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

LULZBOT®

ALEPH OBJECTS® INCORPORATED

7.44 x 9.69
246 mm x 189 mm

LulzBot® Mini User Manual

Aleph Objects, Inc.

.186
4.7244mm

LULZBOT®
MINI
User Manual
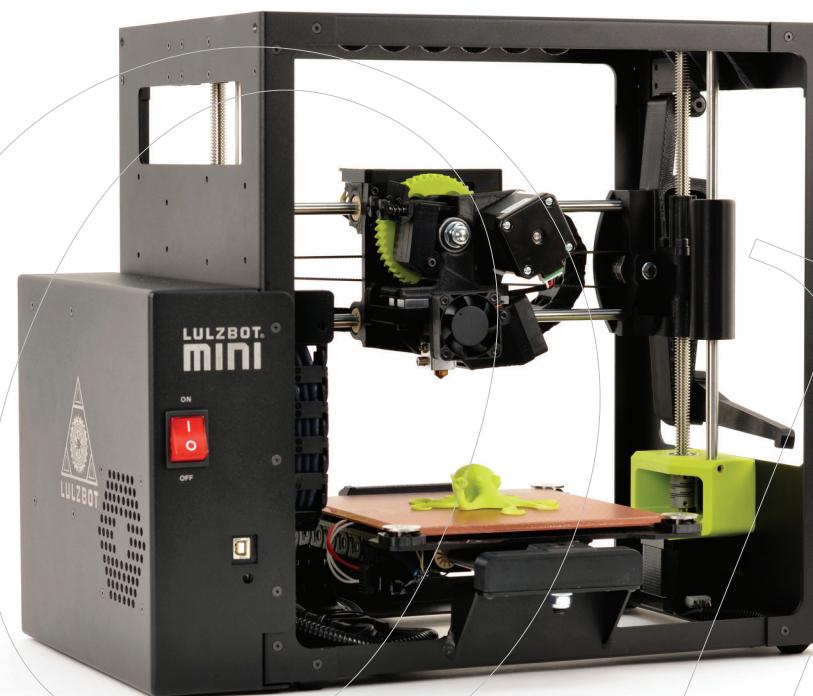
7.44 x 9.69
246 mm x 189 mm

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

LULZBOT

MINI

User Manual

# LULZBOT MINI USER MANUAL

LULZBOT®

Aleph Objects, Inc.

LulzBot® Mini User Manual

by Aleph Objects, Inc.

Copyright © 2016 Aleph Objects, Inc.

Contents

CONTENTS

iv

CONTENTS

List of Figures

WARNINGS
Safety Information

Read Me First!

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

Hazards and Warnings

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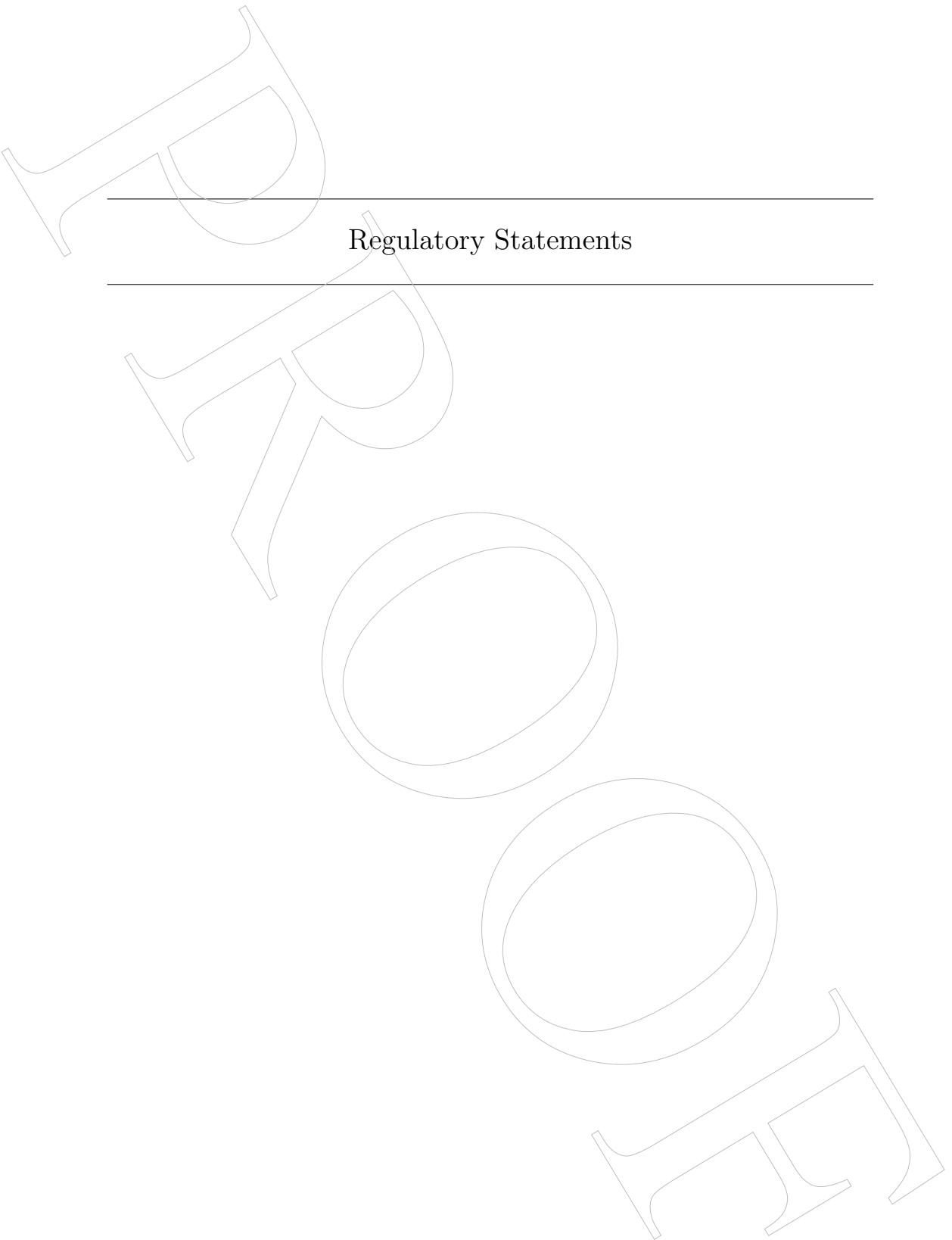
When the printer is operational take care to never put your fingers near any moving parts including belts, pulleys, or gears. Tie back long hair or clothing that can get caught in the moving parts of the printer.

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 Statements

Regulatory Compliance Statement Class B

Federal Communications Commission Statement

This device complies with part 15 class B of the FCC Rules. Operation is subject to the following two conditions:

1. This device may not cause harmful interference and

2. This device must accept any interference received, including interference that may cause undesired operation.

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.

FCC Warning: Changes or modifications not approved by the party responsible for compliance could void the users authority to operate the equipment.

Industry Canada Statement

CAN ICES-3 (B)/NMB-3(B) 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.

3D Printer Software

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 3D 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.

1.3 **Installing Drivers**

GNU/Linux and OS X users will not need to install a driver to communicate with the LulzBot Mini 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/.OBJ files sized in millimeters. If an .STL/.OBJ 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`

17

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.

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.
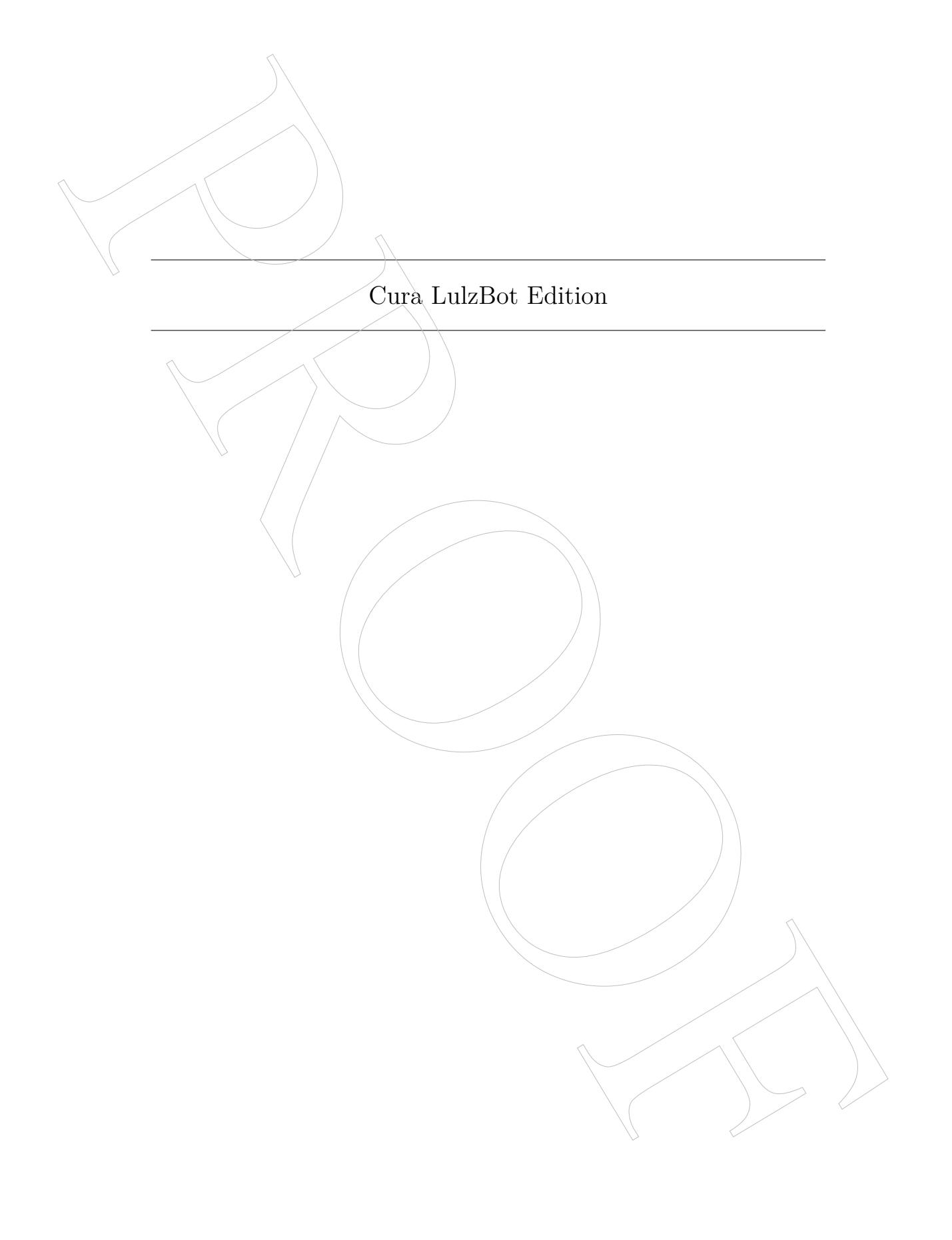
Source Files

Aleph Objects, Inc., the maker of the LulzBot Mini 3D printer, completely supports Free Software, Libre Innovation, and Open Source Hardware. Along with the LulzBot Mini 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`

Cura LulzBot Edition

2.1 **Cura LulzBot Edition**

Installation and Setup

Cura LulzBot Edition is available for download on our website at LulzBot.com/Cura. Uninstall any previous versions of Cura you may have been using when upgrading. 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, and tool head type.

It is important to select the correct printer, hot end, tool head, and if asked, the nozzle diameter as Cura uses custom profiles and machine settings based upon the selections made.

