[0. Overview: 2](#_Toc190169229)

[1. Prerequisites: 2](#_Toc190169230)

[2. Install Printer Compatible Slicing Software: 2](#_Toc190169231)

[3. Select a Design to Print: 3](#_Toc190169232)

[A. Do It Yourself 3](#_Toc190169233)

[B. Use a free design 3](#_Toc190169234)

[C. Purchase a design for use 3](#_Toc190169235)

[4. Download the Design to a Flashdrive: 4](#_Toc190169236)

[5. Run the slicing software: 4](#_Toc190169237)

[6. Save the .gx code to a USB drive: 4](#_Toc190169238)

[7. Load the Filament. 4](#_Toc190169239)

[8. Preparing the printer bed: 5](#_Toc190169240)

[9. Leveling the Bed 6](#_Toc190169241)

[10. Plug USB flash drive into printer and load .gx code: 7](#_Toc190169242)

[11. Start print: 8](#_Toc190169243)

[12. Monitor print progress: 8](#_Toc190169244)

[13. Print complete, remove from printer: 8](#_Toc190169245)

[14. Print bed cleanup: 8](#_Toc190169246)

[15. Post processing, options for finishing the print: 8](#_Toc190169247)

[Safety 9](#_Toc190169248)

[Appendix A: FlashPrint 5 Basics 12](#_Toc190169249)

[Appendix B: Adventurer 3 Menus 16](#_Toc190169250)

[Main Menu 17](#_Toc190169251)

[Build Menu – Source Select 17](#_Toc190169252)

[Build Menu – FileSelect 17](#_Toc190169253)

[Tools Menu 18](#_Toc190169254)

[Network Sub-Menu 18](#_Toc190169255)

[Preheat Sub-Menu 18](#_Toc190169256)

[Settings Sub-Menu 18](#_Toc190169257)

[Settings – Calibrate Sub-Menu 18](#_Toc190169258)

[About Sub-menu 18](#_Toc190169259)

[Filament Menu 19](#_Toc190169260)

[Load – load filament 19](#_Toc190169261)

[1) heats extruder to accept new filament 19](#_Toc190169262)

[Change – unload filament 19](#_Toc190169263)

[1) heats extruder to release cold filament 19](#_Toc190169264)

[Manual – advance filament 19](#_Toc190169265)

[1) manually advance film by a predetermined length 19](#_Toc190169266)

[Appendix C: Print Beds 19](#_Toc190169267)

[END OF FORMAL DOCUMENT 22](#_Toc190169268)

[START OF NOTES 23](#_Toc190169269)

[**Finishing PLA 3D Printed Parts** 24](#_Toc190169270)

[**PLA 3D Printed Part Priming and Painting** 26](#_Toc190169271)

[**PLA 3D Printed Part Final Results** 26](#_Toc190169272)

[**5-Post Processing with PLA Filament:** 27](#_Toc190169273)

[**How to polish/smoothen the prints with PLA filament?** 28](#_Toc190169274)

[**Conclusion** 28](#_Toc190169275)

# Overview:

This document discussed the creation of 3D printed objects using Fused Deposition Modeling (FDM) which is a 3D printing technique that uses a heated thermoplastic filament laid down in layers by a computer-controlled print head/nozzle and bed moving in 3 axis to create a three-dimensional object.

The document focuses on using Flashforge Adventurer 3 3D printers and PLA filament, however the knowledge and techniques outlined here should be easily transferable to other FDM printers. Non-PLA filaments may require different software settings and/or techniques for post-processing.

|  |
| --- |
| **NOTE: This document DOES NOT address the creation of 3D designs using CAD (Computer Aided Drafting) or other software.** |

# Prerequisites:

1. A computer with Internet access on which software to prepare the design for printing can be installed.
2. Availability of an FDM 3D printer, printer filament (e.g. PLA), and hand tools required to remove the print from the bed and clean the bed. Optional: a glue stick or masking tape to assist in bed adhesion, chemicals such as isopropyl alcohol for use in cleaning the bed.
3. A USB flash drive (aka thumb drive) compatible with both the computer and the 3D printer (USB-A port). Depending on the devices used this may require a USB port adapter.

|  |
| --- |
| **NOTE: Flashforge Adventurer 3 printers will not recognize a flash drive larger than 32GB.** |

1. *Optional:* A utility capable of unpacking .zip or .rar files, e.g. WinZip, 7-Zip, WinRar.

# Install Printer Compatible Slicing Software:

|  |
| --- |
| **NOTE**: **At this time slicing software is not installed on the college (CCM) computers. You must install the slicing software and generate the .gx code file(s) on your own computer.** |

Slicing software is used to convert .stl 3D drawing files to .gx code files which are readable by the printer. Most 3D printer makers will provide or recommend slicing software for use with their printer.

FlashPrint from Flashforge is recommended when printing on Flashforge 3D printers. Versions of FlashPrint are available for Windows, Mac , and Linux operating systems. The FlashPrint software can be downloaded from the Flashforge website <https://flashforge.com/pages/software-flashprint>.

# Select a Design to Print:

## Do It Yourself

You may create an original 3D design or modify an existing free design (ensure you are aware of any copyright restrictions for use imparted by the original designer), This can be done using any 3D modeling software capable of generating files in the .stl format. Examples of such software are;

* Tinker CAD (Free)
* Blender (Free)
* Freecad (Free)
* AutoCAD (Paid)
* Solidworks (Paid)

## Use a free design

Many creators share their designs for free, allowing others to use them as-is, or modify them for their own use. Ensure you are aware of any copyright restrictions for use imparted by the original designer using licensing such as Creative Commons ; (<https://creativecommons.org/share-your-work/cclicenses/>).

