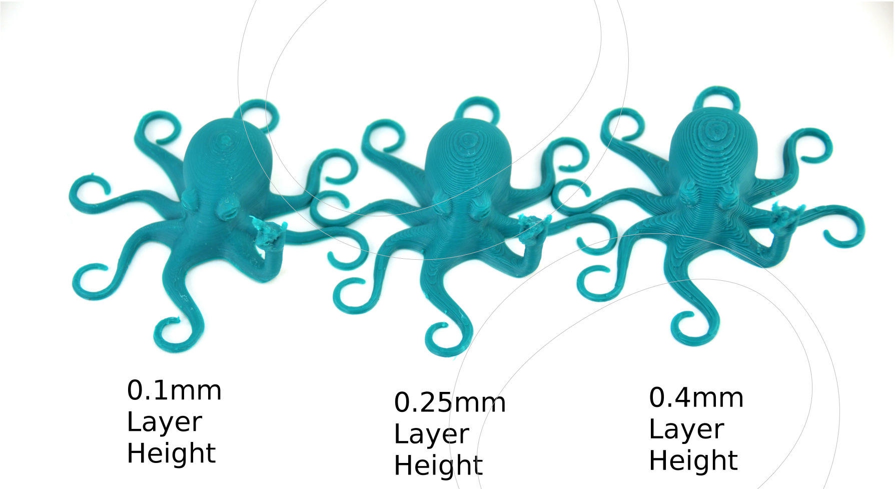


Figure 2.14: Differences in Layer Height

DRAFT

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 (mm)

Also known as "Surface Layers" 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 object. 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 infill and perimeters will be printed. We recommend leaving this on.

Print Speed (mm/s)

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: page 30. 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: page 30. 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.

Touching Buildplate

This causes the support material to build up between the heated bed and the object. The red example is Touching Buildplate.

Everywhere

This prints support material between the heated bed and object as well as between the object and itself. The green example is Support Everywhere.

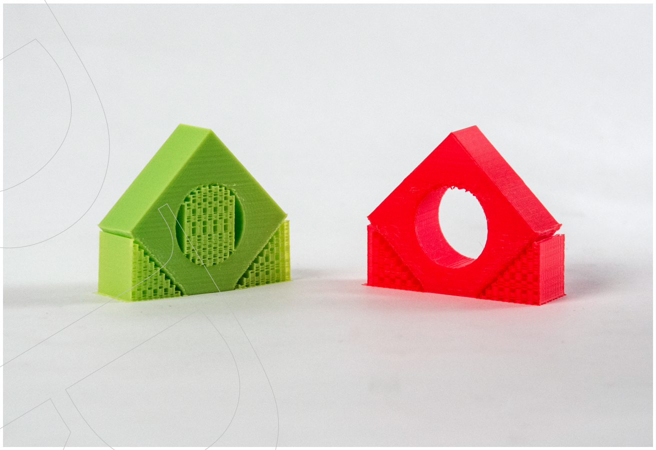


Figure 2.15: Support Types

`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`

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`

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.

PROOF

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.

2.8 Advanced Tab Options

Nozzle Size (mm)

This defines your nozzle size. The slicing engine uses this value combined with your other settings to determine how quickly to feed filament into your hot end. The TAZ ships with a 0.5mm nozzle standard.

Retraction Speed (mm/s)

Retraction Speed determines the speed at which your filament is reversed out of the hot end for travel moves and when changing direction during printing. We recommend keeping this set to less than 25mm/s to help protect the gears.

Retraction Distance

Retraction Distance determines how much filament is pulled out of your hot end on travel moves and when changing direction. You will want to adjust this depending on temperature settings and filament type. Higher thermal retaining filaments such as PLA behave better with a longer retraction distance. We have found anywhere from 1mm to 3mm is a good starting range.

Initial Layer Thickness

This will control how thick your first printed layer height is printed onto the heated bed. A larger initial layer height is more forgiving and will help first layer adhesion. All of our standard profiles have a 0.425mm initial layer height. This eliminates the need for adjustments when switching between filaments. Your LulzBot TAZ automatic bed leveling system could be affected if you change this from the standard profiles. Adjust at your own risk.

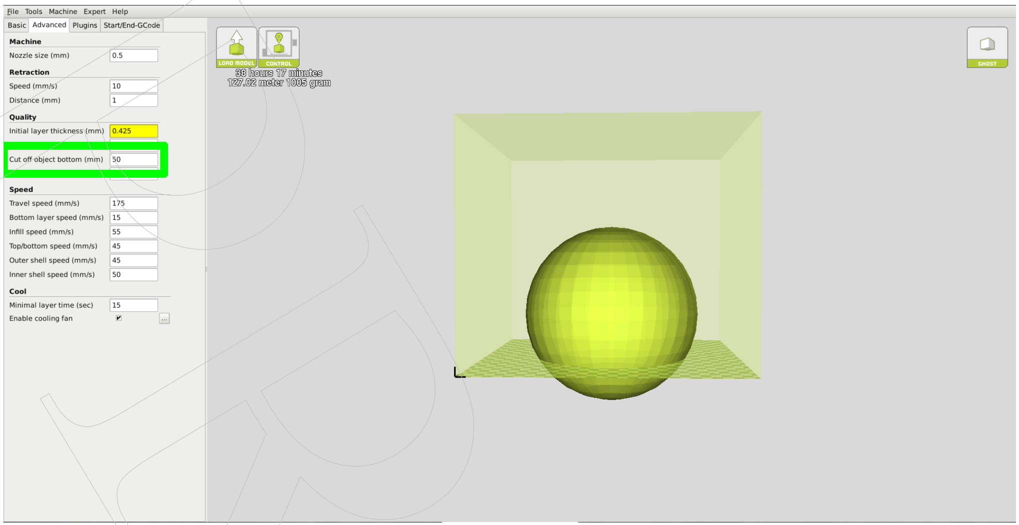
Initial Layer Line Width

This will control how wide your first extruded filament path is for the initial layer. A wider line width will help with bed adhesion. We have found 125% to be a good starting place. For models with moving, printed in place parts, a smaller initial layer line width is recommended. Your LulzBot TAZ automatic bed leveling system could be affected if you change this from the standard profiles. Adjust at your own risk.

Cut Off Object Bottom (mm)

This setting is used to help print models that were not specifically designed for FFF printing. In particular, it is for models that do not have a flat surface to adhere to the plate. It will sink your object Xmm into the build plate, creating a nice flat surface to begin your print. You can also use this option to remove the lower portion of your model, or if carefully measured, cut your part in half for printing object as two halves.

PROOF



38 hours 17 minutes
127.02 meter 1005 gram

Machine

Nozzle size (mm): 0.5

Retraction

Speed (mm/s): 10
Distance (mm): 1

Quality

Initial layer thickness (mm): 0.425
Cut off object bottom (mm): 50

Speed

Travel speed (mm/s): 175
Bottom layer speed (mm/s): 15
Infill speed (mm/s): 55
Top/bottom speed (mm/s): 45
Outer shell speed (mm/s): 45
Inner shell speed (mm/s): 50

Cool

Minimal layer time (sec): 15
Enable cooling fan ☑

Figure 2.16: Cutoff Example

Dual Extrusion Overlap

This will determine how far your Dual Extruders will overlap when laying down material. This will help adhesion between the two different colors or types of filament. This setting is only used when the printer is equipped with two hot ends and extruders.

Travel Speed

This setting will determine how fast your print head moves while not extruding filament. A normal travel speed of 125 - 175mm/s is recommended.

Bottom Layer Speed

This will control your initial layer speed. In general, a slower initial layer speed will help with first layer adhesion.

Infill Speed

This is how fast your print head speed will be while laying down the interior portion of your model. Faster speeds are usually tolerable here, as none of the infill will be visible from the outside of your object. If you go too fast compared to your inner and outer shells, you can have adhesion issues or blobs of filament left behind from the print head.

Outer Shell Speed

This will be the outermost surface of the model. This is the most important speed setting, as it controls the speed of your print head on the visible layers. As a general rule of thumb, the slower you go the better looking print you will get.

Inner Shell Speed

This affects vertical walls that are in between the outer shell and infill. This will not be visible but will help support the outer shell and the infill. We recommend keeping this speed setting between your infill and your outer shell speed.

Minimal Layer Time

This will determine a minimum amount of time your printer will spend laying down each layer. If your layer print time falls below this your printer will automatically slow down to reach this time before moving onto the next layer. Tweaking this can help get cleaner, crisper prints.

Enable Cooling Fan

Enables operation of your extruder active cooling fan. The fan settings can be adjusted in the `Expert Settings` options.

