Rocky Mountain
Ham Radio

3D Printing

And Amateur Radio

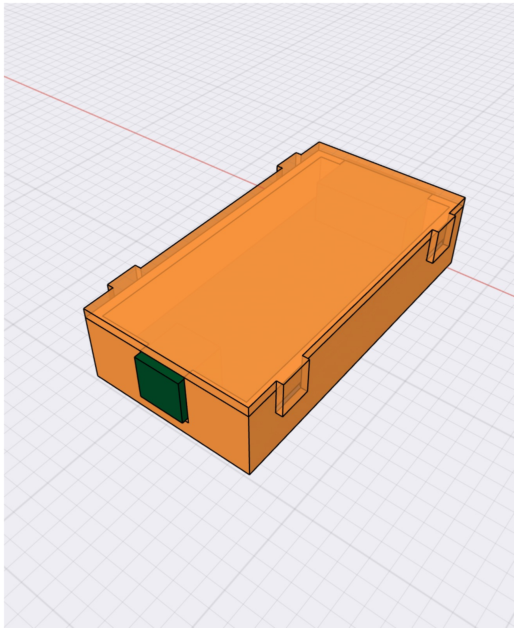
James Cizek – KI0KN

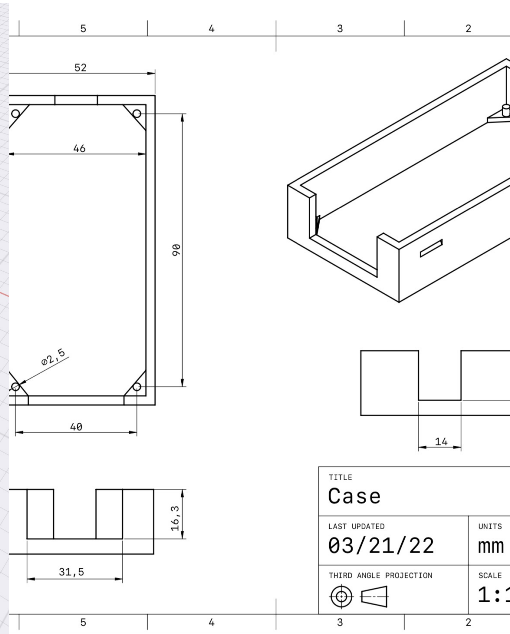
Jim Dixon – KA6ETE

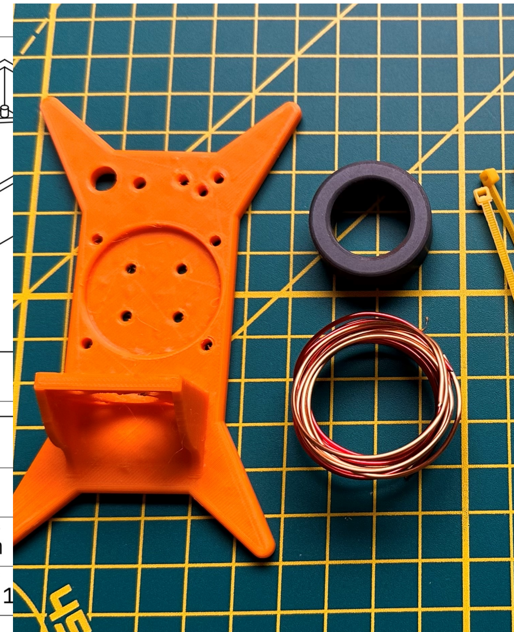
Agenda

- What is 3D Printing?
- Most Common Types of 3D Printing
- Process for 3D Printing/Modeling
- Design or Find a print Model
- Common Types of FDM Filament
- Slicing Your 3D Model for Printing
- Main Parts of an FDM Printer
- Preparing to Print with your FDM Printer
- Resin Printing (SLA)
- Common Types of Liquid Resin
- Resin Printing Steps
- Troubleshooting

Rocky Mountain
Ham Radio







Introduction

- What is 3D Printing?

3D printing, also known as additive manufacturing, is a fast-growing technology transforming how we create objects. This technology also allows us to quickly prototype designs for use in amateur radio as well as make items for long term use.

Most Common Types of 3D Printing

FDM - Fused Deposition Modeling (FDM) is a popular and widely used 3D printing technology. In FDM printing, a thermoplastic filament is melted and extruded layer by layer to create a three-dimensional object. The printer follows a precise digital design, depositing the melted material in a controlled manner, allowing it to cool and solidify.

SLA (Resin) - Stereolithography (SLA) printing is a 3D printing technology that utilizes a liquid photopolymer resin to build objects layer by layer. In SLA printing, a UV laser selectively solidifies the liquid resin according to a precise digital model. The build platform is submerged in the resin, and as each layer solidifies, the platform moves, or the build area changes to accommodate the next layer.

Process for 3D Printing/Modeling

1. Designing the 3D Model:
Create a digital 3D model of the object using computer-aided design (CAD) software. This model defines the geometry and structure of the intended 3D-printed object.

2. File Preparation:
The 3D model is exported as a standard file format (such as STL) that the 3D printer can interpret. This file contains information about the object's geometry in a way that the printer can understand.

3. Slicing the Model:
Use slicing software to break down the 3D model into individual layers (slices). Each layer represents a cross-section of the object. The slicing software generates a set of instructions for the 3D printer on how to build each layer.

4. Setting up the 3D Printer:
Load the 3D printer with the appropriate material (filament, resin, powder, etc.) based on the technology being used. Calibrate the printer to ensure accurate printing and set parameters such as layer height, printing temperature, and print speed.