- Download the appropriate installer for your computer operating system. Installation instructions for each operating system are available at LulzBot.com/Cura.

- Install Cura by following the instructions available at LulzBot.com/Cura.

- 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 Mini.

- Select Standard LulzBot Mini Press Next.

- Select finish.

2.2 **Quick Print Settings**
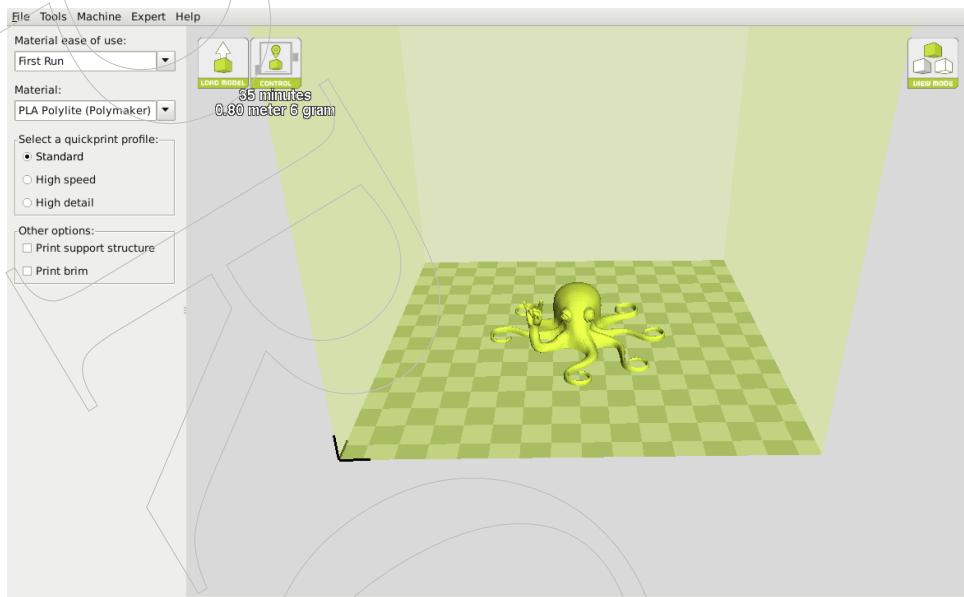
Figure 2.1: Quick Print Settings

After setting up Cura for the first time, you will be shown the main interface screen. (Fig. 2.1, page 23):

Material Selection

We have the different filament types 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 Mini ships with a filament sample for the first print. Refer to the included Quick Start Guide for the proper "First Run" settings.

Different Filament Manufacturers

Different manufacturers have different formulations for their specific brand. These different formulations may have different ideal settings. We typically consume 6kg - 10kg of filament when developing these profile settings. We highly recommend using the filament brands listed in Cura LulzBot Edition.

23

Beautiful 3D printed objects start with reliable and consistent filament. Our profiles will be good starting points for other manufacturers but they may not be ideal and may need updating on your part.

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 have fewer profiles.

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 will reduce printing time.

High Speed

Designed for the fast prints, where overall model finish is not of concern. Most commonly used for quick iteration of designs found in rapid prototyping.

Printing Support Material

The LulzBot Mini 3D printer is able to print models that have angles and overhangs, even without support material depending on the overhang distance and angle. Turn this option on if sections of your model are extending in mid air. This will build up material underneath the portion extending in mid air, preventing gravity from making it droop.

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 3D 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. You can view your model from different angles by holding down the right mouse button and dragging.
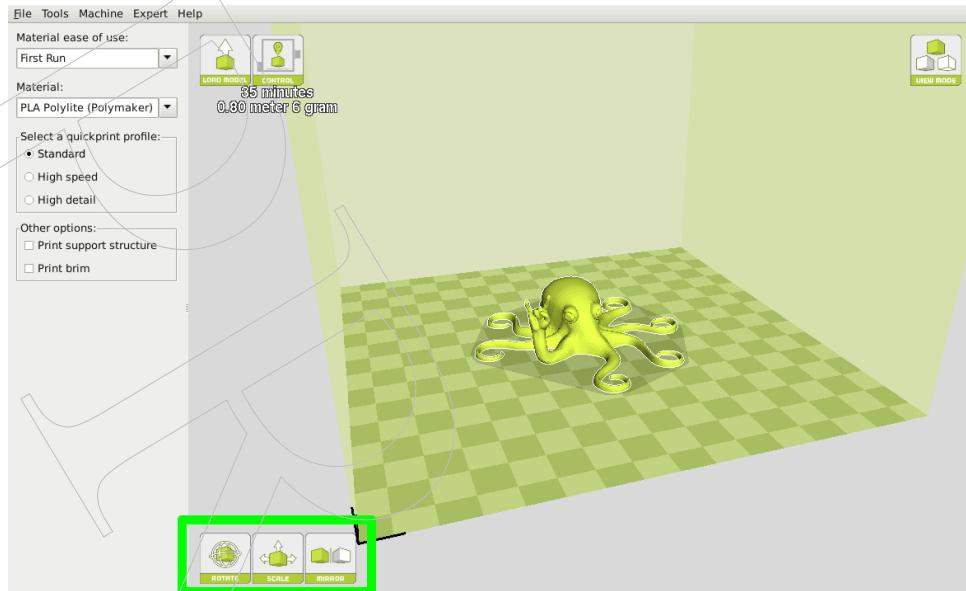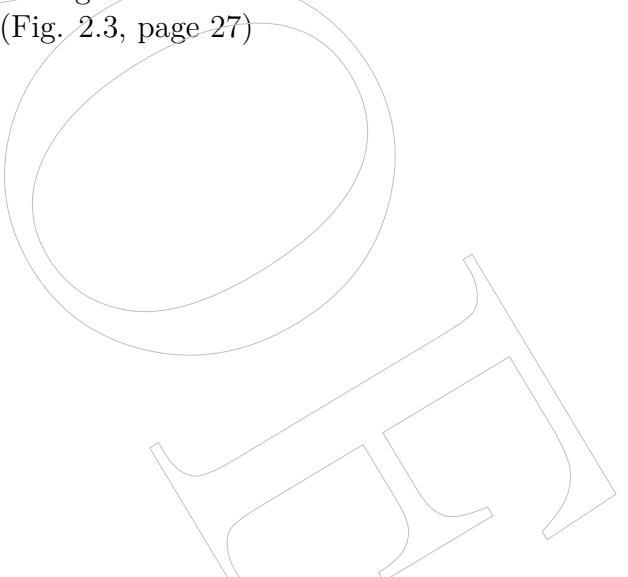
Figure 2.2: Options after selecting model

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. (Fig. 2.3, page 27)
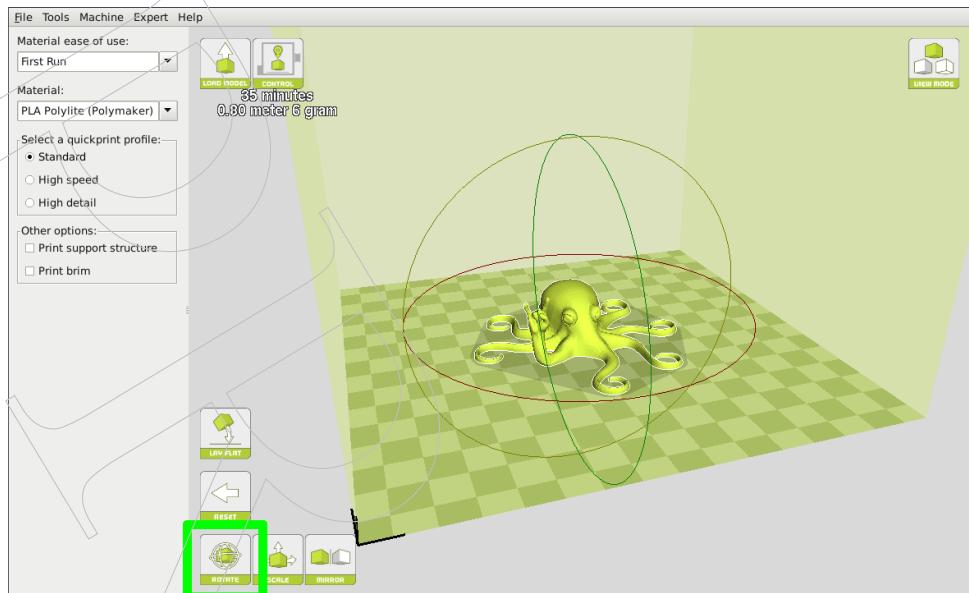
Figure 2.3: Rotating your Model

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.
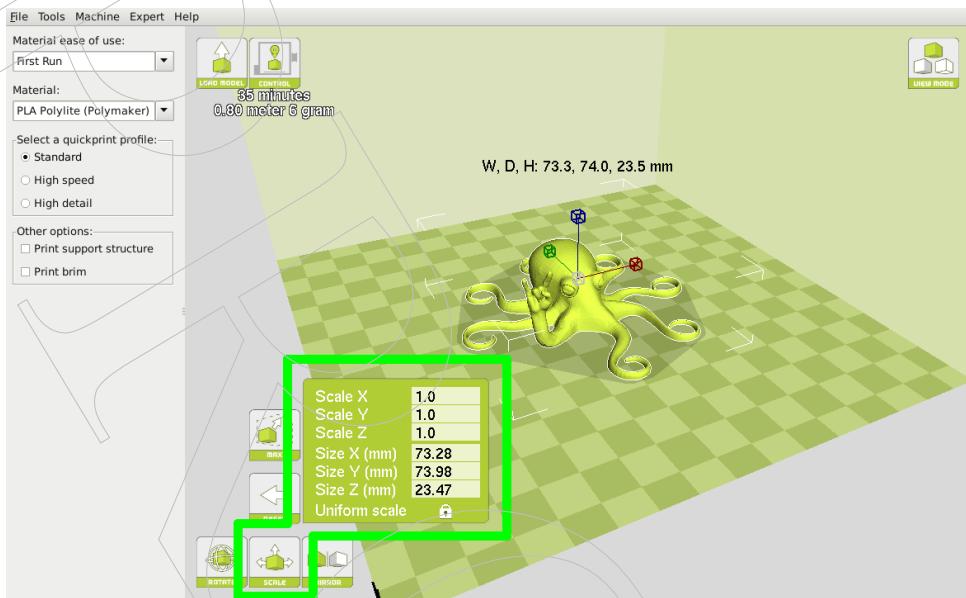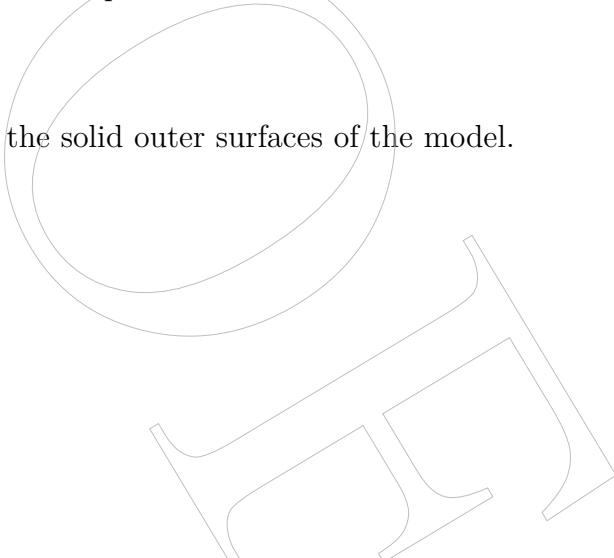
Figure 2.4: Scaling your Model