2.9 Plugins

Plugins are custom settings which will alter your print at specific points. The two that come preloaded with Cura are `Tweak at Z`, and `Pause at Height`. More plugins and information can be found here: `http://wiki.ultimaker.com/Category:CuraPlugin` To activate one of these highlight the desired plugin and click the drop-down arrow directly below the Plugins box. These plugins affect how GCODE is generated. Use at your own risk. Be sure to turn these off before switching back to quick print mode. They will affect quickprint settings if left on.

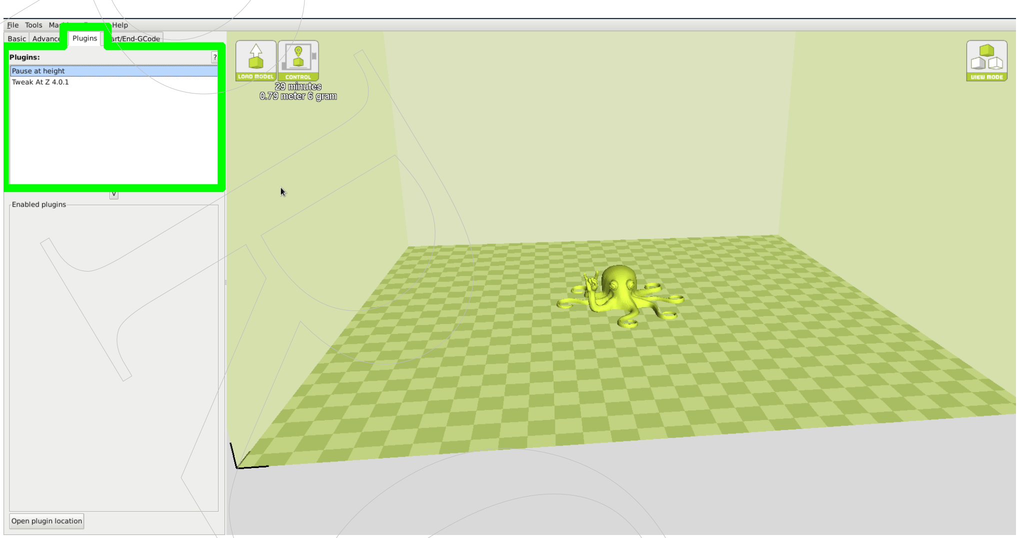

File
Tools
Ma
Help
Basic
Advance
Plugins
art/End-GCode
Plugins:
?
Pause at height
Tweak At Z 4.0.1
Enabled plugins
Open plugin location
LOAD MODEL
CONTROL
29 minutes
0.79 meter 6 gram
VIEW MODE

Figure 2.17: View of Plugins

Tweak at Z

Make basic changes at specified Z heights. You can define the Z height or layer count at which you want to make a change. Then choose how you would like to change your settings. You can alter temperatures, fan speeds, and print speeds. Fine tuning these for specific STL files, can produce cleaner prints.

Pause at Z Height

Pause your print at a specified height. You can also specify where to move the print head and how much filament to retract. This will prevent “blobs” from accumulating on your print while paused. This setting is most commonly used when switching colors of filaments in the middle of a print.

2.10 Start and End GCODE Settings

Custom GCODE allows for complex automatic printer movements and operations. By adding custom GCODE into the start or end of your file, you can alter how it prints. A comprehensive list of GCODE commands can be found here: `http://reprap.org/wiki/G-code` We recommend new users to leave this as provided in the profiles at `https://www.lulzbot.com/cura`

TAZ 6 Specific Considerations

Please be cautious when changing any of these start and end GCODE settings. This is where your automatic bed leveling commands are stored. If improperly altered, your printer will no longer automatically compensate for the heated bed position and can potentially damage components on the printer. If you are uncertain of the change you are trying to make to start/end GCODE, please contact us at `Support@LulzBot.com` before hand.

Changing Wiping and Probing Temperatures

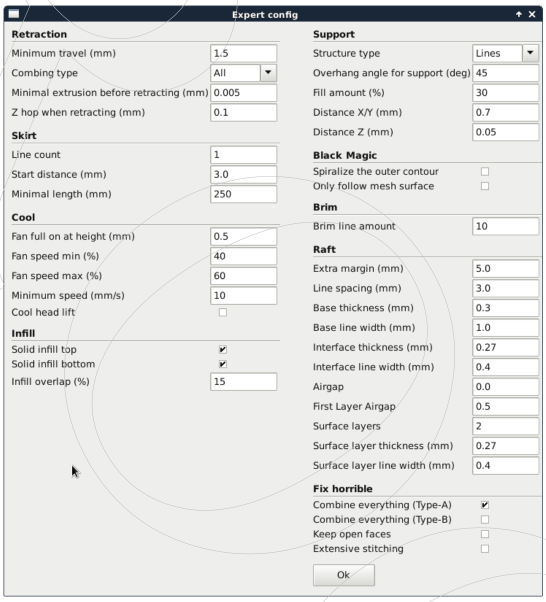
We have set wiping and probing temperatures that we have found work well with specific types of filament and filament manufacturers. If you are using a type and/or manufacturer of filament not listed in the quickprint settings, you may need to adjust these temperatures for optimal probing. In the start GCODE section, there will be three separate temperatures you can adjust. What these GCODE lines do will be described in the green text to the right of the command. The ones you will want to adjust are:

- `M109 SXXX` ; soften filament for z homing
- `M104 SYYY` ; wipe temp
- `M109 SZZZ` ; heat to probe temp

By changing the variables `(XXX, YYY, ZZZ)`, you can change what temperature your printer will soften, wipe, and probe. Our temperatures for softening, wiping, and probing will be a good starting point for other manufacturer's filament. When making adjustments, we recommend `+/- 5°C` changes at a time. `Be sure to watch the probing sequence when` `experimenting with new temperatures.` Be prepared to power off the 3D printer if the probing sequence fails.

2.11 Expert Settings

Expert settings will give you more specific options for your retraction, skirt, active cooling, infill, support, brim, raft, and special settings. To gain access to this section you go to `Expert` > `Open Full Settings` or on your keyboard press `Control + E`.



Retraction

Minimum travel (mm)	1.5
Combing type	All
Minimal extrusion before retracting (mm)	0.005
Z hop when retracting (mm)	0.1

Skirt

Line count	1
Start distance (mm)	3.0
Minimal length (mm)	250

Cool

Fan full on at height (mm)	0.5
Fan speed min (%)	40
Fan speed max (%)	60
Minimum speed (mm/s)	10
Cool head lift	☐

Infill

Solid infill top	☑
Solid infill bottom	☑
Infill overlap (%)	15

Support

Structure type	Lines
Overhang angle for support (deg)	45
Fill amount (%)	30
Distance X/Y (mm)	0.7
Distance Z (mm)	0.05

Black Magic

Spiralize the outer contour	☐
Only follow mesh surface	☐

Brim

Brim line amount	10

Raft

Extra margin (mm)	5.0
Line spacing (mm)	3.0
Base thickness (mm)	0.3
Base line width (mm)	1.0
Interface thickness (mm)	0.27
Interface line width (mm)	0.4
Airgap	0.0
First Layer Airgap	0.5
Surface layers	2
Surface layer thickness (mm)	0.27
Surface layer line width (mm)	0.4

Fix horrible

Combine everything (Type-A)	☑
Combine everything (Type-B)	☐
Keep open faces	☐
Extensive stitching	☐

Figure 2.18: View Expert Settings

2.12 Retraction

Retraction pulls filament out of your nozzle when it is not extruding to prevent your print head from dripping on your object. This section is where you will control how your extruder retracts its filament.

Minimum Travel

This sets the minimum travel distance of your print head in order to retract. If your print head is not moving this far during travel moves, it will not retract.

Combing

This option prevents your print head from traveling over holes in the X/Y plane when printing. This will slightly increase print time, but will prevent strings from getting caught on the holes during travel moves. We recommend keeping this setting on.

Minimal Extrusion Before Retracting

This will prevent a retraction move, if your extruder has not put out Xmm of filament since its last retraction.

Z Hop When Retracting

This will raise your print head Xmm while retracting. This setting helps prevent ooze and strings from being deposited on your print.

2.13 Skirt

Skirt creates a line around the outside of your object. Most commonly used to prime the extruder, in order to prevent missed filament at the beginning of a print. Leave this setting on. Turning this setting off will allow for utilizing the full build volume.

Line Count

This will define the number of loops the Skirt creates around the outside of your object. Smaller models will require more loops to properly prime the extruder.

Start Distance

This will define the distance away from your model that the skirt will be created.

Minimal Length

This will define the minimum extruded line length for the skirt. This will over ride your line count, producing as many lines as required to reach the minimal length.

2.14 Cool

This section will define how your extruder cooling fan will operate during the print. If your print speeds are slowed down due to minimal layer time, the fan will run between minimum and maximum speed based upon how much the layer is slowed down.