Examples of sites where models can be downloaded for free are;

Thingiverse <https://www.thingiverse.com>

Creality World <https://www.crealitycloud.com/> (limited free designs)

MyMiniFactory <https://www.myminifactory.com/> (limited free designs)

Cults 3D <https://cults3d.com/en>

CGTrader <https://www.cgtrader.com/> (limited free designs)

TurboSquid <https://www.turbosquid.com/> (limited free designs)

Free3D <https://free3d.com/> (limited free designs)

Printables <https://www.printables.com/model>

Pinshape <https://pinshape.com>

GrabCAD <https://grabcad.com/library>

Zortrax Library <https://library.zortrax.com/>

## Purchase a design for use

Some creators require payment to use their designs. The ability to modify, re-mix, or use the designs may be governed by individual licenses found on the sites.

Creality World <https://www.crealitycloud.com/> (limited free designs)

MyMiniFactory <https://www.myminifactory.com/> (limited free designs)

CGTrader <https://www.cgtrader.com/> (limited free designs)

YouMagine <https://www.youmagine.com/>

# Download the Design to a Flashdrive:

Download the chosen design and move it to a folder where you can if necessary, unpack it using the appropriate extraction utility (e.g. WinZip, 7-Zip, WinRar). Extract the files to a separate sub-directory.

# Run the slicing software:

If the slicing software is installed double-clicking the chosen .stl file should open the slicing software and display an image of what the file should look like after 3D printing. Alternatively you can open a file using “File”, “Load file” and navigating to the file, then clicking Open. (See Appendix A for recommended changes to default settings when using FlashPrint.)

# Save the .gx code to a USB drive:

|  |
| --- |
| **NOTE: Flashforge Adventurer 3 printers will not recognize a flash drive larger than 32GB.** |

When slicing is finished click on the green down arrow in the top right of the screen and select the appropriate location (flash drive) to store the file. (See Appendix A: FlashPrint Basics)

# Load the Filament.

In order to load the filament the printer must be turned on. The power switch and the power cord connection are on the right side of the printer in the lower back corner. When turned on the display on the front of the printer will be lit up and displaying text and/or graphics.

The most common filament for use in these printers is PLA which serves as a good general purpose filament for 3D printing. Other types of filament can be used however you may need to change the parameters in the slicing software to use these filaments.

Filament is generally provided on reels which can be attached to the printer to ensure proper unspooling, without kinks or tight bends which can cause print failures. The filament is typically pushed into a heated nozzle at a defined rate by gears which grip the film.

Previously used filament which has been unloaded from the printer will have a long string on the end as a result of being previously heated and forced through the extruder nozzle . Prior to loading the filament ensure that the end of the filament is cleanly cut, ideally at a small angle to assist in feeding into the drive gears. Ensure the printer is plugged in and turned on!

Select the filament you intend to use to print the model and prepare the end of the filament by removing any previously heated/extruded material. Gently push the filament into the filament intake on the right side of the printer near the top front corner. The filament intake is just below the yellow label with a black arrow which indicates the filament’s direction of travel.

On the printer’s main menu screen select Filament and then Change if there is existing filament loaded. This will unload the current filament. If no filament is currently loaded, select load. This will cause the extruder nozzle to begin heating and when it reaches the proper temperature the drive gears will begin to turn. Gently push the filament into the intake in order to cause it to engage with the drive gears. When it engages you will feel the filament being pulled into the drive mechanism and you can release the filament. The filament will be pushed into the drive mechanism and through the translucent plastic tube (aka Bowden tube) into the print head. Watch for filament to be extruded from the print nozzle (any filament still in the print head from a previous print will be extruded first) and when the filament color you loaded is extruded press OK on the menu touch display. This will stop the filament loading process.

# Preparing the printer bed:

**Why is bed adhesion important for 3D printing?**

Bed adhesion is an important part of the 3D printing process. A key requirement for a usable print is that the first few layers printed MUST adhere to the print bed. Luckily, there are special products designed to create the perfect amount of bed adhesion. Here are the reasons why bed adhesion is so crucial in 3D printing;

**Not enough adhesion**

If the 3D printed parts are loose enough to move around during the printing process, it can ruin the entire build. 3D printing requires exact measurements, so any sort of movement can misalign the entire build.

**Too much adhesion**

Alternatively, if the build plate has too much adhesion, it can be difficult to remove the plastic components. Any particularly thin pieces can potentially become damaged by the greater force required to remove them from the adhesion.

**Bed is not Level**

In order for propre adhesion to occur the distance between the nozzle’s plain of travel and the bed must remain constant for each layer during the print job. While the bed on these printers cannot be mechanically leveled it is possible to maintain the same distance between the nozzle’s plain of travel and the bed by adjusting the height of the head as it travels using an algorithm built into the latest revision of the printer firmware. The algorithm relies on knowing the position of 9 different points on the bed in order to perform the leveling calculations.

|  |
| --- |
| **Note: Prior to performing the bed leveling procedure ensure the bed is clear and any coatings, such as masking tape, intended to assist in adhesion have been applied. Changing the bed coatings or even removing the bed to take off a print and re-inserting the bed can affect the bed level.** |

In order to calibrate the nozzle height go to ‘Settings”, then “Calibration”. The printer will move the nozzle to the center of the bed and lower it until it is several millimeters off the bed. Using the up and down indicators on the menu lower the head until it just possible to insert a sheet of paper (or the small card in each reel of filament) between the nozzle and the bed. When trying to slide the paper/card between the nozzle and the bed there should be a small amount of resistance. When the adjustment is satisfactory press “next”. You will then be asked “Do you want to continue and complete auto-calibration?”. Press “Yes”.