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 29):
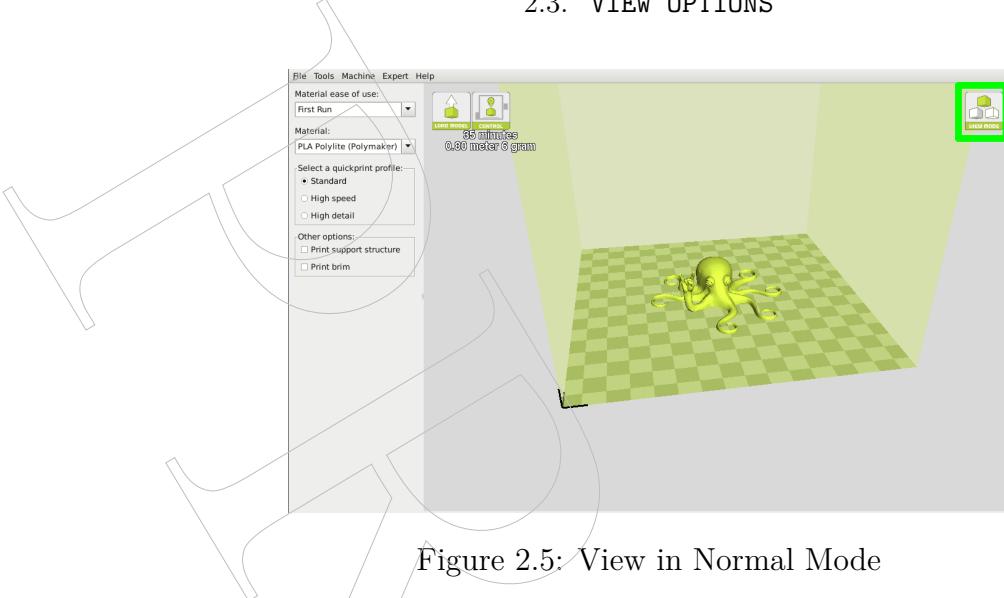
Figure 2.5: View in Normal Mode

Overhang

Overhang mode shows where your model may need support material. In Fig. 2.6, page 29 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.

Figure 2.6: View in Overhang

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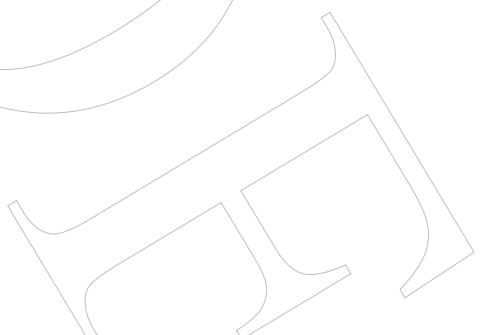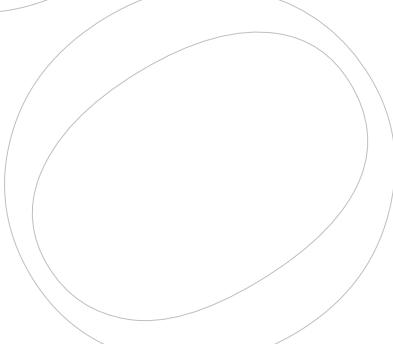
Ghost view mode makes the model translucent to allow you to see what is behind it.

Figure 2.7: View in Ghost

Xray

X-ray allows you to look inside of the object. This is helpful for detecting any manifold errors or other possible issues with your model. Problem areas will be highlighted in red. (Fig. 2.8, page 31)
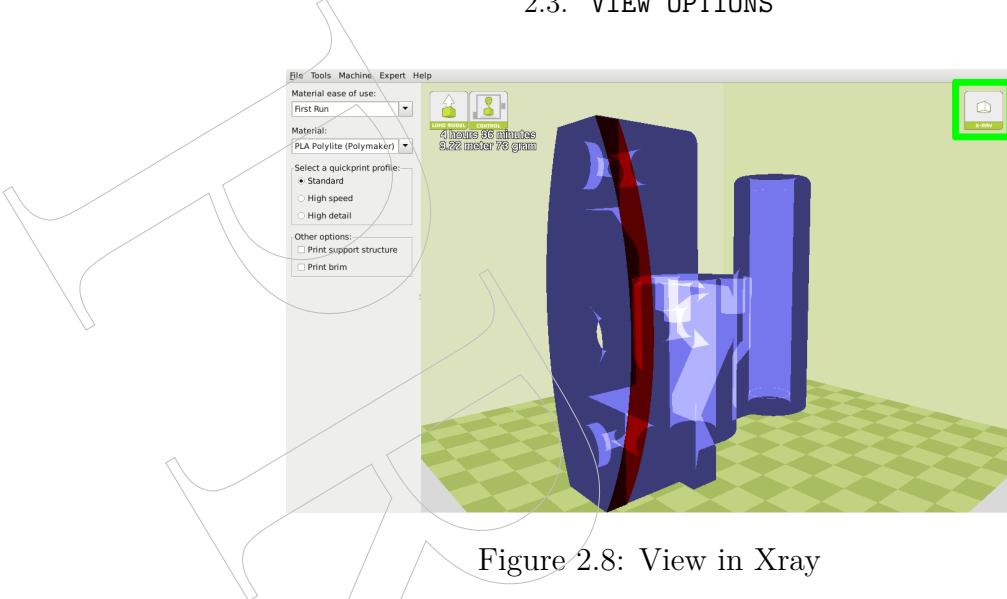
Figure 2.8: View in Xray

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, support structure will be shown in blue.
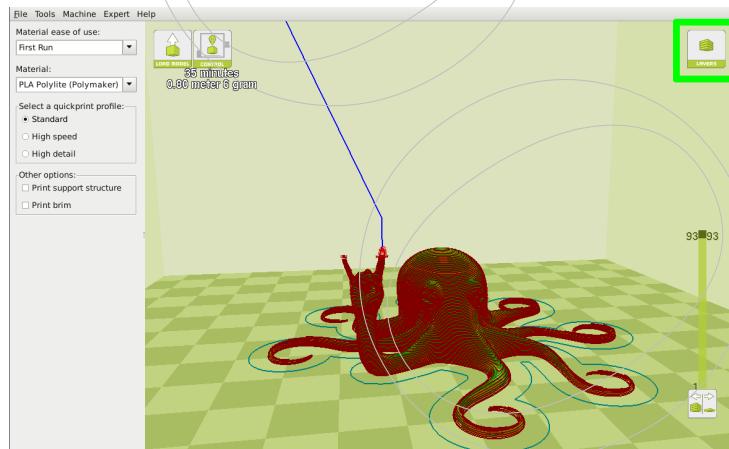
Figure 2.9: View in Layers
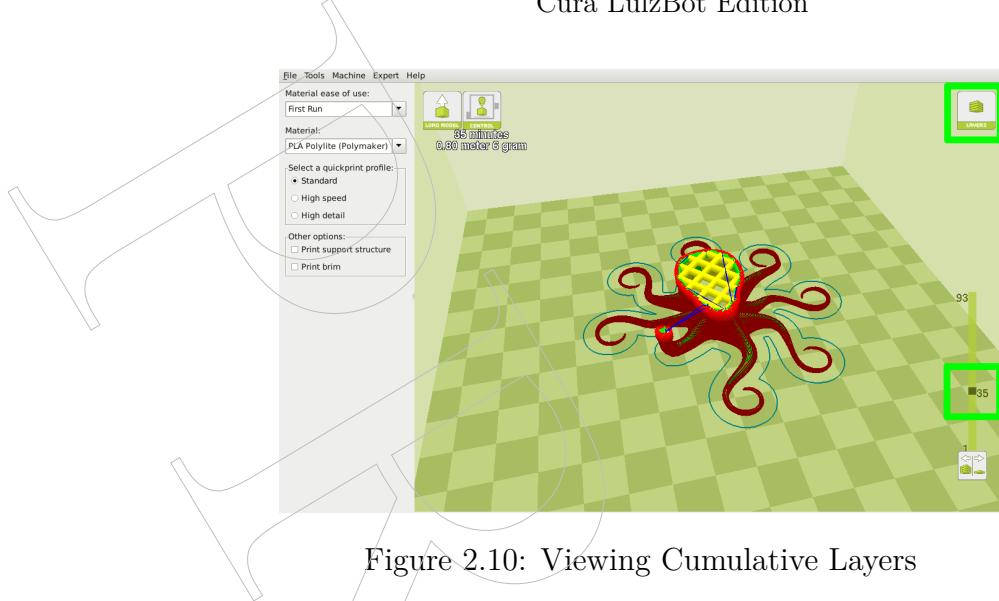
Figure 2.10: Viewing Cumulative Layers
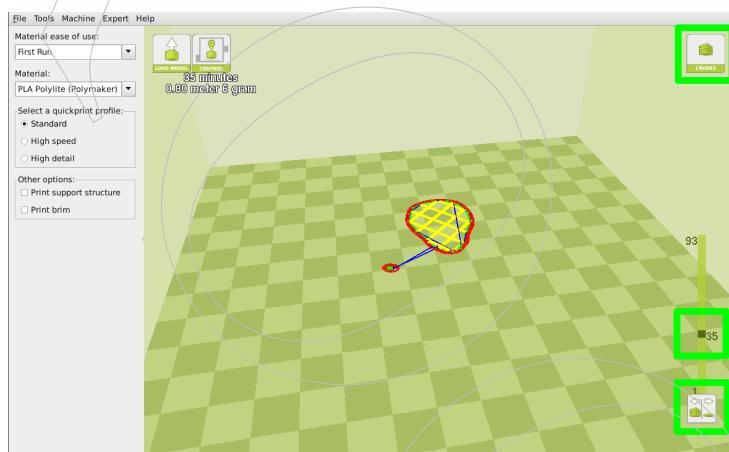
Figure 2.11: Viewing An Individual Layer

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.
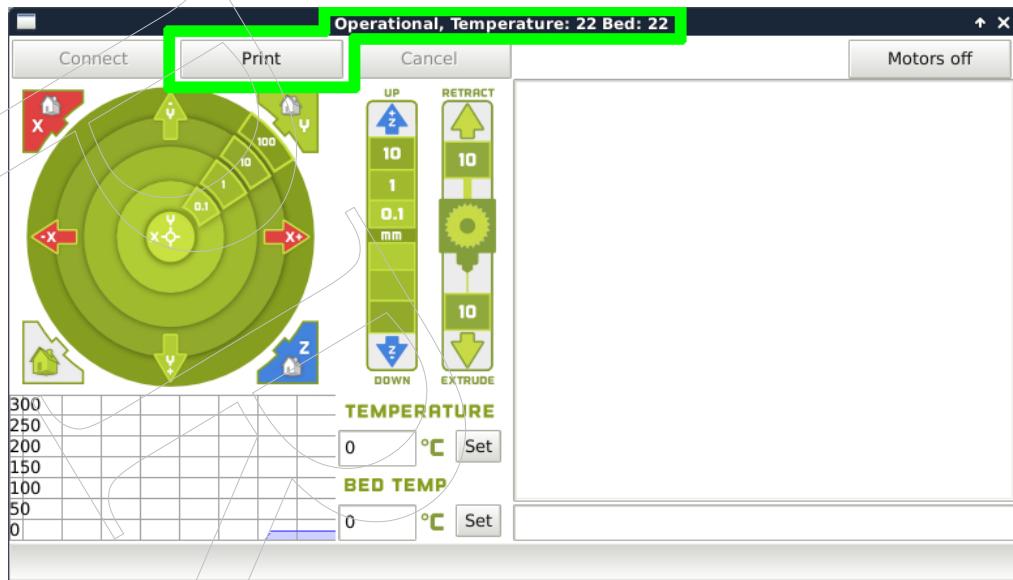
Figure 2.12: Control Screen

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 your print and automatically move your print head away from your object. This will allow color changes or material changes mid print.