Fan on at Full Height

This is your Z height where your fan will be turned on to its minimum percentage setting. Especially helpful with high temperature retaining filaments such as PLA. This will be scaled between 0%, and your minimum fan speed based upon layer height; with it being disabled for the first layer.

Fan Speed Min

This will be the speed your fan runs when enabled at full height. Once the Z height is reached for Fan on at Full Height, this will be the speed your fan runs at.

Fan Speed Max

This is the fastest speed at which your fan will ever run. When your print speed is slowed down due to minimal layer time, your fan will run between minimum and maximum speed. The maximum fan speed is reached when your printer must be slowed by 50% or greater.

PROOF

2.15 Support

You define how your support material is generated here. You must have some form of support turned on in the basic settings in order for these settings to have an effect.

Structure Type

You can choose between a Grid or a Line pattern for your support material. The grid will be a checkerboard pattern in the X and Y direction. The line option will produce lines in along the Y-axis for support. The grid will provide stronger support than the line option, but will be harder to remove.

Overhang Angle for Support

This will determine where support material is generated. In general you will be able to print a model with 45 to 90 degree angles in relation to the bed without support. We recommend leaving this setting at 45 degrees.

Fill Amount

This will determine how dense your support material is printed, similar to Infill Percentage. The higher percentage, the better support, but it will be harder to remove the support material, will use more material, and will lead to a larger total printing time.

Distance X/Y

This will determine how far away from your object in the X/Y plane that the support material is being placed.

Distance Z

This will determine how far away your support material is from your object in the vertical direction. A smaller number here makes for better support, but makes it harder to remove.

2.16 **Black Magic**

This section allows you to transform your model into a hollow shell, a single layer thick.

Spiralize the Outer Contour

This causes your Z-axis to be constantly moving upward as printing your single outer wall shell. The results are no layer change lines, giving a much smoother surface. This setting is typically only used for artistic objects as they will be fragile.

Only Follow Mesh Surface

This will cause your print to follow the outside of your model, building it completely hollow with a single wall outer shell. The only difference between this and Spiralize, is that the Z-axis moves regularly. That is, it prints a layer and then moves up vertically to the next one.

2.17 **Brim**

Brim circles the base of the print while making contact, helping adhere the print to the heated plate. This is only one layer thick, and easily removed post-print. This section defines how the brim is formed when brim is activated in basic settings.

Brim Line Amount

This will determine the distance the brim will cover around the outside of your object. The more brim used, the better your part will adhere to the plate.

2.18 **Raft**

Raft is a platform built underneath your object, designed to help adhesion and prevent warping. It will lay down support material, and then a platform on top of the supports. Your model will be built on top of this platform. The bottom surface of your printed part will not be as clean or as even when using this option. Raft is typically not required.

Extra Margin

This determines the distance around the outside of your object that the raft is created. Can be helpful for ensuring no warping of the lower layers.

Line Spacing

This will determine the spacing between “support” lines for the raft. A small spacing makes the support structures closer together improving strength of the raft, but uses more material.

Base Thickness

This defines how thick your raft will be.

Base Line Width

This will define how wide your “support” material is for the raft. This setting will determine how well the surface layers of the raft print.

Interface Thickness

This will determine how thick the surface layers of the raft are. The surface layers are the platform that is built upon the supports.

Interface Line Width

This will determine how wide the top layers of the platform will be. In general, you can keep this set to your nozzle size, as surface quality of the removable raft is not important.

Airgap

This will define the distance between your raft and your print. A larger gap will make your part easier to remove, but will make the bottom of your

Proof

print look worse.

Surface Layers

This will determine the number of layers that create the “platform” of your raft. If you have a wide line spacing, you may want to increase this number to ensure a solid platform.

2.19 Fix Horrible

These are some of the more advanced and experimental options. They are designed to help repair models with errors to make them suitable for 3D printing. `They do not always work.` Please be cautious when using these options as they can have unintended effects on your print quality.

Combine Everything (Type-A)

This will attempt to fix all external mesh errors, while keeping internal holes intact. This can accidentally fill in intentional internal holes.

Combine Everything (Type-B)

This will ignore all internal holes of the model and only focus on the external holes. This is helpful when only the outside finish of the model is important.

Keep Open Faces

This will ignore all manifold errors in the object. It can create issues generating the GCODE as Cura does not know how to interpret the open holes. This option should only be used if you are sure that the holes in the mesh are intended. In general, you should not use this option.

Extensive Stitching

This causes Cura to automatically add triangle meshes in an attempt to fix manifold errors. This algorithm will greatly increase GCODE generation time and may end up adding in un-intended meshes. It is recommended that you repair your model through MeshLab, FreeCAD or your preferred CAD program before attempting this option.

2.20 Dual Extrusion

The LulzBot TAZ has the ability to add dual extrusion functionality with the Dual Extruder Tool Head. We only recommend the Dual Extruder Tool Head for advanced users. Installation and operation of the dual extruder will require you to flash your firmware, calibrate your extruders, level two heads to a single plane, define extruder offsets, and fine tune your slicing profile. This can be a little overwhelming for new users. We recommended that you have a couple of months of single extruder printing experience before making the switch.

Tool Head Installation Instructions

Refer to appropriate Dual Extruder Tool Head Quick Start Guide for the specific steps required to install and operate your new tool head. Failure to follow the proper steps may result in damage to the tool head or 3D printer. Installation and operating instructions are available at `OHAI.LulzBot.com/accessories`. The following steps are only meant to be an overview of the typical process.

Updating Firmware

In order to use the Dual Extruder you will need to update your firmware to activate the second hot end.

- Power on your 3D printer and connect it to your computer through USB.
- Select `Machine > Machine Settings`.
- In the top right-hand corner you will see a `Change Tool Head` button, select this.
- Next you will need to select the proper hot end type. Please compare photos with your existing tool head. Choose the correct hot end and select `Next`.

- Select the photo that corresponds with your tool head and press `Next`.
- The final step will be `flashing the firmware`. Choose `Flash the firmware` to be sure to update the printers firmware to the proper settings. This must be done to allow your printer to operate properly.
- Once the progress bar has completed select `OK > Next > Finish`.
- You can verify a proper firmware update by looking at the LCD screen. You should now see three separate temperature readings.

You can revert the firmware back to the stock configuration for your LulzBot 3D printer by selecting `Machine > Machine Settings > Change Tool Head`. Doing so will overwrite any of your current firmware settings.

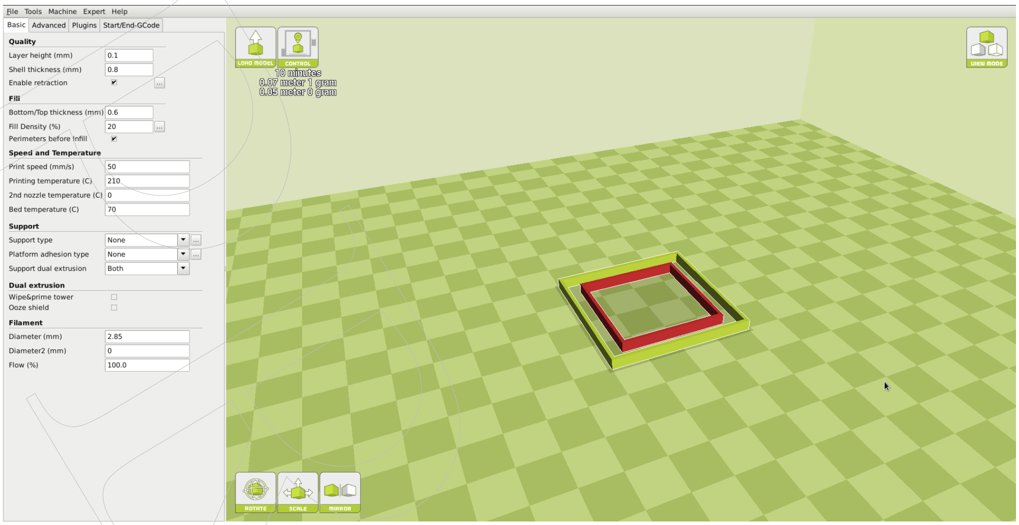
Calibrating Extruders

Each individual extruder will have its own unique set of Extruder steps per unit, or Esteps. Your Dual Extruder was calibrated before leaving the production floor. The number for each extruder will be located on the back of the tool head. This will need to be updated within your printer's firmware.

- Update your Esteps for the rear extruder and E1steps for the front extruder through your LCD screen. `Configuration > Advanced Settings > E/E1 Steps > New Number`. Push in on the knob to exit out of the Esteps entry.
- Store your new Esteps `Configuration > Store Memory`.

Defining Second Extruder Offset

To get crisper looking prints, you will want to really dial in your offsets. With the outer and inner square on your build platform, `left-click` on the outer square, then `right-click` on the inner square and select `Dual Extrusion Merge`. The red square will be printed with your front extruder, and the yellow square will be printed with the rear extruder.