The head will reposition to the next calibration point and you will repeat the adjustment 8 more times, pressing “ok” when each point is correctly adjusted. After the last point is adjusted press “Complete” and the printer will store the values for each position and return the head/nozzle to its rest point.

**Coating the Bed**

While it does take some set up, the price of the supplies and the quality of the printed parts are unmatched. These techniques essentially “glue” the print to the build surface.

* [Blue painter’s tape](https://tapemanblue.com/products/blue-painters-tape) is a great option for creating bed adhesion. The porous nature of the blue tape allows PLA nylon, and other filaments to stick directly to the back of the tape. Painter’s tape is also heat resistant so it can withstand heated beds and the heat of the filament.
* [Disappearing purple glue stick](https://www.amazon.com/dp/B00ATJLLC8?tag=all3dptrx00071-20) works well because you know you have covered the build plate visually. (In contrast, with clear glue stick it’s sometimes a bit harder to tell if sufficient coverage has been applied, although there are [good options worth considering](https://all3dp.com/2/what-s-the-best-glue-stick-for-3d-printing/).) This can be used with the blue painter’s tape mentioned above to provide a greater likelihood of adhesion.
* [Aqua Net hairspray](https://www.amazon.com/dp/B002K33AFM?tag=all3dptrx00071-20) works quite well due to the lack of interfering additives in it. You can use [other brands of hairspray](https://all3dp.com/2/best-hairspray-for-3d-printing/), but they should have few additives to work well for bed adhesion.
* A salt solution works by enabling a molten filament to better wet the build surface by changing its composition.
* [Wolfbite](https://airwolf3d.com/shop/wolfbite-prevents-3d-printed-parts-from-warping/) is a glass bed adhesive that helps stick your prints to the print bed when they would otherwise warp. Conveniently, there are multiple types of Wolfbite specific to each type of filament.

# Leveling the Bed

The Adventurer 3 printer are equipped with a “bed leveling” function which improves adhesion of the first layer to the bed and print quality. This does not affect the bed itself but uses and algorithm to help position the extruder a consistent distance from the bed even if thebed is not level. It does this by having the user adjust the level at 9 points on the bed, the center of the bed and 8 positions around the perimeter.

**Required Tools:** cardstock labeled “Monoprice 3D Printing Filament Suggested Temperatures” (shipped with each spool of Monoprice filament). These cards are made of cardstock that is 0.2mm thick and are ideal for adjusting the print head height off the bed.

To level the bed;

* go to “Settings” on the Tools Menu
* select Calibration, this will position the extruder head over the center of the bed.
* Adjust the extruder nozzle up/down (using the on-screen arrow buttons) to allow the card stock to slide under the nozzle with some friction
* When adjusted to allow the card stock to slide in and out with some friction press “next” on the control panel.
* When asked “Do you want to continue and complete the auto-calibration?” it is recommended you press “Yes” to calibrate the nozzle height at 8 additional points. Pressing “No” will store the nozzle position at the center of the bed which is not sufficient information to allow the printer to generate an algorithm to simulate a level bed.
* Pressing “yes” will cause the extruder head to move to the left rear corner of the bed where the process of using the card to set the print head can be performed again. When the extruder head is correctly positioned press “next”
* Repeat the above step for the next 7 points around the bed perimeter. When the extruder head is correctly positioned at the last (front-right) position press “ok”, then “Completed”.
* The extruder head will then reposition to its home position. The extruder head height calibration process is now complete.

# Plug USB flash drive into printer and load .gx code:

From the printer front panel select Build (If you do not see the Build option press the back arrow on the bottom of the display (display is touch sensitive) until the menu shows Build. Tools and Filament. This is the “main” menu.

When Build is pressed the display will show storage media on which build code is available and the total storage capacity of each media. The top storage shown is the printer’s internal storage. The symbol below it indicated if a USB flashdrive is plugged in. If a drive is present its total storage capacity will be displayed next to the flashdrive symbol.

To load code from the flashdrive press on the flashdrive symbol. This will open a menu showing files stored on the flashdrive. Follow the screen prompts to find the file to print the pres on the file name to load it. When loaded the expected brint time and the amount of filament required will be shown. Press the Build symbol to start the print.

The printer will then go through a sequence of moves intended to relocate the print head and the bed to a “known” position so that the print will not go over the edges of the bed.

The print head will pause several inches above the bed and go through a preparation process where it heats the bed and the extruder nozzle to the required temperature.

# Start print:

Press the Build symbol to start the print.

The printer will then go through a sequence of moves intended to relocate the print head and the bed to a “known” position so that the print will not go over the edges of the bed.

The print head will pause several inches above the bed and go through a preparation process where it heats the bed and the extruder nozzle to the required temperature.

# Monitor print progress:

It’s a good idea to keep an eye on prints particularly early on in the print process when issues with adhesion will appear.

# Print complete, remove from printer:

Please remove your prints and clean the bed when the printing process is complete.