Automatic Bed Leveling

Before each print your Mini 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 Mini 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 your Mini does not re-wipe after a failed probe, update your firmware. Directions for updating firmware can be found in section 4.3 on page 64. If probe failures are 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 48

33

for details on changing these temps.

Recommended Temperatures

Different filaments have different ideal temperatures for extrusion, bed adhesion, and part removal. Your LulzBot Mini will have these automatically set when using our recommended profiles. We have found that for certain materials a glue stick 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.

Filament Type	Bed Preparation	Hot End 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
corkFill	Clean PEI	185-195	70	50
woodFill	Clean PEI	185-195	60	50
Linen	Clean PEI	195-205	60	50
Silk	Clean PEI	165-175	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
High Temp PLA	Clean PEI	220-230	60	40
nGen	Glue stick	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
PVA	Glue stick	180-200	60	45
PC-Max	Glue stick	240-270	100	50
colorFabb_HT	Glue stick	260-280	110	50
colorFabb_XT	Glue stick	240-250	80	50

Table 2.1: Recommended Temperatures
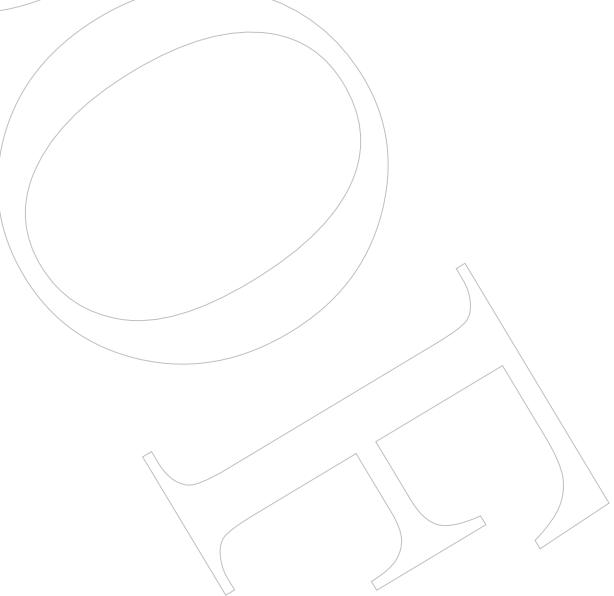
2.5 `Removing Your First Print`

After your first print has finished, wait for the part to cool down. Your parts will be easier to remove if you allow your heated print surface to cool to the optimal temperature. This will allow the plastic to contract, making it easier to remove. `The Y-axis will move forward once it reaches the ideal print removal temperature.`

Once the heated print surface 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 the print and print surface. Once underneath the part rotate the blade- lifting with the sharp edge under the part, to gently pop the print off.

2.6 `Full Settings`

When you first switch to Full Settings, Cura will need to know what filament, manufacturer, 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. As new filament is released for sale in the LulzBot store tested profiles will be available here: `LulzBot.com/Cura`. Download the profile that matches your filament and quality needs and load the file into Cura by selecting `File > Open Profile`. This will automatically update all of the Cura settings for that specific filament.
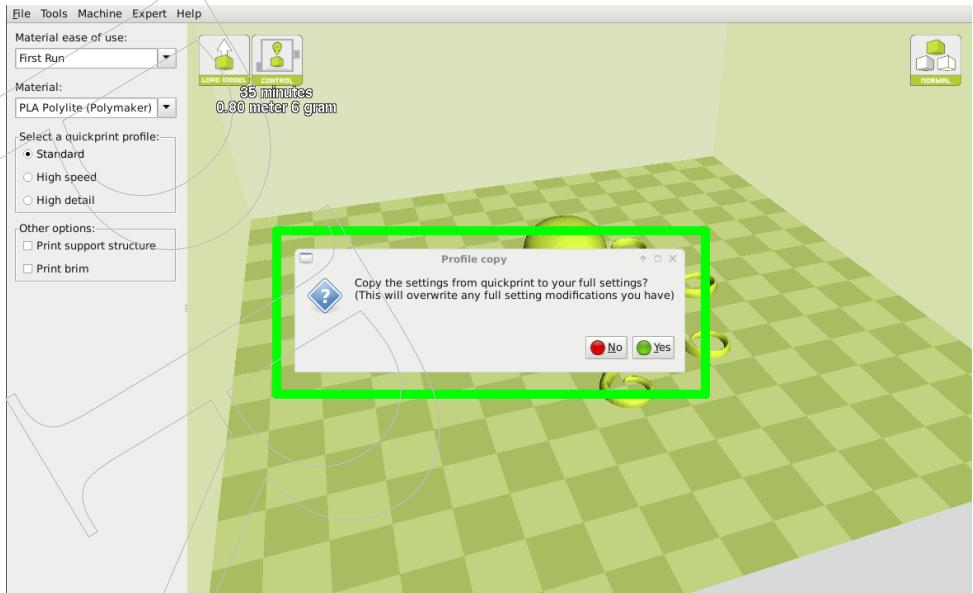
Figure 2.13: Transferring a Profile

Once the switch has been made to full settings, you will now have access to a wide variety of options. You will notice 4 new tabs: Basic, Advanced, Plugins, Start/End-Gcode. In the following sections we will describe each option, and how they will affect your prints.
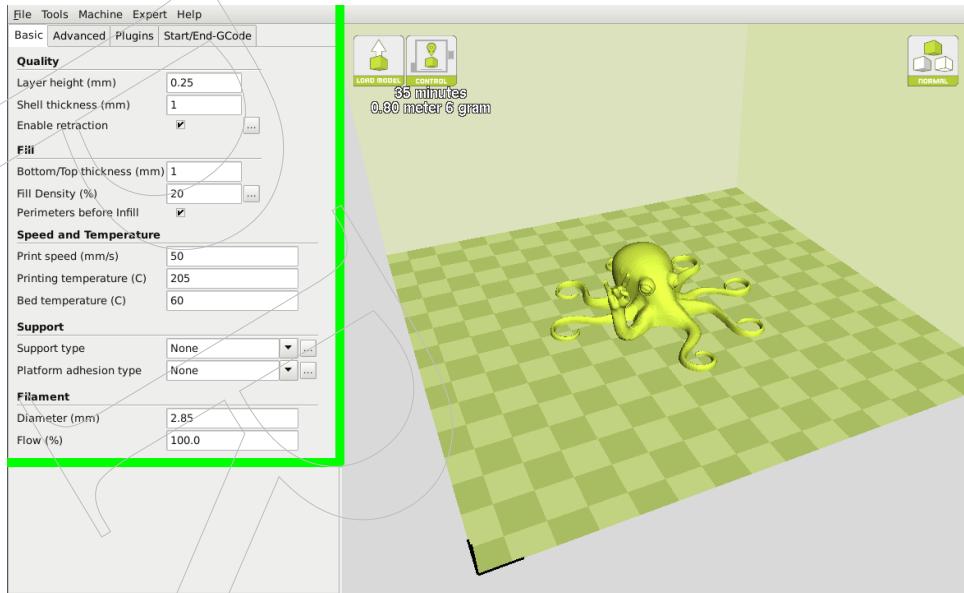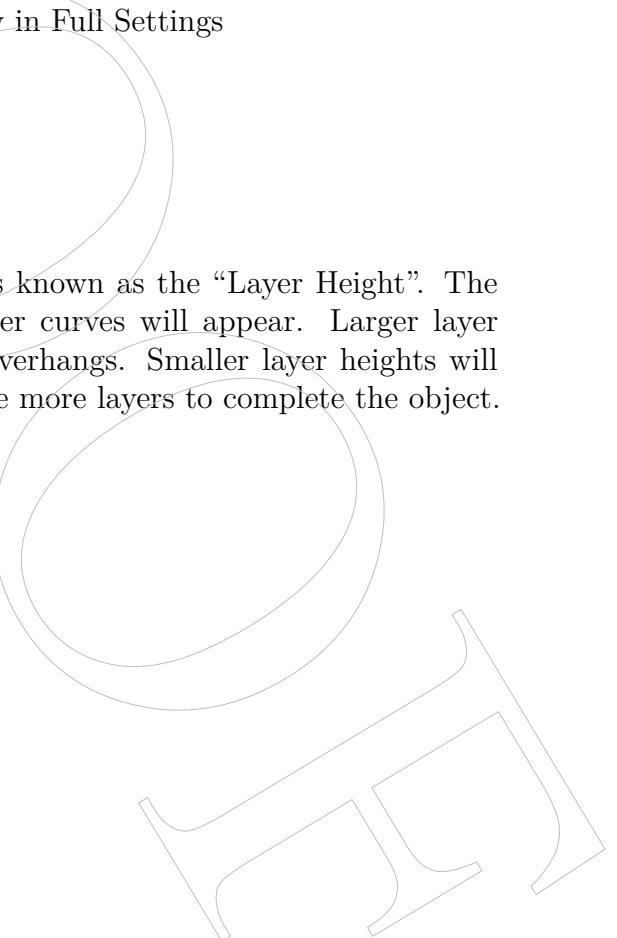
Figure 2.14: View in Full Settings

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.

0.4mm   0.25mm   0.1mm
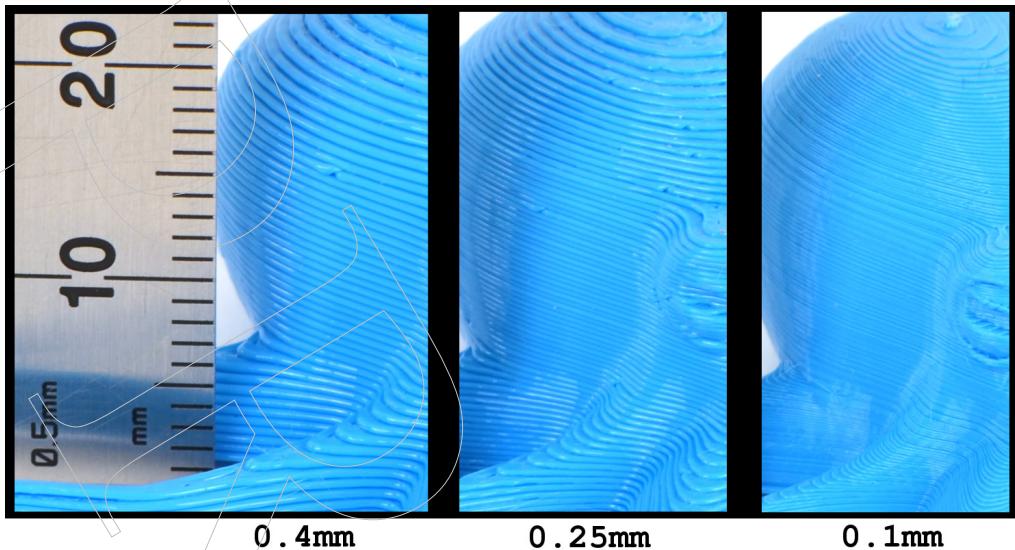
Figure 2.15: Differences in Layer Height

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 LulzBot Mini 3D printer is equipped with a 0.5mm nozzle. A setting of 1.0mm or 1.5mm is a sufficient for most prints.

Enable Retraction

Retraction pulls filament out of the hot end upon travel moves– when the 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 surface which can be helpful for strength, bridging, and quality purposes. We recommend keeping this number as a multiple of your layer height.