Quality

Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	☑

Fill

Bottom/Top thickness (mm)	0.6
Fill Density (%)	20
Perimeters before Infill	☑

Speed and Temperature

Print speed (mm/s)	50
Printing temperature (C)	210
2nd nozzle temperature (C)	0
Bed temperature (C)	70

Support

Support type	None
Platform adhesion type	None
Support dual extrusion	Both

Dual extrusion

Wipe&prime tower	☐
Ooze shield	☐

Filament

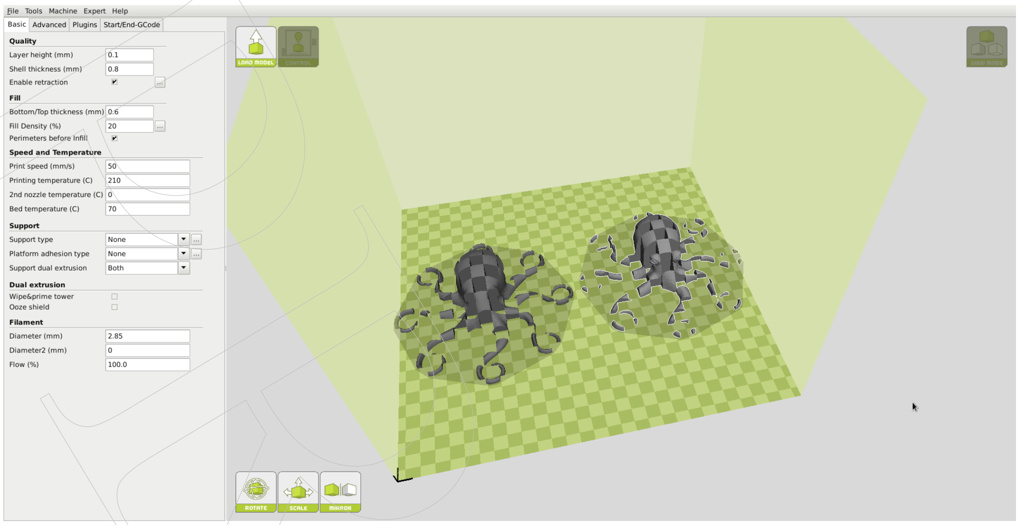
Diameter (mm)	2.85
Diameter2 (mm)	0
Flow (%)	100.0

10 minutes
0.07 meter 1 gram
0.05 meter 0 gram

After printing the squares, you will want to measure Top, Bottom, Left, and Right gap. Enter these numbers into our offset calculator found here: `https://www.lulzbot.com/dual-extruder-calibration-calculator` This will produce new offsets, that will need to be updated in the Machine Settings menu. Repeat as many times as desired to truly fine tune the offset.

First Dual Print

When you are ready to produce your first dual extrusion print, you will need to combine the two separate STL files. One STL file will be for each print head. `Left Click` on whichever STL you would like printed with the `Rear Tool head`. Then `Right Click` on whichever STL file you would like printed with the `Front Tool head` and select dual extrusion merge. You will now see a single model in two colors on the build plate. The red section will be printed with the front extruder, while the green section will be printed with the rear extruder.


File
Tools
Machine
Expert
Help
Basic
Advanced
Plugins
Start/End-GCode
LOAD MODEL
CONTROL
VIEW MODE
ROTATE
SCALE
MIRROR

Quality
Layer height (mm): 0.1
Shell thickness (mm): 0.8
Enable retraction: ☑

Fill
Bottom/Top thickness (mm): 0.6
Fill Density (%): 20
Perimeters before Infill: ☑

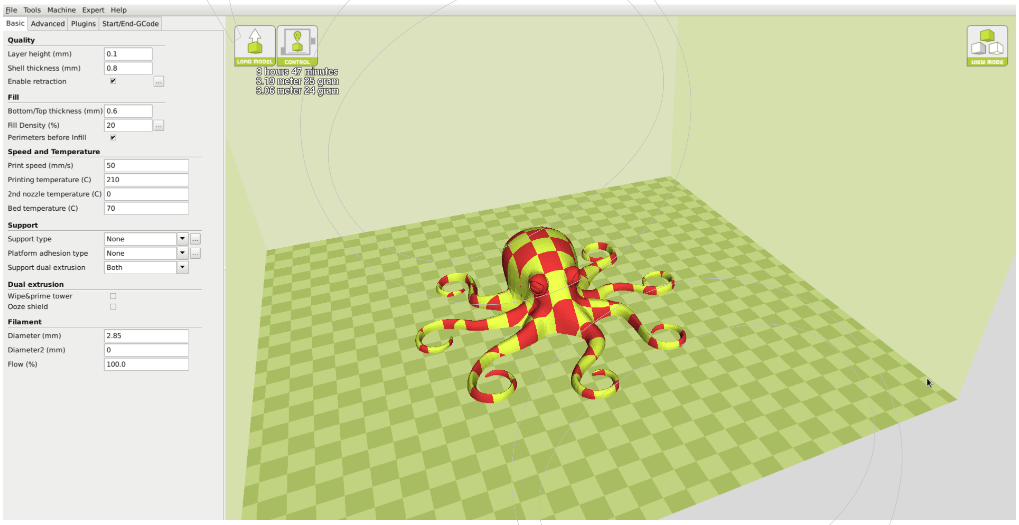
Speed and Temperature
Print speed (mm/s): 50
Printing temperature (C): 210
2nd nozzle temperature (C): 0
Bed temperature (C): 70

Support
Support type: None
Platform adhesion type: None
Support dual extrusion: Both

Dual extrusion
Wipe&prime tower: ☐
Ooze shield: ☐

Filament
Diameter (mm): 2.85
Diameter2 (mm): 0
Flow (%): 100.0

Figure 2.19: Before Merge


File
Tools
Machine
Expert
Help
Basic
Advanced
Plugins
Start/End-GCode
LOAD MODEL
CONTROL
VIEW MODE

9 hours 47 minutes
3.19 meter 25 gram
3.06 meter 24 gram

Quality
Layer height (mm): 0.1
Shell thickness (mm): 0.8
Enable retraction: ☑

Fill
Bottom/Top thickness (mm): 0.6
Fill Density (%): 20
Perimeters before Infill: ☑

Speed and Temperature
Print speed (mm/s): 50
Printing temperature (C): 210
2nd nozzle temperature (C): 0
Bed temperature (C): 70

Support
Support type: None
Platform adhesion type: None
Support dual extrusion: Both

Dual extrusion
Wipe&prime tower: ☐
Ooze shield: ☐

Filament
Diameter (mm): 2.85
Diameter2 (mm): 0
Flow (%): 100.0

Figure 2.20: After Merge

First Print

After your object has been merged as intended, you will be ready to start the print. After selecting print from your SD card or the control panel, your printer will start the probing and cleaning process. It will clean both nozzles, but only probe with the rear. If you are getting contact with your printer nozzles on previous printed parts, you will need to level the front hot end to the rear.

PROOF

PROOF

Printing with the Graphic LCD

PROOF

The Graphical LCD Controller, or GLCD, allows you to print with the LulzBot TAZ 3D printer without needing to have a computer connected or using host software such as Cura LulzBot Edition. This will allow for more efficient space in the workspace and free up a computer for other tasks.

In the following sections you will find general information on using the GLCD, how to transfer .gcode files to the included SD card, heat up the printer, start a print, and make configuration adjustments.

3.1 Graphical LCD Controller, Cura or Other Host?

The Graphical LCD Controller is perfect for normal day to day printing and will be used in the majority of your print jobs. However, in some instances, you will want to use Cura instead of the GLCD. A few examples of when you would want to plug the USB cable back in and use Cura:

- A number of manual movements are required to perform calibration checks. Because of this it is easier and faster to make the required manual movements within Cura. Calibration checks can be done with the GLCD, but require a number of repetitive menu selections.
- 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.

3.2 Multiple Connections

Because the LulzBot TAZ 3D printer can be controlled by the GLCD and by host software, caution is advised when connecting to the printer through USB as the print can be interrupted when connecting or disconnecting the USB cable. A general rule is: once you have started a print with either the GLCD, Cura, or another printer host, for the rest of the print use only that controller. When printing with the GLCD, never try to connect through USB in the Cura host software; wait until the print is complete, and then connect in Cura.

3.3 Putting Print Files on the SD Card

To print from the GLCD, transfer .gcode print files onto the SD card. Follow the normal steps, as explained in the Cura chapter, to create .gcode print files on your computer. Insert the SD card into your computer using a SD reader slot or USB SD card reader. Open a file browser- / -manager and locate the created .gcode files; drag and drop or paste the .gcode files to the SD card. Once the files have transferred, eject the SD card from your computer and insert it back into the SD card slot on the left side of the printer, at the top near the GLCD.