When the print is complete remove the printer bed by grasping in the center and pulling straight out on the front of the bed. The metal beds have some flexibility and if you grab it in each hand and twist in opposite directions the bed will flex slightly and small prints will release from the bed. For larger prints it may be necessary to slip a paint scraper od some other form of rigid blade under the print edge and scrape to release the rest of the print.

# Print bed cleanup:

As a courtesy to other users of the lab printers please clean the bed after use., using caution not to damage the bed.

Cleaning the bed may be as simple as using the paint scraper to remove any filament still stuck to the bed. If you used a glue stick to create adhesion and the bed feels rough to the touch you can also try scraping to remove glue residue. If it still feels rough after scraping spray the bed surface with isopropyl alcohol and wait a few minutes for the alcohol to begin to break down the residue then scrape again.

# Post processing, options for finishing the print:

FDM printing, particularly when using PLA can result in a rough surface which may be acceptable for the intended use. If you are looking for a more finished, smoother print it is necessary to perform some form of post processing on the print to smooth the surface and fill any gaps. a=Additionally When the print is completed the print quality may be adequate to the intended use

### Safety

There are many methods for post processing a 3D print which can include the use of sharp tools and/or the use of chemicals. SAFETY is always an important factor. When performing postprocessing which includes the use of sharp tools always cut/work away from yourself. When using processes which create dust (e.g. sanding) or require chemicals always wear an appropriate respirator and protective equipment such as goggles and gloves, and work in a well-ventilated area away from sparks or open flames.

Support Removal

3D printing using the FDM method generally builds models by depositing filament on a previously extruded layers however sometimes it is necessary to bridge filament across a gap or opening in the print. Where the gap or opening I fairly short a new layer of filament can be deposited with out the need for supports below the layer to prevent sagging or bowing. Where the gap is larger if may be necessary to prevent some sort of support for the upper layers. As part of post processing these supports need to be removed. This is often done using small wire cutters, in order to remove the material without leaving stubs “flush” cutting pliers, commonly used to remove excess component lead material from through hole printed circuit boards are recommended. Side cutters are the most common style of flush cutters but in some cases end cutters may be preferred.

Sanding

The most common purposes for sanding a model are;

* Remove excess Material from the model
* Create a smooth finish on the model’s surface

Sanding uses abrasives to remove unwanted material and can be performed by hand or with power tools. Hand sanding is often used for delicate models but can be extremely time consuming. Sanding with power tools is much faster but care must be used not to remove required material or to allow heat generated by the sanding process to affect the model.

Hand sanding can be performed using sheets or partial sheets of sandpaper or rigid foam blocks coated in sandpapers. The choice will generally depend on the shape of the finished surface, sandpaper can be formed to the models shape when handheld or backed by flexible foam while a rigid foam block or sandpaper placed on a flat surface can be used to create a flat surface on the model.

Where a significant amount of material needs to be removed an electric sander for flat surfaces or rotary tool (e.g. a Dremel) for contoured surfaces can be used.

Sandpaper Grades

Sandpaper is graded by its grit size. Grit is a rating of the size of the abrasive particles embedded in the sandpaper, the smaller the number the more coarse the sandpaper, the higher the number the finer the sandpaper. Coarse sandpaper removes more material with each pass of the sandpaper over a surface than fine sandpaper does.

Typical git sizes are:

* 40 grit: Coarse
* 80 grit: Medium
* 100 grit: Medium
* 120 grit: Fine
* 220 grit: Fine
* 440 grit: Extra Fine

Sanding Methods

Wet sanding uses sandpaper lubricated with water, resulting in a smoother, finer finish with less dust

Dry sanding uses sandpaper without lubrication, which is faster for initial material removal and shaping, but can produce more dust and leave a less refined surface

Joining

Joining is the process by which separately created parts of a model are permanently attached/bonded to each other to create a single permanent model. Typical bonding methods for 3D printed components are;

1. Gluing uses the adhesiveness of a chemical to stick two or more parts together. Depending on the adhesive used this can create a temporary or a permanent bond. Foe PLA filament cyanoacrylate (CA) glues( also known as super glues), epoxy resins, or acetone can be used to glue PLA components together.
2. Chemical welding occurs through the formation of new chemical bonds between neighboring molecules at the contacting surfaces.[[1]](#footnote-1)
   1. Friction welding involves getting a piece of filament moving at a high speed using a power tool such as a Dremel so that friction is created when it comes into contact with printed parts. The friction creates heat which will cause the moving filament to soften enough so it can be manipulated like hot glue. When cooled this creates a strong bond, as if it had been one piece all along. The result is a fairly strong weld between your printed parts but requires some finishing work to make the joint look nice. Due to its lower working temperature , meaning it requires less friction, PLA is the preferred filament to use for this method.

Using a soldering iron to weld PLA filament together is another method however care must be used  since it risks melting the printed piece or burning the filament. When using a soldering iron to weld PLA it is best to use a tip dedicated to PLA welding. Tips used for other purposed, such as soldering, may leave contaminants in the weld.

A 3D printing pen designed for use with PLA can also be used to fill or join model pieces.

Filler material

Spot putty

Bondo spot putty is a 2 part product commonly used for auto body repair that works well on PLA. When applied in thin coats it dries quickly and is easily sanded but can create large amounts of dust when not applied sparingly. Follow the manufacturer’s usage and safety instructions however mixing in some acetone will provide a more pliable product with a longer working time. Sand, prime and paint to finish.