39

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 change 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. This setting will be overridden in certain parts of the print based upon your Advanced Speed Settings. If no speeds are determined in the advanced settings tab, your printer will default to this setting.

Printing Temperature

This is where you will set the hot end temperature for your specific material and manufacturer. It will be preset when using the recommended quickprint profiles. This can be adjusted for fine tuning of different filament manufacturers to help layer adhesion or stringing. See Recommended Temperatures Table on page 35. Your hot end is capable of reaching a maximum temperature of 300°C.

Bed Temperature

This setting controls 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 printed object adhesion and release from different filament manufacturers. See the Recommended Temperatures Table on page 35. The print surface is capable of reaching a maximum temperature of 135°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 or design 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. (Fig. 2.16, page 41)

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. (Fig. 2.16, page 41)
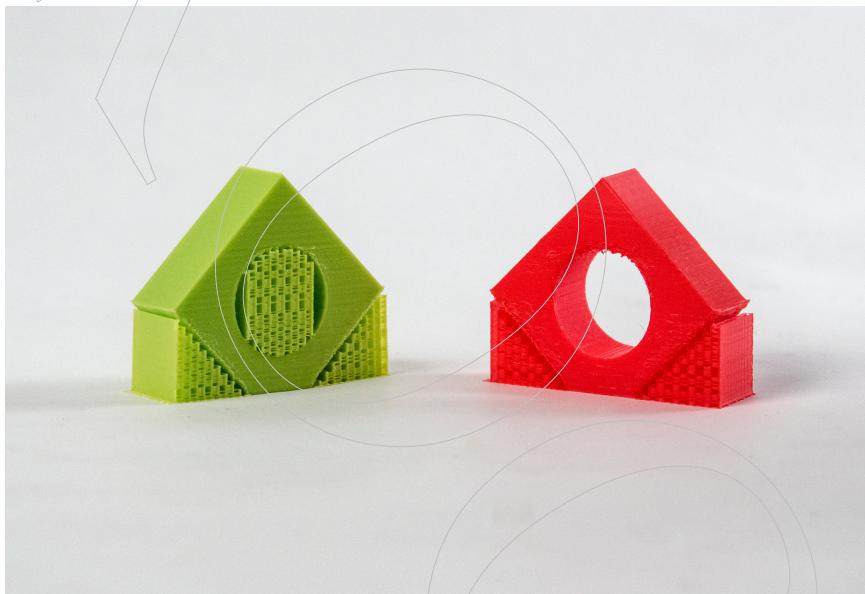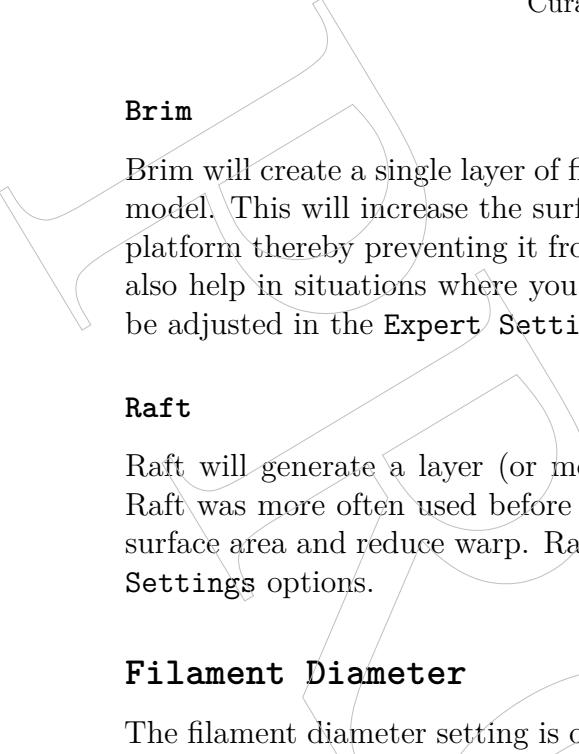

Figure 2.16: Support Types

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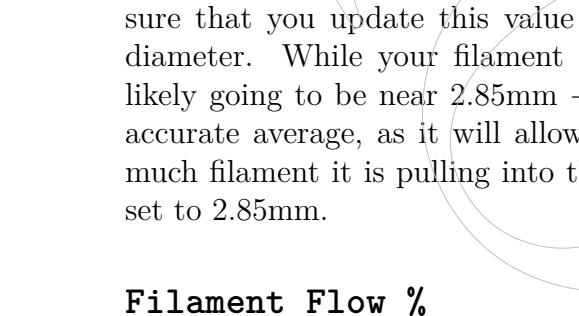
Some models have a small surface area contacting the plate. This can create adhesion issues causing your part to pop off at some point during the print. To fix this, use either Brim or Raft. Raft is better used when a model has small heated bed contact points and overhangs.

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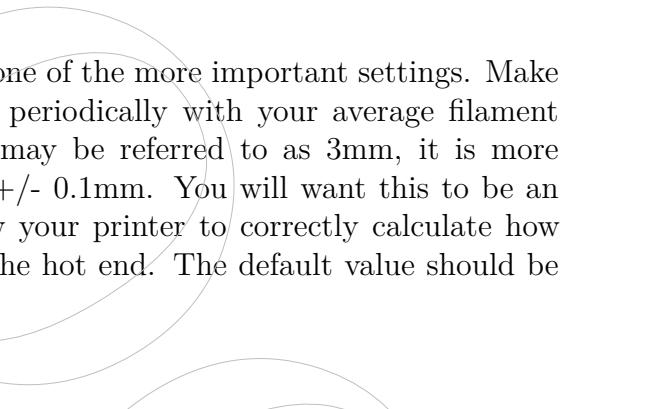
Brim will create a single layer of filament, contacting and surrounding your model. This will increase the surface area of the part contacting the build platform thereby preventing it from popping off the heated bed. Brim will also help in situations where you are seeing corner lift. Brim settings can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

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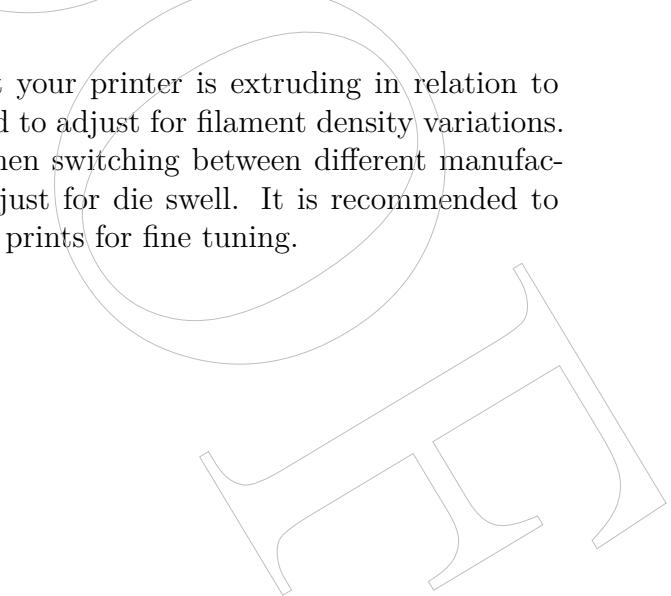
Raft will generate a layer (or more) of material underneath your object. Raft was more often used before the addition of heated plates to increase surface area and reduce warp. Raft settings can be adjusted in the `Expert Settings` options.

Filament Diameter

The filament diameter setting is one of the more important settings. Make sure that you update this value periodically with your average filament diameter. While your filament may be referred to as 3mm, it is more likely going to be near 2.85mm +/- 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 changed 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**
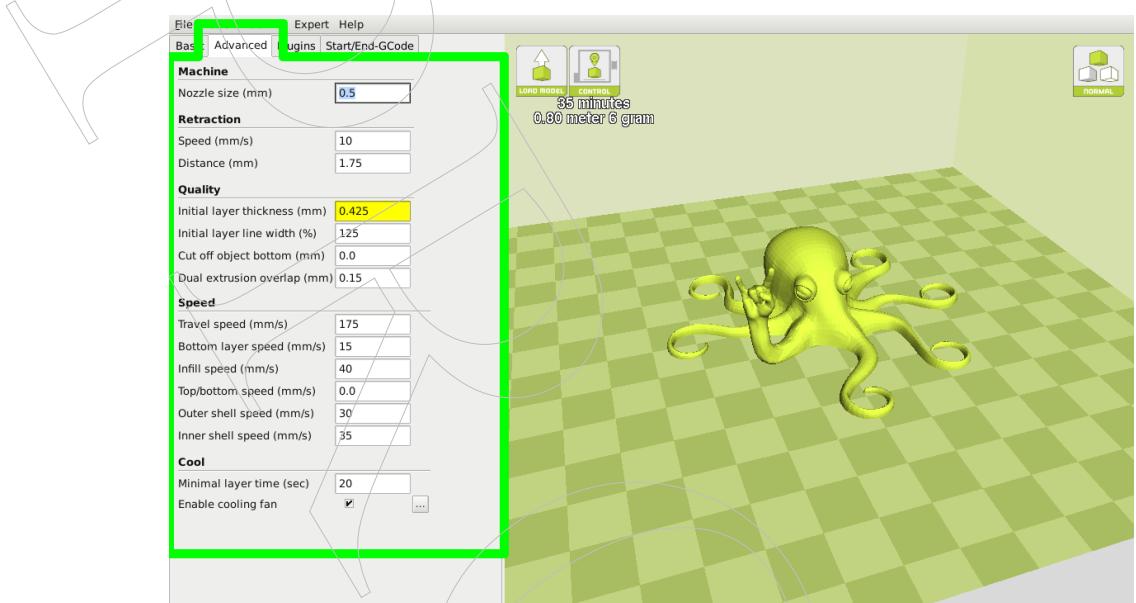
Figure 2.17: View of Advanced Tab

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, and how to generate the tool path. The Mini ships with a 0.5mm nozzle.

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 Mini automatic bed leveling system could be affected if you change this from the standard profiles. Adjust at your own risk. If you want to change first layer "squish" see Z Offset 4.4, page 65

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 Mini automatic bed leveling system could be affected if you change this from the standard profiles. Adjust at your own risk. If you want to change how much your first few layers "squish" to the sides, see Z Offset 4.4, on page 65

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 one object as two (or more) pieces. (Fig. 2.18, page 45)
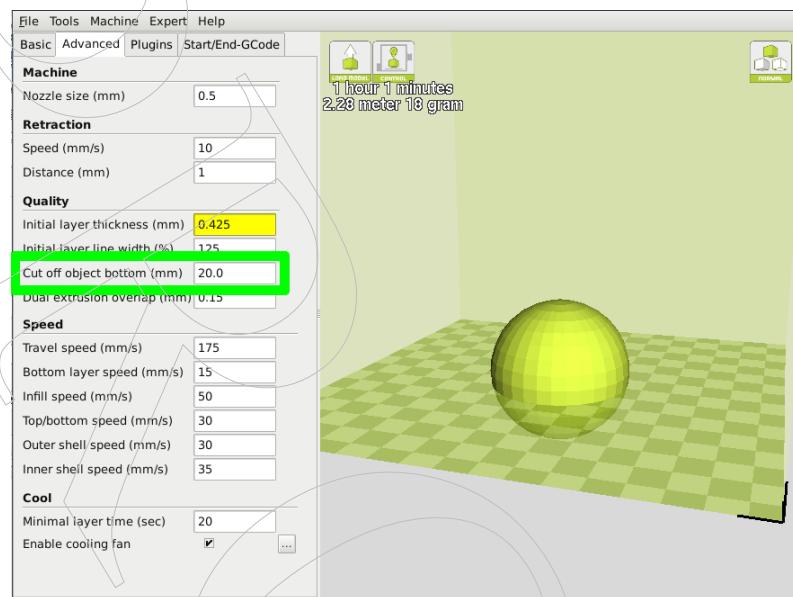
Figure 2.18: Cutoff Example

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 a printer is equipped with two hot ends and two extruders.