3.4 Printing With the Graphical LCD Controller

The Graphical LCD Controller Status Screen

The GLCD screen will turn on you power up your LulzBot TAZ 3D printer. The start-up screen will display the Status screen (fig. 3.1, page 57). The Status screen is the default screen for the GLCD, presenting the current status of the printer. There are a number of live statuses shown on

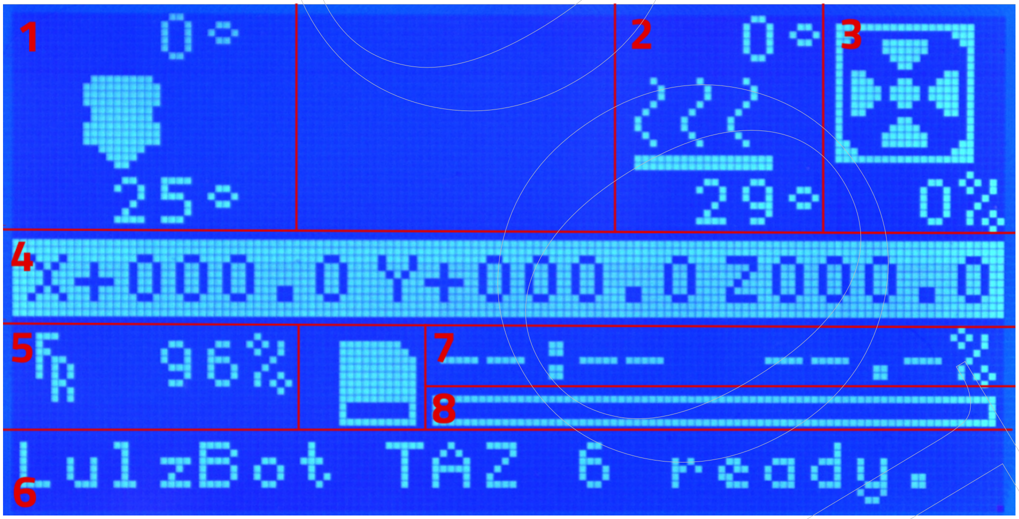


Figure 3.1: GLCD Info Screen

the Status screen; these will give you current temperatures, tool head coordinates, print status, and more. The different numbered sections of the status screen are shown in Figure 3.1 above. Follow the key below for more information on each section.

1. Hot end temperatures: represents the current temperature (bottom) and set temperature (top) of up to three nozzles. The RAMBo control electronics currently only supports two hot ends.
2. Heat bed temperature: represents the current temperature (bottom) and set temperature (top) of the heat bed.
3. Fan speed: represents the current optional extruder fan speed. The fan is set to off (0%) by default; the fan is not recommended for ABS. If you print with PLA filament, you will want to use the fan. If cooling is turned on in Cura, this portion of the display will change to reflect the .g-code embedded extruder fan control instructions.
4. Tool head coordinates: represents the current tool head coordinates on the X, Y, and Z axes.
5. Feed rate: represents the current feed rate setting. The feed rate is set to 100% by default; this matches the speed set in the gcode generation. When on, the status screen selection knob can be turned to increase or decrease the feed rate during the print. Increasing/decreasing the feed rate will increase/decrease the speed of the print.
6. Printer status: lists the current status of the printer including: SD card status, current printing file, or completed print time.
7. Current print time: lists the length of time for the current print job.
8. Progress bar: represents the progress of the current print job. The print is finished when the bar is completely white.

Using the Selection Knob

To navigate through the GLCD menu use the selection knob by rotating to scroll through selections and pressing the knob to make a selection. From the main status screen, press the knob to move into the menu screen

(Figure 3.2, page 59). To move backwards in the menu tree, select the top most menu selection on the current screen. Selections that will move you backwards through the menu tree are noted by an upwards-facing arrow. Note that if the menu is left idle it will automatically move back to the main status screen.

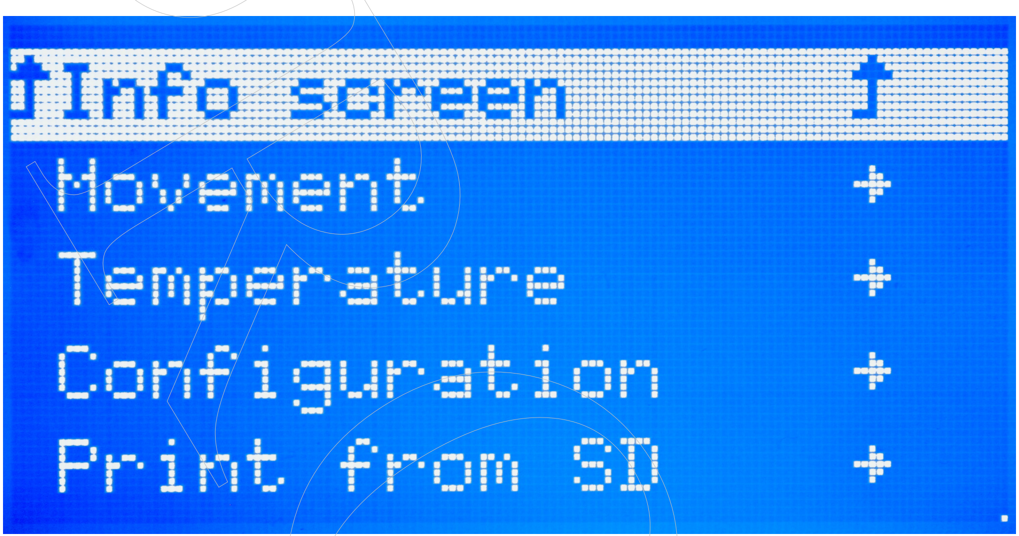


Figure 3.2: Main Menu

Selecting a File From the SD and Starting a Print

Once the SD card has the desired GCODE file loaded onto it, you can begin your first print. From the main menu select the `Print from SD` option. You will now see a selection of the directories and .gcode files on the SD card. Navigate through the menu to locate the file you would like to print. Select the desired file to begin the print.

Making Manual Movements With the Graphical LCD Controller

As noted previously (page 56), making numerous manual movements is easier done using Cura. However, you can make manual movements with the GLCD. Navigate to `Movement -> Move Axis`. You will select the length of the movement and then which axis to move.

When at the move screen; turn the selection knob clockwise to move the axis in millimeters in the positive direction and counter-clockwise for the negative direction.

Tune

Once you begin a print, you will notice your movement option changes to tune. This section will allow you to make changes to individual settings while a print is running. You can adjust overall speed, nozzle temperature, bed temperature, flow %, check apprx layer height, and even change filament. `When adjusting these settings, we recommend making small changes at a time.`

Change Filament

You can change filament through the GLCD mid print by going to `Tune -> Change Filament`. This will park your print head in the front left corner, and wait for you to resume the print. During this time you can change colors, or even types of filament. Once your filament change has completed, press in your GLCD knob to continue where it left off. `If you manually move the tool head while changing filament, your printer will lose place of it's position.`

3.5 Configuration Options

Out of the box, the LulzBot TAZ 3D printer is already calibrated for printing. However, the GLCD does allow tuning of the more advanced configuration settings. We highly suggest you not modify the configuration settings unless you are certain it is necessary. The configuration section contains settings that control how your printer operates.

Changing EEPROM Settings

The `Store Memory` and `Load Memory` functions will store and load the changes you make using the GLCD. You must use the `Store Memory` function to save the adjusted settings when the printer is powered on. If you ever need to revert to the original factory settings navigate to `Control -> Restore Failsafe`. Clicking `Restore to Factory` will set all configuration settings back to the original factory settings in the firmware.

For more information on the configuration settings please see the LulzBot TAZ support page at `LulzBot.com/support`.