Wood Putty

Wood putty works well for filling small areas but can be subject to shrinkage and long set times. Apply sparingly. Can be purchased in multiple wood shades (e.g.MinWax wood filler). Follow the manufacturer’s usage and safety instructions. Sand, prime and paint to finish.

Epoxy

XTC-3D

XTX-3D is a 2 part specialty product made for finishing 3D prints. When combined it can be brushed onto any 3D print. The coating self-levels and wets out uniformly without leaving brush strokes, however it is subject to pooling in depressions in the model. Working time is 10 minutes and cure time is about 4 hours. Colors and metal effects can be added at time of mixing, Cures to a hard, impact resistant coating that can be sanded, primed and painted.

Primer

Painting

## Appendix A: FlashPrint Basics

Recommended Changes to Settings

The FlashPrint 5 software has many configuration settings available but there are 3 key settings that should be confirmed prior to printing on the Adventurer 3 printer.

A screenshot of a computer

Description automatically generated

1. Under the Printer tab confirm that Machine Type is set to “Adventurer 3”.
2. Ensure the Nozzle Size is set appropriately for the nozzle in your printer. NOTE: Adventurer 3D printers installed in the Information Technologies Lab (EH 209) are a mix of 0.3mm and 0.4mm nozzles.

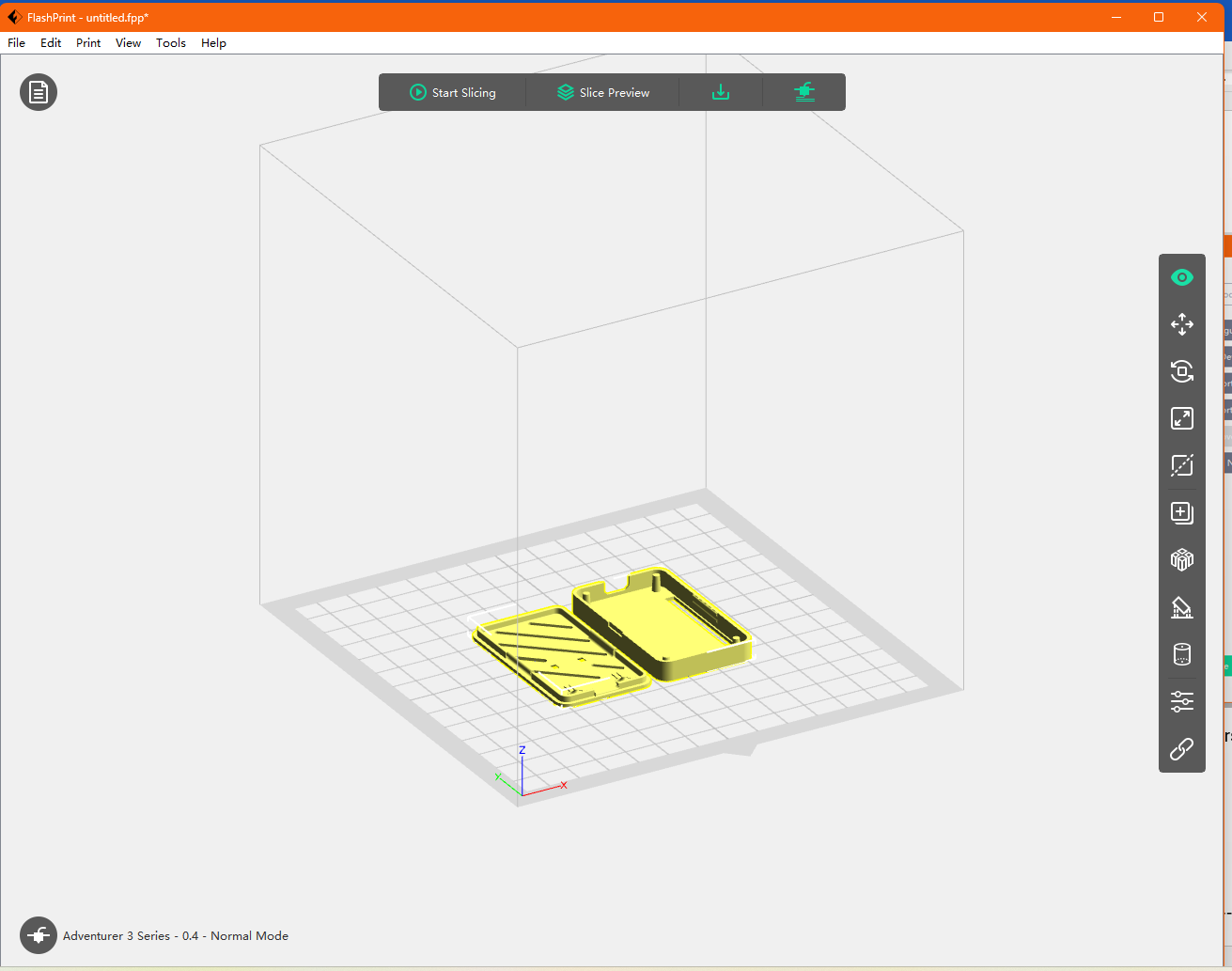
A screenshot of a computer

Description automatically generated

1. Since #D printers commonly encounter minor errors with adhesion in the first few layers it is recommended that “Enable Raft on the Raft tab be set to “Yes”.

Once the recommended changes are made you can select “Slice”.

When slicing is complete;



Click on the green down arrow in the dark bar at the top of the screen and the File Save dialog box will open.

A screenshot of a computer

Description automatically generated

Ensure “Save as type” is set to \*.gx, ensure the file path is set to your USB flashdrive and press Save.