Travel Speed

This setting will determine how fast your print head moves while not extruding filament. A normal travel speed of 125mm/s - 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 globs 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 speed 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. If the tool head moves too slowly and lingers over the object, it can cause those areas to droop from excess heat, reducing final quality and finish.

Enable Cooling Fan

Enables operation of your extruder active cooling fan. The fan settings can be adjusted in the `Expert Settings` options. See section 2.14 on page 52 for details. In general, smaller parts will require more cooling for proper results.

2.9 `Plugins`

Plugins are custom settings which 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.`
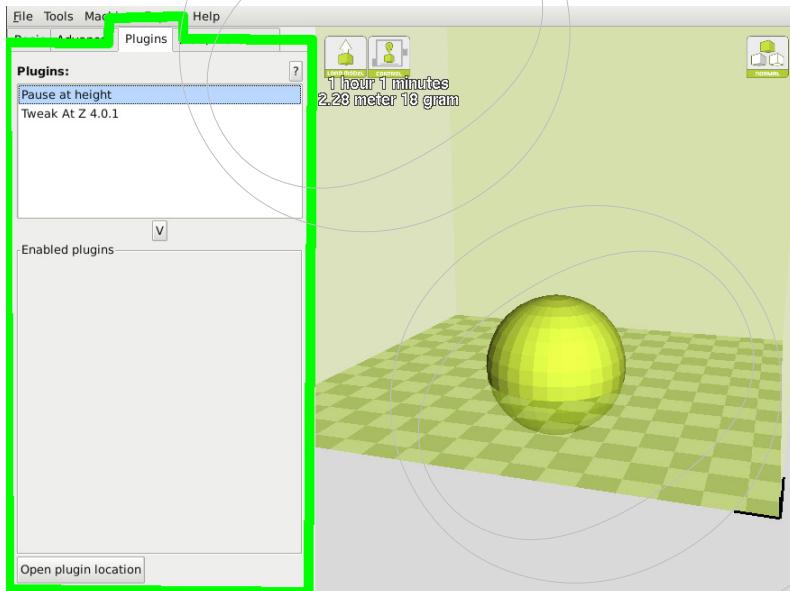

Figure 2.19: View of Plugins

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 LulzBot.com/download

Mini 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 beforehand.

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.

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`.
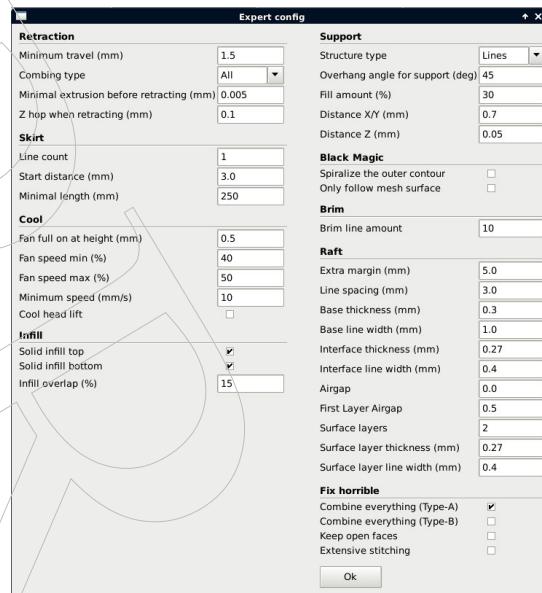
Figure 2.20: View Expert Settings

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 at least 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 input X mm of filament into the hot end. This is length of filament input into the hot end, not length extruded.

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. This is 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. The distance from the model will consume a portion of the maximum build volume in the X/Y plane.

Minimal Length

This will define the minimum extruded line length for the skirt. This will override 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.

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. A conservative setting for this is 45 degrees.

Fill Amount

This will determine how dense your support material is printed, similar to Infill Percentage. The higher the percentage, the better support, but it will be harder to remove the support material, will use more material, and will lead to a longer 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. A larger distance entered here can make the material easier to remove, but it will not support as well.

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. This will also ignore the base layer, and the top layer. The difference between this and Spiralize, is that the Z-axis moves regularly. That is, it prints a layer and then moves up vertically to begin 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. The distance around the base of the object will consume some of the maximum build volume in the X/Y plane. 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 the raft thickness.

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 determines the width of the top layers of the support platform. 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 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 accidentaly 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 **3D Model Attribution**

Octopus_v04.stl

Octopus_v04.stl by yeoldebrian was used under the CC BY-SA license `thingiverse.com/thing:7900`

Rocktopus.stl

Rocktopus by Kent Johnson is licensed under CC BY-SA 4.0 and derived from work by dietz1 and yeoldebrian. LulzBot.com/rocktopus

Support.stl
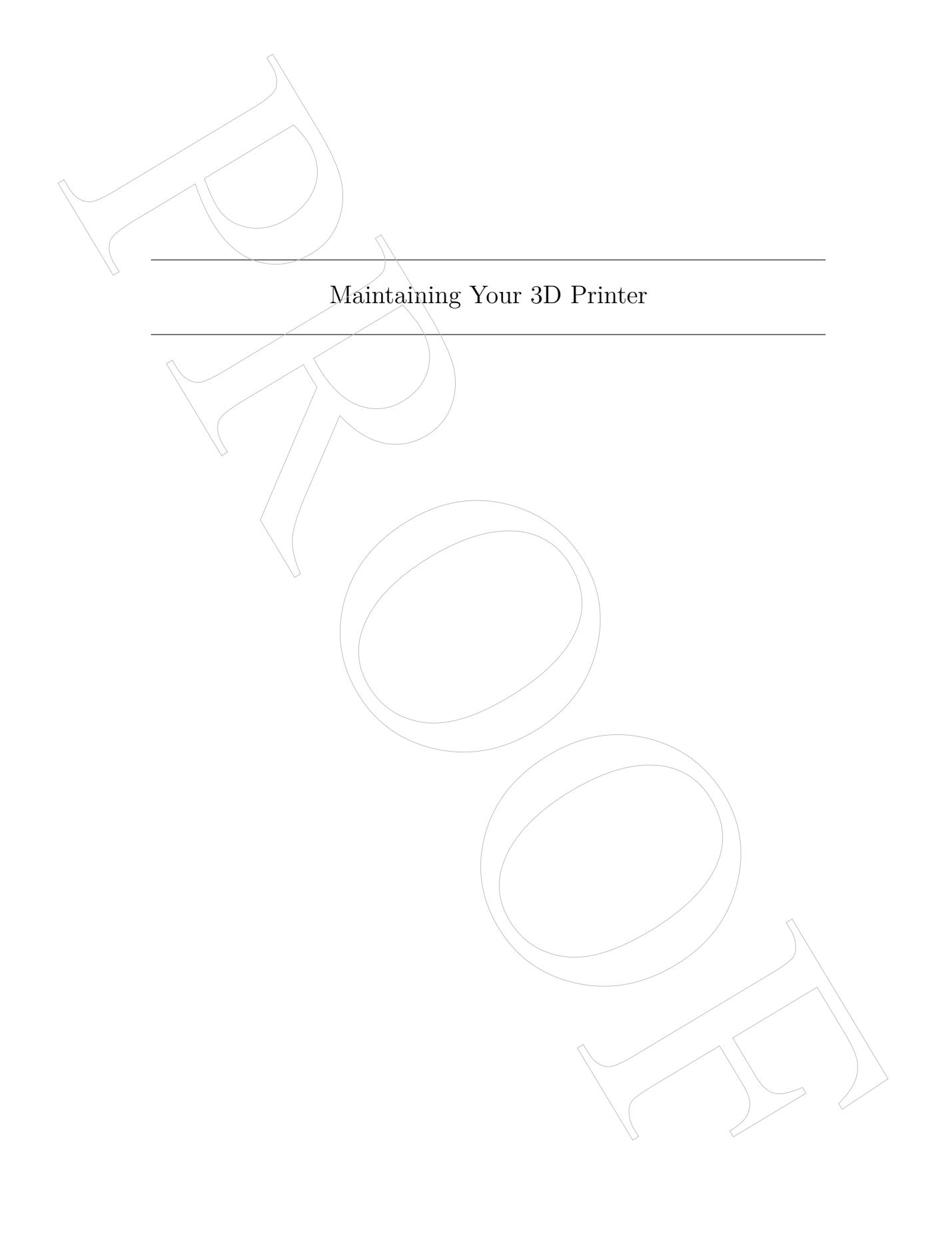
Support Model by Aleph Objects, Inc is licensed under CC BY-SA 4.0 license.

Overhang_test.stl

Make: 2015 3D Printer Shoot Out Test Models by MAKE was used under the CC BY-SA license. thingiverse.com/thing:533472

X_end_idler_Mini_v2.6.stl

X_end_idler_Mini_v2.6.stl by Aleph Objects, Inc is licensed under CC BY-SA 4.0 license.

Maintaining Your 3D Printer

3.1 `Overview`

Little maintenance is required keep your LulzBot Mini 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.

3.2 `Smooth Rods`

Wipe the smooth steel rods with a green scrub pad, clean cloth, or paper towel. The linear bushings leave a solid lubricant that builds up over time. Squeaking noises while the printer is in operation is likely a sign that the smooth rods need to be cleaned. `NOTE: Never apply any lubricant or cleaning agent to the smooth rods as the bushings are self-lubricating.`

3.3 `PEI Print Surface`

The PEI print surface is the key to well-balanced part adhesion and release. While long-lived, it will need to be replaced periodically and is considered a consumable item. 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 guides here: `LulzBot.com/learn`

3.4 `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 dental 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.

3.5 `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 binding and prevent full movement.

3.6 `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 as it can potentially short the control board.

3.7 `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 both sides are covered in plastic. 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, it process will fail.