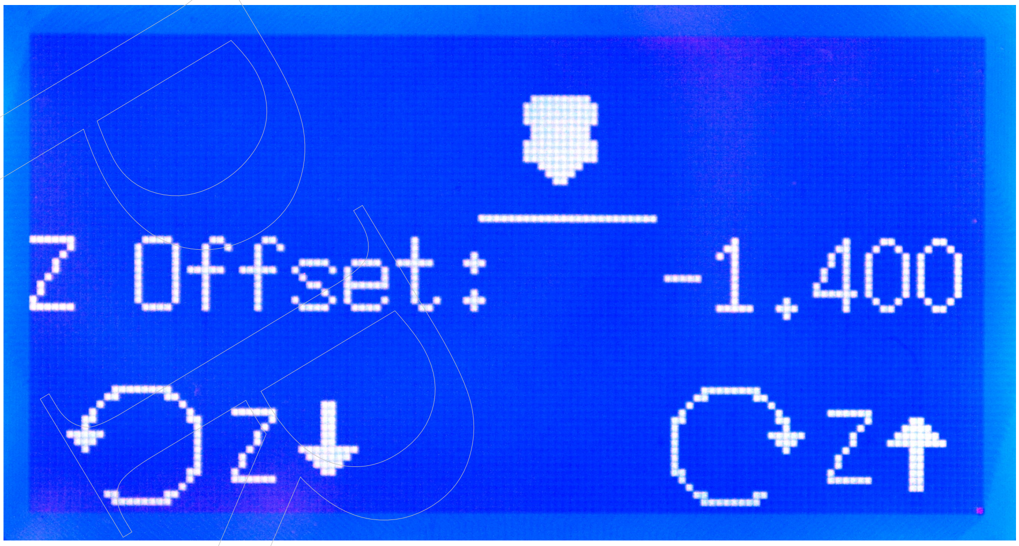
3.6 Advanced Settings

In the advanced settings, you can control/modify your Z offset, maximum velocity, acceleration, jerk, and esteps/mm. Once again we highly suggest you not modify the configuration settings unless you are certain it is necessary. Making incorrect adjustments of these settings can have adverse affects on your prints, and potentially damage your machine.

Z Offset

Your LulzBot TAZ 3D printer has the ability to change the first layer height (Z offset) directly through the GLCD, even while printing the first layer. Using the GLCD, navigate to `Configuration -> Advanced Settings -> Z Offset`. While in this screen you can rotate the GLCD knob counter-clockwise to bring your nozzle closer to the print surface, and rotate it clockwise to bring it farther away from the bed. `Push in the GLCD knob to save your settings.`

PROOF



Z Offset: -1.400

Figure 3.3: Z Offset Screen

`Acceleration`

Acceleration is the derivative of speed, and determines how quickly your print head and bed will reach defined speeds. We have this set to 500mm/s^2 to help prevent resonance and shadowing. This can be increased for sharper corners and faster turns, however it can lead backlash issues.

`Jerk Settings`

Jerk is the derivative of accleration and will determine the maximum speed change allowed at any given time. `VXY- Jerk` will control jerk setting for your X and Y-axis. `VZ - Jerk` will control the jerk setting for your Z-axis. `VE - Jerk` will control the jerk settings for your extruder.

`Vmax`

The Vmax settings will determine the maximum speed that your printer can move on any specific motor(s). We recommend leaving these settings as is. Increasing these speeds too much can lead to skipped steps and/or binding. There will be an individual section for `X, Y, Z, and Extruder` motors.

`Vmin`

This sets the minimum overall speeds. We have this set to 0 by default to allow greater control through your preferred slicing program.

`VTrav Min`

This will set the minimum travel speed when the tool head is not extruding. We have this set to 0 by default to allow greater control through your preferred slicing program.

`Amax`

The `Amax` settings will allow you to define a maximum acceleration limit for each motor. Increasing these may reduce print time, but can result in in shadowing and resonance issues.

`A-Retract`

This sets the maximum acceleration for retraction moves of the extruder. Going too high can lead to stripping, and going too low can lead to dripping. We have this set to 3000mm/s^2 by default.

`Steps/mm`

Your LulzBot TAZ 3D printer is factory calibrated on all axis for accurate movement. These settings control that movement, and if adjusted your objects will not come out at the proper dimensions. `We only recommend adjusting your Esteps/mm` as this can be fine tuned for individual tool heads and filaments.

PROOF

Graphical LCD Controller Menu Map

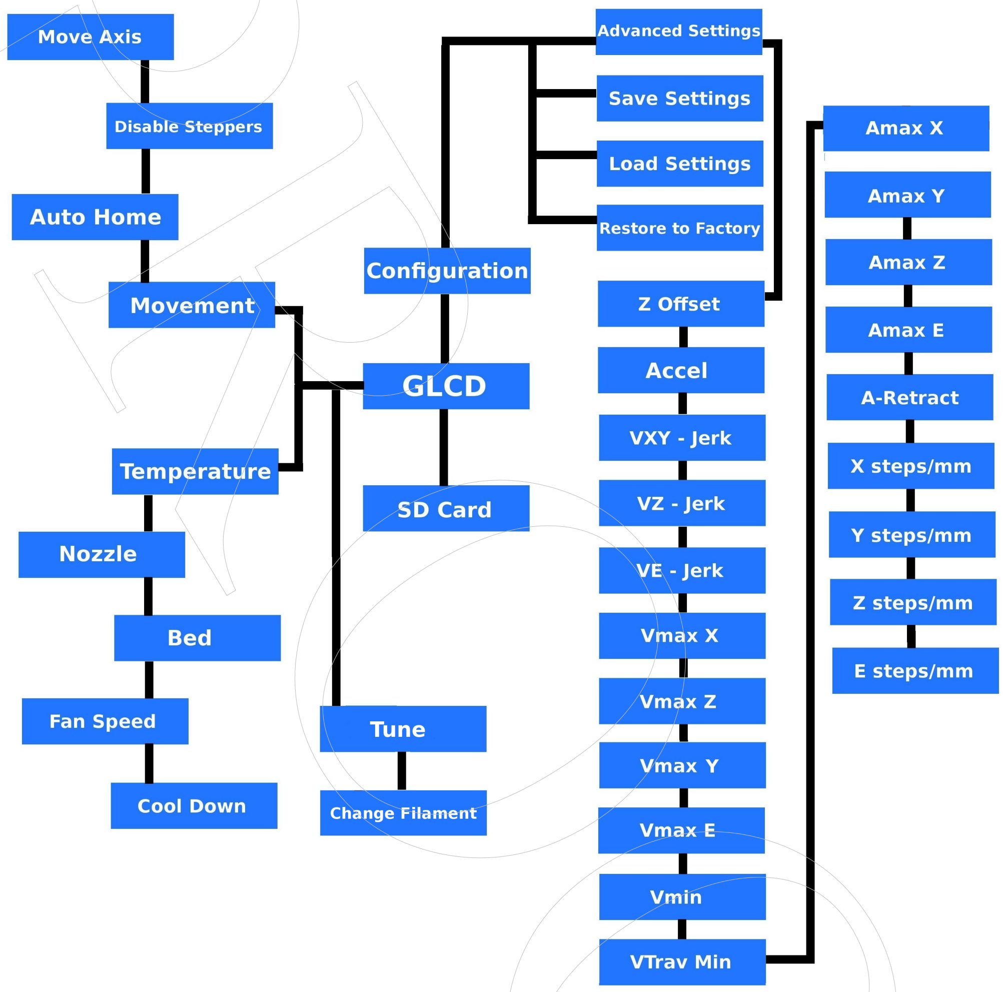

Move Axis
Disable Steppers
Auto Home
Movement
Temperature
Nozzle
Bed
Fan Speed
Cool Down
Configuration
GLCD
SD Card
Tune
Change Filament
Advanced Settings
Save Settings
Load Settings
Restore to Factory
Z Offset
Accel
VXY - Jerk
VZ - Jerk
VE - Jerk
Vmax X
Vmax Z
Vmax Y
Vmax E
Vmin
VTrav Min
Amax X
Amax Y
Amax Z
Amax E
A-Retract
X steps/mm
Y steps/mm
Z steps/mm
E steps/mm

Figure 3.4: GLCD Map

PROOF

Maintaining Your 3D Printer

PROOF

4.1 Overview

Little maintenance is required keep your LulzBot® TAZ 3D printer running. Depending on your rate of use you will want to perform a quick check of your printer every 2 to 4 weeks. The following maintenance guidelines will keep your printer printing quality parts.

4.2 Smooth Rods

Wipe the smooth steel rods with a clean cloth or paper towel. The linear bushings leave a solid lubricant that builds up over time. Hearing squeaking noises while the printer is printing is likely a sign that the smooth rods need to be cleaned. NOTE: Never apply any lubricant or cleaning agent to the smooth rods; the bushings are self-lubricating.

4.3 Lead Screw Drive Rods

Periodically, you may want to wipe down the threaded rods with a lithium-based grease. Never use any petroleum based grease, which may compromise the plastic parts. We recommend Lucas white lithium grease NLGI #2. Apply the lithium grease both above and below the X ends on the threaded rods and wipe down the threaded rods. Use your preferred printer host software or the Graphical LCD controller to drive the Z-axis up and down to help further distribute the lubricant. Wipe off any excess with a clean cloth.