Your printable file is now saved to the USB drive.

## Appendix B: Adventurer 3 Menus

|  |  |  |
| --- | --- | --- |
| Main Menu A screen shot of a phone  AI-generated content may be incorrect. |  |  |
| Build Menu – Source Select Local Storage or USB Thumbdrive | Build Menu – FileSelect Fie list | Build Menu – Copy /Start |
|  |  |  | |
| Tools Menu |  |  |
| Network Sub-Menu |  |  |
| Preheat Sub-Menu |  |  |
| Settings Sub-Menu |  |  |
| Settings – Calibrate Sub-Menu Adjust extruder height |  |  |
| About Sub-menu |  |  |
| Filament Menu | Load – load filamentheats extruder to accept new filament  1. advances filament through bowden tube and extruder  Change – unload filamentheats extruder to release cold filament  1. retracts filament from extruder and Bowden tube  Manual – advance filamentmanually advance film by a predetermined length |  |
|  |  |  |
|  |  |  |

## Appendix C: Dealing With Nozzle Clogs

The most common cause of a full (no material can be extruded through the nozzle) or partial

## Appendix D: Print Beds

Source: [3D Printer Bed: How to Choose Your Build Plate/Surface | All3DP](https://all3dp.com/2/3d-printer-bed-how-to-choose-the-right-build-plate/)

Print beds come in several different materials designed to help filament adhere to, and at the proper time release from, the print bed.

Spring steel smooth PEI (Polyetherimide)

Spring steel smooth PEI is a more recently popular build plate, which has been featured on stock versions of the [Prusa i3 MK3S+](https://all3dp.com/1/original-prusa-i3-mk3s-plus-review-3d-printer-specs/). This type of build plate is coated in PEI and primarily made of spring steel, which is a flexible steel material that’s also ferromagnetic, allowing you to easily remove and reattach the bed to a printer with a magnetic bed surface. The flexible aspect of the bed comes in handy for removing prints because you can bend the surface to pop prints off and the steel will bend back.

While spring steel is a great base material for a build plate, it’s not as valuable as an adhesive layer, which is why the top surface layer of this type of build plate is a smooth sheet of Polyetherimide. Commonly known by the acronym PEI, this thermoplastic material is well-known for providing excellent adhesion for many 3D printing filaments without extra adhesive on top.

The smooth variation of PEI yields a shiny finish on the bottom layer of prints, but it can also make removing the part difficult (if prints stick too well to the smooth area). Also, the smoothness of the bed means you have to fine-tune your [Z offset](https://all3dp.com/2/z-offset-3d-printing-how-to-adjust-it/) to ensure that the first layer comes out well.

Smooth PEI provides excellent adhesion, especially for parts that require a small contact point to the build surface. This makes it compatible with a wide range of materials, including PLA, PETG, TPU, ABS, ASA, and some engineering-grade materials. Similarly to glass build plates, you’ll want to make sure you have a protective layer of adhesive applied before printing PETG or TPU, as these materials can [stick too well](https://all3dp.com/2/how-to-remove-3d-print-from-bed-3d-print-stuck-to-bed/) to this surface and potentially damage it without a release layer added.

* **Features:** Flexible for easy part removal, magnetic for quick bed attachment, great adhesion for most materials, yields shiny finish for the bottom of prints
* **Compatible materials:** PLA, ABS, PETG, TPU, nylon
* **Examples:** Prusa, [Creality](https://www.amazon.com/dp/B092ZVTJSK?tag=all3dptrx00080-20" \t "_blank), [Fulament](https://www.amazon.com/dp/B07SH1S2C1?tag=all3dptrx00080-20" \t "_blank)
* **Price range:** $25-$50

 S[pring steel textured PEI build plates](https://all3dp.com/2/prusa-pei-sheet-textured-powder-coated-steel/) also use a spring steel base, meaning this bed is flexible and magnetic for easy part removal and bed attachment. However, this build plate option differs from the previous one because it uses a textured PEI surface rather than a smooth PEI coating.

A textured PEI surface means that there are small hills and valleys across the surface. As you might expect, the variations across the bed are reflected on the bottom of printed models, but this type of bed offers a few important benefits.

Mainly, textured PEI beds make part removal easier because prints will stick to less surface area than on a smooth PEI sheet, and the dips and highs of the bed will force prints to pop off once the bed cools. You also have more wiggle room when it comes to adjusting the Z offset because not every point of the bed is the same height. While it’s a little more expensive than other types of build plates such as glass, spring steel textured PEI beds have many features that could justify your purchase.

Textured PEI is very similar to smooth PEI in its material compatibility, while purposely reducing adhesion with its textured surface for a straightforward release of printed parts. You’ll find similar performance with PLA, PETG, and ABS, ASA, while TPU should be a little easier to handle as it won’t stick to the build surface quite as excessively.

* **Features:** Flexible for easy part removal, magnetic for easy bed attachment, great adhesion for most materials, more wiggle room for Z offset
* **Compatible materials:** PLA, ABS, PETG, TPU, nylon
* **Examples:** Prusa, [Geeetech](https://www.amazon.com/dp/B09MKLKPGD?tag=all3dptrx00080-20" \t "_blank)
* **Price range:** $20-$40

Glass

Glass, the stock type of build plate on many budget 3D printers, like the [Ender 3 V2](https://all3dp.com/1/creality-ender-3-v2-review-3d-printer-specs/), and a very low-cost option. You can’t just take off your room’s windows and print on them, though. Most glass 3D printing surfaces are actually tempered glass, including Carborundum coating, which is a strengthened and more heat-resistant version of regular glass that prevents the sheet from deforming during printing.