3.8 `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
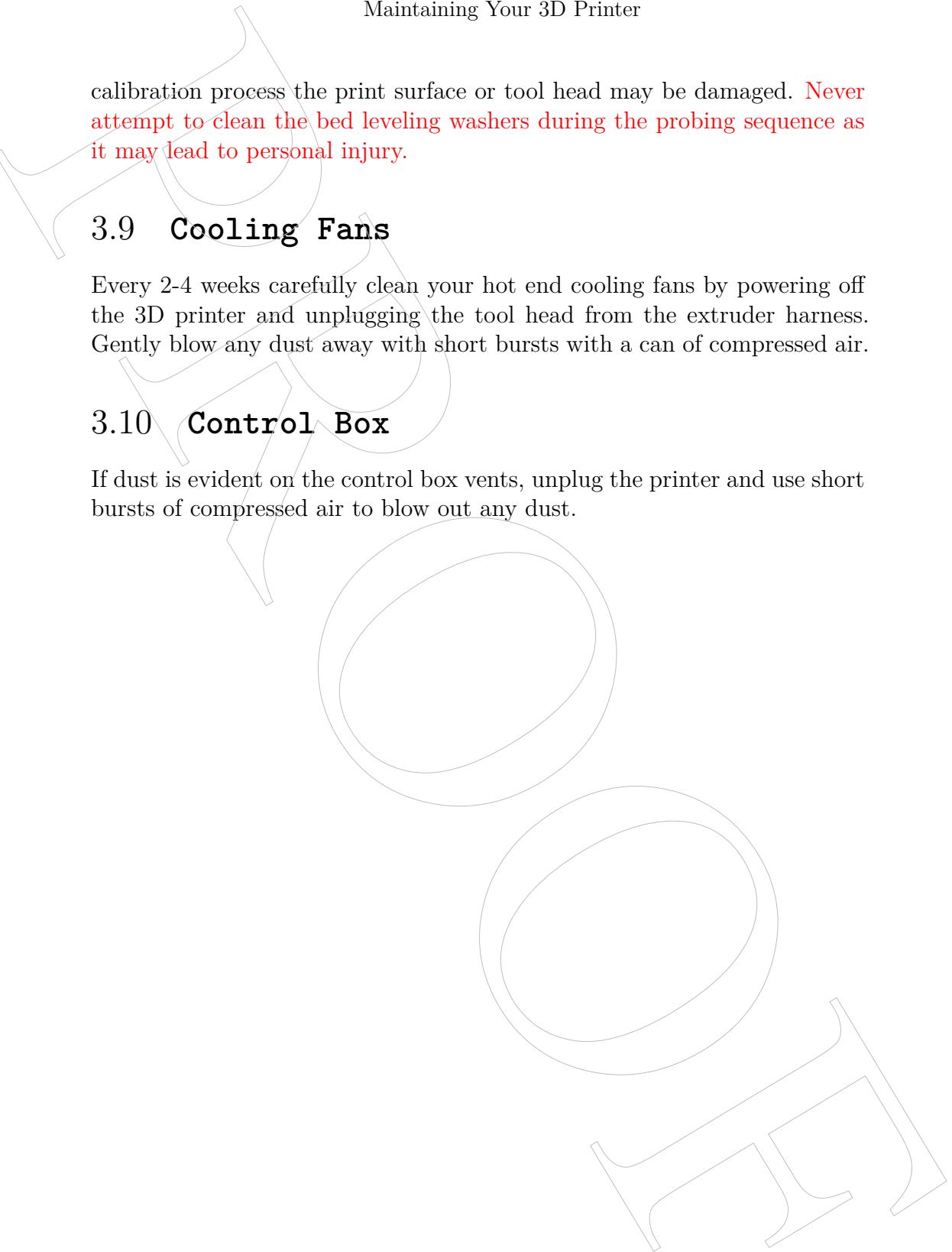
calibration process the print surface or tool head may be damaged. Never attempt to clean the bed leveling washers during the probing sequence as it may lead to personal injury.
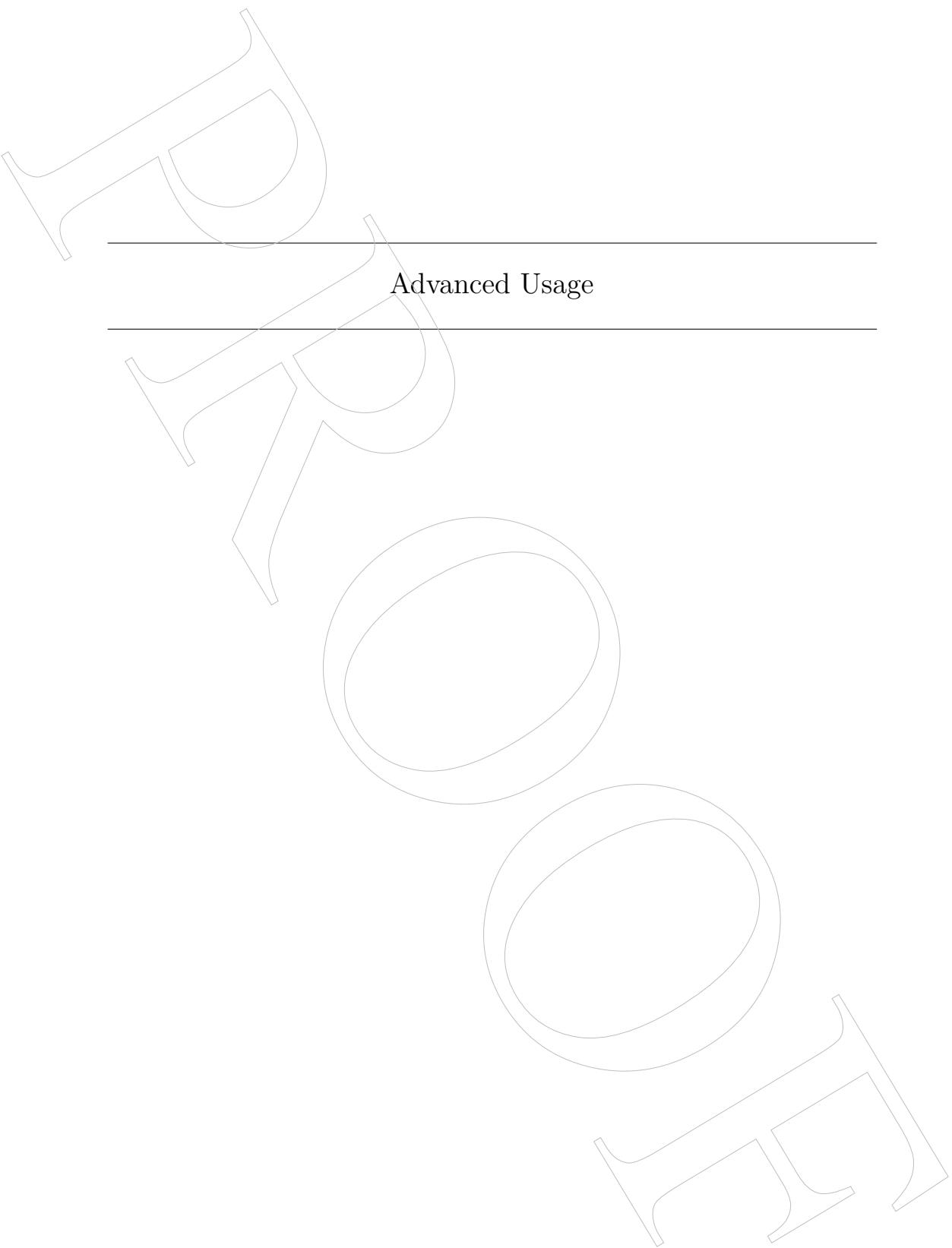
3.9 `Cooling Fans`

Every 2-4 weeks carefully clean your hot end cooling fans by powering off the 3D printer and unplugging the tool head from the extruder harness. Gently blow any dust away with short bursts with a can of compressed air.

3.10 `Control Box`

If dust is evident on the control box vents, unplug the printer and use short bursts of compressed air to blow out any dust.

Advanced Usage

4.1 **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 Mini 3D printer. Some of these instructions are items and materials not included with the Mini. With any of these additional items or materials, follow safety and usage guidelines as instructed by the manufacturer.

4.2 **Changing nozzles**

Hot end 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.

4.3 **Firmware Flashing**

We include the latest version of 3D printer firmware with each update of Cura LulzBot Edition. In order to update your firmware:

- Download the latest version of Cura LulzBot Edition from `LulzBot.com/Cura`.

- Install Cura by following the specific instructions for your operating system.

- Start Cura by launching it from your list of installed applications.

- Add a new machine by selecting `Machine > Add New Machine`.

- Select `LulzBot Mini`.

- Select the appropriate tool head.

- Install the latest stable firmware by selecting `Machine > Install Default Firmware`.

4.4 Z Offset

Your LulzBot Mini 3D printer has the ability to change the first layer height (Z offset) directly through Cura. In the lower right hand corner of the Control Window in Cura, enter the following commands. (The green text explains the command, do not enter it.)

- `M851` ; This will report your current Z offset

- `M851 ZXXX` ; This will update your Z offset to XXX

- `M500` ; Save new offset

To decrease the Z-offset between your nozzle tip and the print surface (decreasing the gap,) subtract from your current offset when changing XXX. To increase the Z-offset between the nozzle tip and the print surface (increasing the gap,) add to your current offset when changing XXX. When updating your offset, be sure to make small changes between prints. We recommend making changes using small, 0.1 mm increments at a time.

4.5 **Bed Adhesion**

You may find that during printing, printed objects lift off the print surface on the corners. Activate the `Brim` setting in Cura to help increase the surface area of the first layer of the print to improve part adhesion. If the corners of the printed object still lift, clean the PEI surface with IPA/ Isopropyl Alcohol and sand the surface with fine 2000-3000 grit sandpaper. While almost all 3D printing filaments will adhere well to the PEI print surface, some materials may need an application of a PVA glue solution/gluestick for improved print adhesion. These materials include nylon- and polycarbonate-based filament. Flexible materials such as Ninjaflex, SemiFlex, and some co-polyester-based filaments can adhere too well and can be released easier from the PEI print surface with an

application of a PVA glue solution/glue stick to the heated bed prior to starting the printing process.

ABS/Acetone Glue

Acetone is not included or required with the LulzBot Mini 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-, Polyester-, polycarbonate-, and Amphora™-based filament print adhesion can be improved with the application of an unscented PVA-based glue solution/glue stick. Flexible filaments such as NinjaFlex® and SemiFlex™ can adhere too well and require an application of PVA-based glue to the print surface for easy print removal. Failure to properly treat the print surface can lead to a permanent bond to the PEI print surface and will require replacement which may not be covered under warranty.

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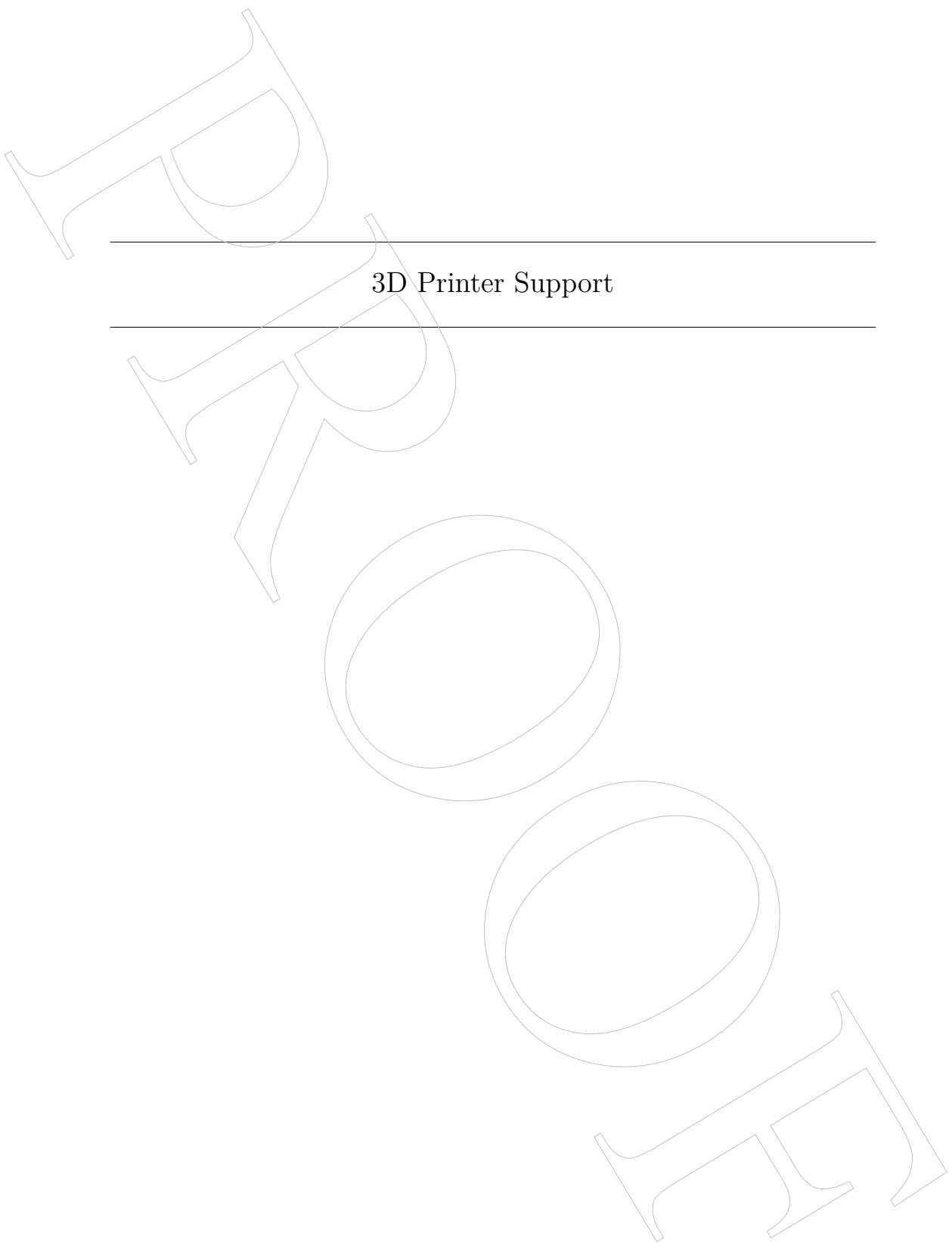
Your LulzBot Mini 3D printer is set up to use 3mm plastic filament by default and may be capable of printing 1.75mm filament with no hardware modification. While many of our advanced users are able to do so, your results may vary. More information can be found in our User Forums at: https://forum.lulzbot.com/viewtopic.php?f=16&t=1923

Hardware and Software Source Code

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

- 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.