4.4 PEI Print Surface

After repeated use, the PEI print surface will begin to need cleaning. To clean the PEI print surface, wipe clean with watered-down Isopropyl Alcohol `10:1 IPA to water ratio` and a clean cloth. If you encounter prints lifting from the PEI surface, use fine grit sandpaper, typically 2000-2500 grit to clean the PEI print surface. We do not recommend printing on bare glass, as it can lead to glass bed damage or failure. Never use acetone to clean the PEI print surface as doing so can damage the film. You can find our maintenance and trouble shooting guides here:

`https://www.lulzbot.com/learn/tutorials`

4.5 Hobbed Bolt

Filament is pulled through the extruder by a hobbed (or toothed) bolt. After repeated use, the teeth of the hobbed bolt can become filled with plastic. Using the brush or pick from the printer kit, clean out the hobbed bolt teeth. If an extruder jam ever occurs, remove the plastic filament from the extruder and clean out the hobbed bolt.

4.6 Belts

Over long periods or after extensive relocating of the printer you may need to re-tighten the belts on your 3D printer. For the X-axis, using the 2.5mm hex driver, loosen one of the belt clamps. The belts clamps are located on the X-axis carriage. To loosen the belt clamp, loosen the M3 screws on the clamp. Using the needle nose pliers, pull the belt tight. While holding the belt tight, tighten down the M3 screw. The Y-axis belt can be tightened using the same steps as the X-axis using the belt clamps found on the bottom of the Y-axis plate. Make sure not to over tighten the belts as this can cause unneeded stress on the printer.

4.7 Hot End

The hot end should be kept clean of extruded plastic by removing melted plastic strands with tweezers. If melted plastic builds up on the hot end nozzle you can clean it by raising the head off the build plate, and heating to extrusion temperature. Using a thick leather glove and a blue shop towel, carefully wipe off the outside of the hot end. Never use a metal wire brush on your hot end while powered as it can potentially short the control board.

4.8 Nozzle Wiping Pad

Over time the nozzle wiping pad will become filled with plastic residue. The pad can be flipped over once and will need to be replaced when full. Replacement nozzle wiping pads are available in our online store at `http://LulzBot.com`. Do not attempt to use a plastic or polymer based wiping pad as it can melt, rather than clean the nozzle. A metal wiping pad should never be used, as it can cause electrical shorts. If the nozzle is not clean during the bed-level calibration process the print surface or tool head can be damaged.

4.9 Bed Leveling Washers

Keep the washers mounted on each corner of the print surface clean and dust free by wiping them periodically with Isopropyl Alcohol (IPA), or a clean dry cloth. If the bed leveling washers are not clean during the bed calibration process the print surface or tool head may be damaged. Never attempt to clean the bed leveling washers during the probing sequence as it may lead to personal injury.

4.10 Cooling Fans

Every 2-4 weeks carefully clean your hot end cooling fans by first unplugging the tool head from the extruder harness. Gently blow any dust away with short bursts with a can of compressed air.

4.11 Electronics

The electronics case holding the RAMBo board may need dust blown out occasionally. Power down the printer and use short bursts of compressed air to blow out any dust.

PROOF

Advanced Usage

PROOF

5.1 Intro

After you become familiar with printing using the default settings, a few advanced techniques may help in getting better and more consistent prints from the LulzBot® TAZ 3D printer. Some of these instructions are items and materials not included with the TAZ. With any of these additional items or materials, follow safety and usage guidelines as instructed by the manufacturer.

5.2 Changing nozzles

Nozzle related issues will not be covered under warranty after nozzle changes.

Your hot end is equipped with a 0.50mm nozzle. This nozzle diameter will print faster than a 0.35mm nozzle. Purchase a tool head from `LulzBot.com` with an alternate nozzle diameter for quick, easy, and simple changes.

Due to the specific torque of 30 inch pounds required to tighten the nozzle when removed, removing the nozzle is not recommended. Failure to properly tighten the nozzle to the specific recommended torque may lead to leaks or damage if over-tightened.

5.3 Bed Adhesion

You may find that during printing, printed parts lift off the print surface on the corners. If you are seeing this turning on `Brim` in Cura will help increase the surface area of the print, improving part adhesion. If the corners of the part still lift, clean the PEI surface with IPA/ Isopropyl Alcohol and sand the surface with fine grit (2000-3000) sandpaper.

ABS/Acetone Glue

Acetone is not included or required with the LulzBot TAZ 3D printer. Acetone and ABS solution is NOT recommended anymore, as the PEI print surface works well without it.

Glue stick/PVA Glue solution

While almost all 3D printing filaments will adhere well to the PEI print surface, Nylon, and Amphora™-based print adhesion can be improved with the application of an unscented PVA glue solution/glue stick. Flexible filaments such as NinjaFlex® and SemiFlex™ can adhere too well and can be released easier from the PEI print surface with an application of an unscented PVA glue solution/glue stick to the print surface prior to starting the printing process.

5.4 Using 1.75mm filament

Your LulzBot TAZ 3D printer is set up to use 3mm plastic filament by default and may be capable of printing 1.75mm filament with no hardware modification. While many of our advanced users are able to do so, your results may vary. More information can be found in our User Forums at: `https://forum.lulzbot.com/viewtopic.php?f=16&t=1923`

PROOF

PROOF

Hardware and Software Source Code

PROOF

The LulzBot® TAZ 3D printer is a Free Software and Open Source Hardware design. All of the source files are available at `http://download.lulzbot.com/TAZ` including:

- The latest version of this document, with LaTeX source code.
- 3D models and print files for all of the printed parts in `.stl`, `.gcode`, and other original source files.
- 3D calibration objects and random novelties.
- Production file for calculating large print runs.
- Design files for all electronics and machined parts.
  - LulzBot® Hexagon Hot End
  - RAMBo electronics
  - Various spec sheets
- Bill of materials including every part needed to build the printer.
  - TAZ
  - LulzBot Hexagon Hot End
- Drawings of components.
  - Aluminum extrusions
  - LulzBot Hexagon Hot End
  - Bed plate
- Software binaries and source code for GNU/Linux and others. Also includes known good configuration files.
  - Cura
  - Marlin

PROOF

3D Printer Support

PROOF

7.1 `LulzBot`

For common technical support questions for your LulzBot® TAZ 3D printer please visit `LulzBot.com/support`. Also, visit `Forum.LulzBot.com` for support and tips from the LulzBot 3D printer community. If you have further questions, e-mail our support team at `Support@LulzBot.com`. Please completely read this manual before contacting for support questions or help. The latest version of this information guide is also available at `http://download.lulzbot.com`. You can also find more information including images, videos, and updated versions of this manual in the Support section of LulzBot.com.

7.2 `Support`

Email: `Support@LulzBot.com`
Phone: +1-970-377-1111 x610

7.3 `Regional Phone Numbers`

Regional phone numbers are available at `LulzBot.com/contact-us`.

7.4 `Community`

Community Support and Resources

- LulzBot 3D printer forum: `Forum.LulzBot.com`
- LulzBot Online Community: `LulzBot.com/community`

PROOF

Warranty Information

PROOF

8.1 Warranty

Warranty information for your LulzBot® TAZ 3D printer can be found at https://www.lulzbot.com/support/warranty

Extended Warranty

Optional extended warranty terms are available for purchase at https://www.lulzbot.com/store/parts/lulzbot-taz-extended-warranty.

8.2 Modification Warning

WARNING: At Aleph Objects, Inc® we respect your freedom to modify your LulzBot desktop 3D printer. However, any modifications or attempted repairs that cause damage are not covered under the Warranty. Questions? Contact Technical Support by emailing Support@LulzBot.com or by calling +1-970-377-1111.

PROOF

Contact Information

PROOF

9.1 Support

Email: `Support@LulzBot.com`
Phone: +1-970-377-1111 x610

9.2 Sales

Email: `Sales@LulzBot.com`
Phone: +1-970-377-1111 x600

9.3 Regional Phone Numbers

Regional phone numbers are available at `LulzBot.com/contact-us`.