Glass build plates usually can come in two styles: smooth or textured. The former yields a smoother bottom layer on prints, but textured glass beds, due to their nuanced rises and dips, make part removal easier because prints will naturally pop off when the bed cools.

Glass printer beds are known for their excellent adhesion – when properly cleaned. They’re a great choice for just about any standard material, with good adhesion for PLA, PETG, TPU, ABS, ASA, and even nylons. When printing with PETG, you’ll need to create a protective layer between the print and your print bed to prevent the material from damaging the glass and destroying the build plate. Glue stick, hairspray, or similar adhesive aids are recommended.

You’ll also have to use bed clips to secure your glass bed to your printer since it’s not magnetic or not otherwise pre-glued. Nonetheless, the low price point, wide availability, and decent adhesive properties of glass beds make this print bed a great option.

* **Features:**Low cost, parts pop off when the bed cools, works great for PLA, produces smooth bottom finish on prints
* **Compatible materials:** PLA, ABS, PETG, TPU, nylon
* **Examples:** Creality, [MatterHackers](https://www.matterhackers.com/store/c/glass-build-plates?aff=7383" \t "_blank)
* **Price range:** $20-$50

# END OF FORMAL DOCUMENT

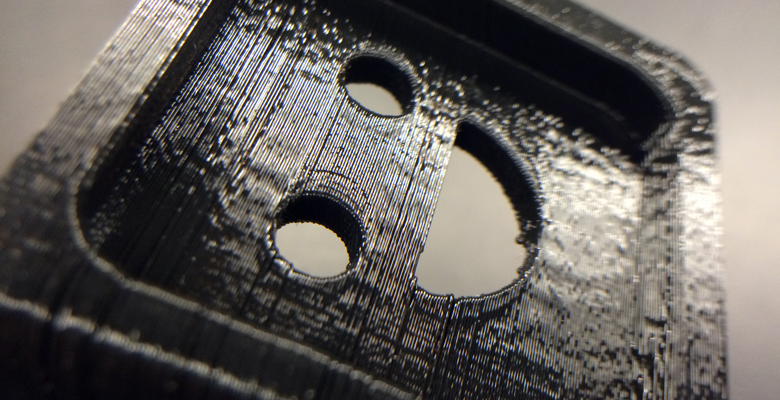
----------------------------------------------------------------------------------------------------------------------------

# START OF NOTES

[How to Finish 3D Printed Parts - Get the Ultimate Guide to 3D printing post-processing, including how to smooth, sand, polish, clean and paint ABS, PLA, and VeroBlack 3D printed parts.](https://www.fictiv.com/articles/how-to-finish-3d-printed-parts#repairing-the-incomplete-layer)

**PLA (Printed on a 5th gen. Replicator)**

Lastly, we have a print of PLA, a notoriously difficult material to finish. However, with some tricks and patience, this print can join the ABS and VeroBlack prints in all their matte finish glory!



Our PLA Replicator print came off the plate with severe striations: see the parallel grooves in the layers of the part above, which could be a result of printer quality. Regardless, we can still smooth out those grooves.

Sanding PLA is difficult, in part because of how soft and gummy it becomes if you sand too aggressively, so we need options for smoothing with minimal effort and abrasion.

**PLA 3D Printed Part Sanding**

If you choose to sand PLA directly, the process is straightforward. Begin with a low (100 – 200) grit sandpaper, sanding away at the bumpy striations and any raft or support material left behind. Particularly in the case of MakerBot supports, it’s easiest to remove them with a flush cutter or pair of pliers and brush them with a rotary multi-tool first before sanding.

Depending on the size and geometry of your part, you’ll likely be stuck in the 100 – 300 grit sandpaper range for a while to smooth out striations and support structure remnants.

Once layering and striations are less prevalent, move through finer grits (400 – 600) sandpaper to achieve a shiny surface ready for priming and painting.

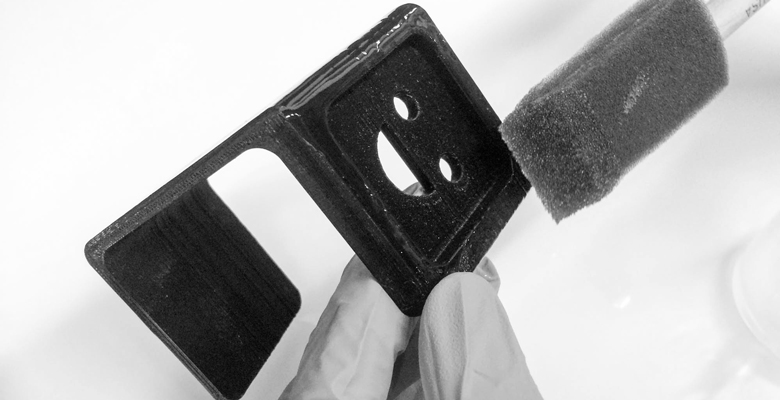
*Pro-Tip: Patience is key with any process you use to finish 3D prints, especially when sanding PLA. Turn on a movie or your favorite show, but don’t zone out! Sand in small circles evenly across the surface of the part. If you’re using a sanding multi tool on PLA, be careful not to*[*overheat/melt your 3D printed part*](https://www.makerbot.com/professional/post-processing/sanding/)*.*

Note: PLA acetone smoothing is not advisable. PLA smoothing is not as forgiving as ABS when it comes to sanding and abrasion, so you will likely spend more time removing the stepping between layers, especially with the severe striations in a print like ours.