  - Mini
  - 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.

  - Cura LulzBot Edition
  - Marlin

- Cura LulzBot Edition Print Profiles (vendor- and filament-specific configuration files).

  - `LulzBot.com/Cura`

3D Printer Support

6.1 `LulzBot`

For common technical support questions for your LulzBot Mini 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 `LulzBot.com/Download`. You can also find more information including images, videos, and updated versions of this manual at LulzBot.com/Support.

6.2 `Support`

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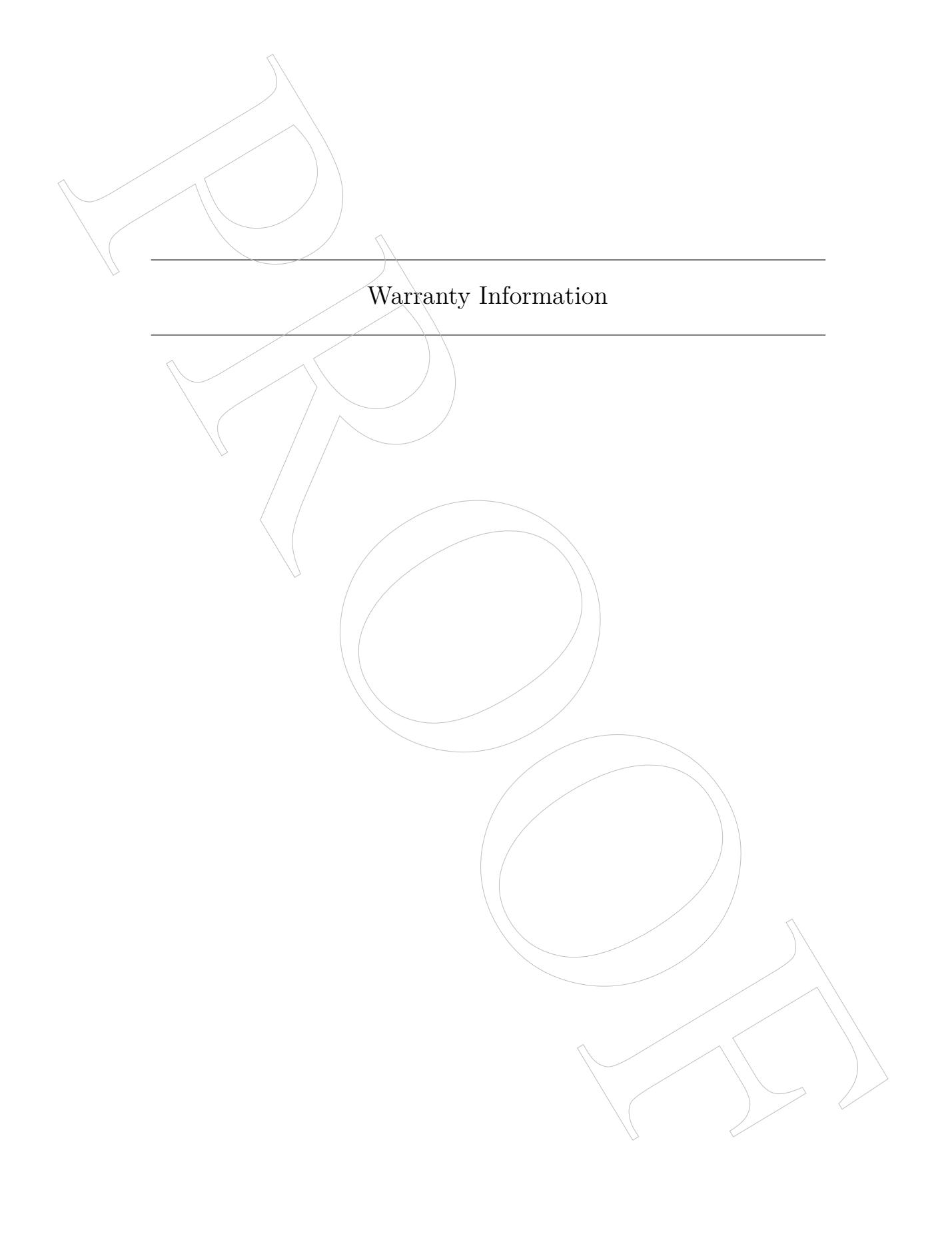
6.3 `Regional Phone Numbers`

The full list of regional phone numbers is available at `LulzBot.com/contact-us` A brief list can be found in this guide in Section 8.3.

6.4 `Community`

Community Support and Resources

- LulzBot 3D printer forum: `Forum.LulzBot.com`

- LulzBot Online Community: `LulzBot.com/community`

Warranty Information

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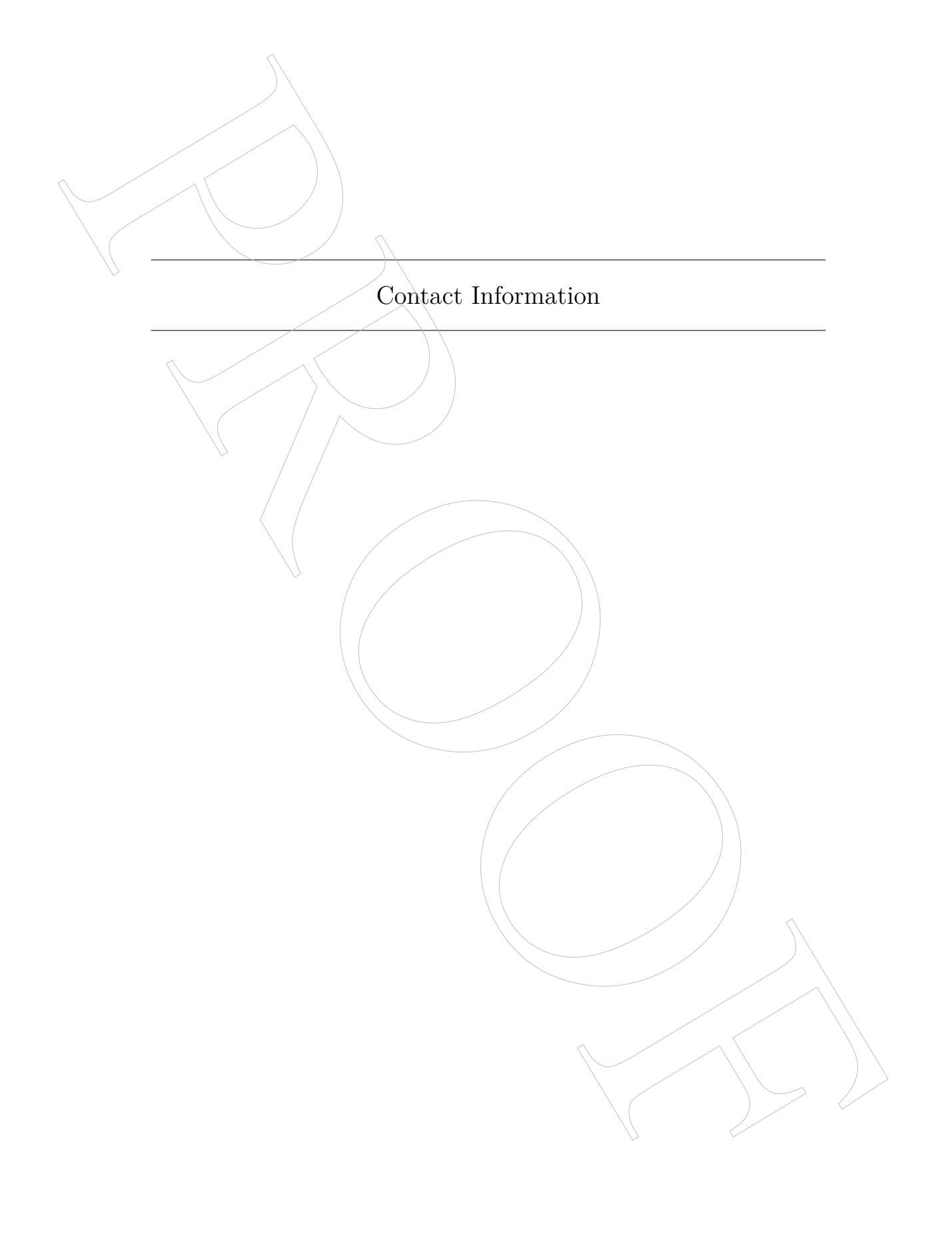
Warranty information for your LulzBot Mini 3D printer can be found at
`https://www.lulzbot.com/support/warranty`

`Extended Warranty`

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

7.2 `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.

Contact Information

8.1 **Support**

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

8.2 **Sales**

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

8.3 **Regional Phone Numbers**

A complete list of regional phone numbers is available at `LulzBot.com/contact-us`.

- Australia (Sydney): +61-280730598

- Canada (Toronto): +1-647-560-0006

- China (Shenzhen): +86-75586243890

- Great Britain (London): +44-2036033166

- Mexico (Mexico City): +52-5541708259

- Netherlands (Amsterdam): +31-208004975

- Spain (Madrid): +34-912901413

- Switzerland (Zurich): +41-445087565

- United States (Loveland): +1-970-377-1111

8.4 **Websites**

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

Index

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 Mini.

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 TAZ 1-4 Budaschnozzle top extrusion temperature to 250°C. The LulzBot Mini 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 Mini can print at a layer heights of 0.05mm to 0.50mm with the standard tool head.

Slic3r Slic3r is a cross-platform 3D model slicing engine used to process a 3 dimensional model into the GCODE (tool path) needed to physically generate the print.

Spool Plastic filament coiled and stored on a plastic reel. 3mm filament is referred 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.

Notes

Use the following pages to record notes and any personal preferences or tweaks you make to your LulzBot 3D printer for future reference.

This page is intentionally left blank

This page is intentionally left blank

This page is intentionally left blank

Colophon

Created with 100% Free/Libre Software
GNU/Linux
LaTeX Memoir

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

LULZBOT®

ALEPH OBJECTS INCORPORATED