9.4 Websites

Aleph Objects, Inc., the makers of the LulzBot® brand of 3D Printers:
`www.AlephObjects.com`
LulzBot 3D Printers and parts:
`www.LulzBot.com`
`Forum.LulzBot.com`

PROOF

Index

Symbols
1.75mm filament, 71

A
ABL, 27
ABS, 70
acetone, 67, 70
Adhesion Type, 35
Advanced Options, 36
advanced techniques, 70
Airgap, 47
aluminum extrusions, 74
Australia, x
auto-leveling, 68
Automatic Bed Leveling, 27

B
Base Line Width, 47
Base Thickness, 47
Basic Options, 32
bed adhesion, 70, 71
bed plate, 74
Bed Temperature, 34
belts, 67
bill of materials, 74
Black Magic, 46
Blender, 14
blowers, 68
BotQueue, 14
bottle, 70
Bottom Layer Speed, 38
Bottom Thickness, 33
Brim, 20, 35, 46
brim, 70
burns, viii
bushings, 66

C
CAD, 13
Calibrating Extruders, 50
calibration, 74
Combine Type-A, 48
Combine Type-B, 48
Combing, 43
community support, 76
Cooling, 44
Cura, 18, 74
Custom GCODE, 41
Cut Off Object, 37

D
damage, 78
download, 12, 74
drive rods, 66
driver, 12
Dual Extrusion, 49
Dual Extrusion Overlap, 38

PROOF

E
eeprom settings, 60
electronics, viii, 68, 74
Enabling Cooling Fan, 39
End GCODE, 41
Expert Settings, 42
extended warranty, 78
Extensive Stitching, 48
Extra Margin, 47
extruder, viii
extruder jam, 67

F
failed repair, 78
fan, 68
Fan Settings, 44
FCC, ix
filament, 20
Filament Diameter, 36
Fill Amount, 45
Fill Density, 33
First Dual Print, 51, 52
First Print, 27
Fix Horrible, 48
Flow Rate, 36
forums, 76
FreeCAD, 13
Freenode, 76
Full Settings, 31

G
g-code, 12
GCODE, 12
gcode, 74
Ghost View, 25
glass, 66, 67
GLCD, 27
Glue stick, 29, 71
gluestick, 71
GNU/Linux, 12–14
Graphical LCD controller, 27
grease, 66

H
hardware, 74
hazards, viii
heater block, viii
Hexagon, 74
hexagon, 74
High Detail, 19
high resolution, 70
High Speed, 20
hobbed bolt, 67
Host, 14
hot end, 67, 70

I
ICES, x
Infill Speed, 38
Initial Layer, 37
Initial Layer Width, 37
Inner Shell Speed, 39
Installation, 18
Interface Line Width, 47
Interface Thickness, 47
IPA, 70
IRC, 76
Isopropyl Alcohol, 66
Isopropyl alcohol, 70

L
latex, 74
Layer Height, 32
Layers View, 26
lead screw, 66
leveling, 68
Line Count, 43

PROOF

Line Spacing, 47
Load Model, 20
lubricant, 66

M
maintenance, 66
Marlin, 74
Material Selection, 20
MatterControl, 15
Minimal Layer Time, 39
Minimal Length, 44
Minimum Travel, 43
modification, 78

N
New Zealand, x
Ninjaflex, 70
Normal View, 23
novelties, 74
nozzle, 67, 70
nozzle cooling fan, 68
Nozzle Size, 36
nozzle wipe, 67
Nylon, 70, 71

O
OctoPrint, 14
Open Faces, 48
OpenSCAD, 13
operating system, 12
Orientation, 21
OS X, 12–15
Outer Shell Speed, 39
Overhang Angle, 45
Overhang View, 24

P
Pause, 29
Pause at Z Height, 40
PEI, 70
PEI surface, 66
Plugins, 39
power supply, viii
print cooling fan, 68
Print Speed, 33
printed parts, 74
Printer Host, 12
printing, 28
Printing from SD, 27
Printing from USB, 28
Printing Temperature, 34
probe, 68
probing, 68
Probing Temperature, 41
PVA, 71

Q
Quick Print Profile, 19
Quick Print Settings, 19

R
Raft, 35, 46
RAMBo, 68, 74
Removing a Print, 30
RepRap, 76
Resolution, 19
Retraction, 33, 42
Retraction Distance, 36
Retraction Speed, 36
Rotating Object, 21

S
SD, 27
SD card, 27
Second Extruder Offset, 50
Shapesmith, 14
Shell Thickness, 33
Skirt, 43

PROOF

Slicers, 12
Slicing, 12
smooth rods, 66
software, 12–14, 74
Source Code, 74
Source Files, 15
Spiralize, 46
Standard Quality, 20
Start Distance, 44
Start GCODE, 41
STL, 13, 14, 20, 74
Support, 45
Support Material, 20, 45
Support Settings, 45
Support Type, 34
Surface Layers, 48

T
technical support, 76
temperature, viii
Temperatures, 29
terms and conditions, 78
threaded extension, 70
threaded rods, 66
Top Thickness, 33
Travel Speed, 38
Tweak at Z, 40

U
Updating Firmware, 49

V
View Options, 23

W
warnings, viii
warping, 70
warranty, 78
Windows, 12–15
wiping pad, 67
Wiping Temperature, 41
wires, viii

X
Xray, 25

Z
Z hop, 43
Z Offset, 61

PROOF

Glossary

.GCODE The file extension for G-Code files

Cura Cura LulzBot Edition is a cross-platform software package that combines a slicing engine with a printer host interface.

3D Printer Also referred to as additive manufacturing, is the process of fabricating objects from 3D model data, through the deposition of a material in accumulative layers.

ABS Acrylonitrile butadiene styrene thermoplastic. Usually extrudes at 230°C with the Budaschnozzle and 240°C - 250°C with the LulzBot Hexagon Hot End.

Acetone A colorless, volatile, flammable liquid ketone, (CH3)2CO, used as a solvent for ABS.

CAD Computer Aided Design

FFF Fused Filament Fabrication– the process of laying down successive layers of extruded filament to create a 3 dimensional object. As each layer of molten plastic is extruded into place, it fuses with the previous layer.

Filament Plastic material in a “string” like form, as it is fed to the printer.

Free Software Free Software (or Free/Libre Software) can be thought of as “free as in free speech, not just free as in free beer”, although most Free Software is available for no cost. Free Software can be copied, modified and is freely available for download.

GCODE The common name for the most widely used CNC programming language.

HDPE High-density polyethylene.

Heater block Machined from aluminum, the heater block generates heat with a heater resistor and uses a thermistor to measure the temperature.

Heater resistor A special type of resistor that is used to apply heat in a small area.

HIPS High-impact polystyrene. Usually extrudes at *230°C* with the Budaschnozzle and *230°C* with the LulzBot Hexagon Hot End

Hot end The hot end is the whole part where the plastic melts, including the nozzle, heater block, thermistor, and heat sink. The LulzBot Hexagon Hot End comes standard on the TAZ 5.

Layer height The thickness of each individual deposited layer of the three-dimensional model when cut with a slicing program.

Laywoo-D3 Wooden filament similar to PLA. Forty percent of its content consists of recycled wood. Usually prints at *180°C* to *210°C*. Color can be changed by varying the extrusion temperature.

Libre Innovation Aleph Objects uses Free/Libre Software to build and improve Open Source Hardware so that everything we create is free to be viewed, copied, and/or modified by anyone.

Nozzle The metal tip at the bottom of the hot end. It has a small hole where the plastic filament comes out of the printer.

Open Source Hardware Open source hardware is hardware whose design is made publicly available so that anyone can study, modify, distribute, make, and sell the design or hardware based on that design. The hardware source, the design from which it is made, is available in the preferred format for making modifications to it. For more information visit *http://www.oshwa.org/definition/*.

PLA Polylactic acid is a corn-based biodegradable polymer. Usually extrudes at *185°C* with the Budaschnozzle and *205°C* with the LulzBot Hexagon Hot End.

Polycarbonate A strong and impact-resistant thermoplastic. Usually extrudes at *300°C*.

PTFE Polytetrafluoroethylene is a synthetic fluoropolymer used in the Budaschnozzle for it's low coefficient of friction. This limits the Budaschnozzle top extrusion temperature to 250°C. The LulzBot TAZ 5 and TAZ 6 does not use a hot end with a PTFE insert and can reach a *300°C* extrusion temperatuere.

PVA Polyvinyl acetate is the main ingrediant in white glue, wood glue and the common school glue stick.

RAMBo [R]epRap [A)]duino-[M]ega compatible [M]other [Bo]ard. Designed by Joynnyr of UltiMachine.
Resolution In general terms, the resolution you print at can be determined by the layer height you use. The LulzBot TAZ can print at a layer height of 0.05mm to 0.50mm.

Slic3r Slic3r is a cross-platform 3D model slicing engine. It's used to process the 3 dimensional model into the GCODE (tool path) needed to physically generate the print.
Spool Plastic filament coiled and stored on a plastic reel. Preferred over 1.75mm filament due to improved feeding and better mounting options.

Thermistor A special type of resistor that changes resistance based on temperature. It is used to measure temperature on the nozzle and the heated bed.
Threaded extension Used to separate the heater block and nozzle from the PEEK insulator. The plastic filament passes through the threaded extension into the melting chamber.

PROOF

PROOF

PROOF

Colophon

Created with 100% Free/Libre Software

GNU/Linux

LaTeX Memoir

PROOF

9 780989 378482

FEED YOUR MACHINE

LulzBot® 3D printers are compatible with many different 3D printing filaments; not only can you 3D print with ABS and PLA, but also with metallic, stone-like, wooden, nylon, and even flexible materials. 3D print with confidence knowing that each filament option has been tested and integrated into Cura LulzBot Edition, the recommended software for your LulzBot® 3D printer.

LULZBOT.COM/FILAMENT

ISBN 978-0-9893784-8-2
90000
9 780989 378482

LULZBOT®

ALEPH
OBJECTS®
INCORPORATED