An alternative method to sanding PLA prints directly is smoothing the PLA print with the XTC-3D first and then sanding the coating.

**Finishing PLA 3D Printed Parts**

Before applying XTC-3D, ensure that your part is thoroughly clean, free of any oils and sanding dust — are you sensing a pattern yet? Wash the part with soap and dry with compressed air to clear any dust, and wear gloves to protect both your part and your hands.



Brush on the first coat of XTC-3D in a thin (1/64”) coat, which will level itself out. It may be difficult to coat an entire part at once, so you can do it in sections as long as you keep overlapping to a minimum between sections. Wait 90 minutes between first and second coats and after you’ve finished, wait 2 hours to allow the shiny surface to become tack-free.

With striations as severe as ours, it will take multiple coats to get a smooth uniform surface, so patience is key. Remember that multiple thin glossy finish coats will level better than thick coats.

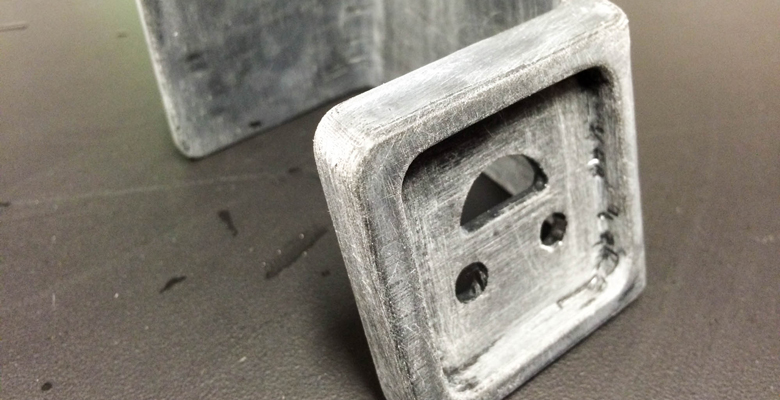
*Pro Tip: Internal pockets are susceptible to pooling, so be careful to remove any pooling that occurs before the XTC-3D begins to dry.*

Beware that XTC-3D may compromise critical dimensions, but applying thin coats will minimize the effects of the additional material.



After the XTC-3D layer has hardened for two hours after application of the last coat, you’re ready for sanding. Start with 300 grit sandpaper and sand in small circular movements to even the surface — wavy patterns may require a coarser grit sandpaper to remove. Focus on getting a level surface, moving through to 600 grit sandpaper.

After you’ve finished sanding, thoroughly clean your part with soap and water and dry it using compressed air before painting.‍



**PLA 3D Printed Part Priming and Painting**

Painting PLA follows the same process as ABS and VeroBlack parts: prime, dry, paint, dry, varnish, dry.

Again, standard spray-painting principles apply:

1. Make sure your surface is oil-free, dust-free and hole-free
2. Shake each paint can for at least two minutes prior to painting
3. Ensure the spray cap is clean to prevent drips
4. Pay attention to how the paint is accumulating on the part and look out for pooling and dripping
5. Paint in many light coats rather than fewer heavy coats — this is critical for 3D printed parts with internal and obscured geometries
6. Only paint in controlled, well-ventilated, and well-lit areas

**PLA 3D Printed Part Final Results**



The XTC-3D coated PLA part is matte, black, and has a smooth surface to the touch — but there are still some problems. Although the XTC-3D has worked well to smooth the part and make sanding quicker and easier, it pooled in the internal pocket and is showing through the paint finish. Also, striations are still visible on the part — another 30 minutes of sanding could have prevented these from showing through the final paint finish.

**5-Post Processing with PLA Filament:**

For the PLA filament 3D prints, hand polishing done with the help of THF or Tetrahydrofuran is the best option. While the acetone option works perfectly with ABS filament, the THF works amazingly with the PLA filament-based prints.

**You will need:**

* Non-Latex Gloves
* THF or Tetrahydrofuran
* A non-dyed, lint-free polishing cloth

**How to polish/smoothen the prints with PLA filament?**

* Remove the excess material from the 3D print created from PLA filament
* Put on the non-latex hand gloves & bring the polishing cloth
* Be in an area that is well-ventilated
* Dip the polishing cloth in the THF & polish the print just as you would a shoe
* Use a circular motion for the best results
* Let the print sit at a place and allow the extra THF to evaporate
* Put up the final smoothened piece in an area of your choice

**Conclusion**

Whether you choose the ABS filament or the PLA option, both come with their own set of pros and cons. However, if you consider the factors such as biodegradability, printed prototypes, material properties, toxicity, and so on, you can surely land on the perfect 3D print. Use it for commercial or personal use; these materials surely help you acquire the best and smooth 3D print. Looking for the best quality 3D printing filaments for your commercial or crafting needs? Try out the range of filaments provided at MakeShaper. All our filaments have been affordably priced without any quality compromise. So, scroll through our collection and purchase your filament today!

1. [Chemical welding of polymer networks - ScienceDirect](https://www.sciencedirect.com/science/article/abs/pii/S2468519422000325#:~:text=Chemical%20welding%20as%20an%20attractive,molecules%20at%20the%20contacting%20surfaces.) [↑](#footnote-ref-1)