5. Printing the Object:
Start the 3D printing process. The printer follows the instructions from the sliced file, building the object layer by layer. The specific method varies depending on the technology; for example, in FDM, the printer extrudes melted filament, while in SLA, it cures liquid resin with UV light.

6. Post-Processing:
After the printing is complete, some 3D-printed objects may require post-processing. This can include removing support structures, cleaning, and curing. Post-processing steps depend on the 3D printing technology and the material used.

7. Inspection and Testing:
Examine the finished object for quality, accuracy, and structural integrity. Depending on the application, additional testing or finishing processes may be necessary.

8. Finished Product:
Once the 3D printing process is complete and any post-processing steps are finished, the final 3D-printed object is ready for use, whether it's a prototype, end-use part, or another application.

Design or Find a 3D Print Model

Regardless if you find or design your own 3D model, you will also have to decide which type of file to use. The two most common and widely used are .stl and .3mf as described below:

STL Files:

An STL (Stereolithography or Standard Tessellation Language) file is a widely used file format for representing three-dimensional (3D) models. It is commonly employed in computer-aided design (CAD) and 3D printing applications. The STL file format represents a 3D object as a collection of triangular facets that make up the surface geometry of the model.

1. Faceted Representation: An STL file describes a 3D object by breaking its surface into a series of small, flat triangles. Each triangle is defined by its vertices in 3D space.

2. Mesh Structure: The triangles in an STL file form a mesh structure, which is a network of connected triangles. The mesh represents the surface of the 3D object.

3. Binary and ASCII Formats: STL files come in two formats—binary (binary STL) and human-readable text (ASCII STL). Binary STL files are more compact and faster to load, while ASCII STL files are easier to read and can be edited manually.

4. No Color or Texture Information: STL files focus on the geometry of the 3D model and do not include information about color, texture, or other visual attributes. They are primarily used for defining the shape and structure of objects.

5. Widespread Compatibility: STL is a widely supported file format and is compatible with various 3D modeling software and 3D printers. It has become a standard for exchanging 3D models in the context of 3D printing.

6. Units of Measurement: STL files typically do not specify the units of measurement used for the model. It is essential to know or set the correct units when importing an STL file into 3D modeling or 3D printing software to ensure accurate scaling.

Design or Find a 3D Print Model

Regardless if you find or design your own 3D model, you will also have to decide which type of file to use. The two most common and widely used are .stl and .3mf as described below:

3MF File:

A 3MF file, also known as a 3D Manufacturing Format file, is a modern and versatile file format for 3D printing and additive manufacturing. Developed by the 3MF Consortium, which includes various companies in the 3D printing industry, the 3MF file format aims to address some limitations of older file formats like STL (Stereolithography).

1. Comprehensive Data: Unlike the STL file format, 3MF files can store a more comprehensive set of data beyond simple geometry. This includes information about color, textures, materials, and other properties of the 3D model.

2. Support for Multiple Objects: 3MF files can contain multiple objects and components within a single file. This makes it easier to manage complex assemblies and multi-part models.

3. Enhanced Geometry Representation: 3MF supports more advanced geometry representation, allowing for smoother curves and surfaces compared to the faceted representation of STL files.

4. Units of Measurement: 3MF files can include information about the units of measurement used in the model, which helps ensure accurate scaling when importing the file into different software applications.

5. XML Structure: The 3MF file format uses XML (eXtensible Markup Language) as its underlying structure. This makes it human-readable and allows for easy editing and customization of the file.

6. Compression: 3MF files support compression, which can reduce file sizes and make them more efficient to store and transfer.

7. Open Standard: The 3MF format is an open standard, and its development involves collaboration from various industry stakeholders. This openness promotes interoperability between different software and hardware solutions in the 3D printing ecosystem.

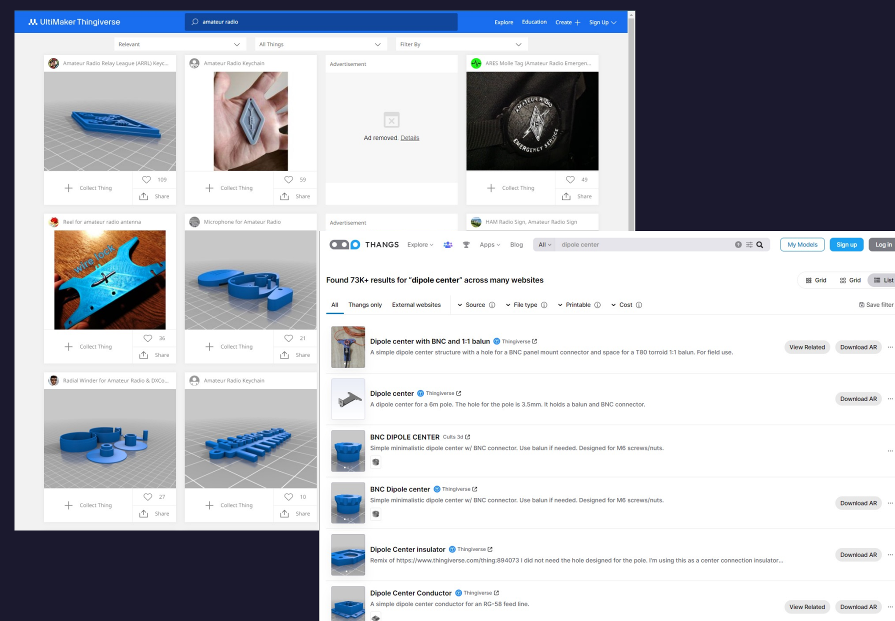
8. Backward Compatibility: While 3MF is designed to offer advanced features, it also maintains backward compatibility with simpler 3D printers and slicers that may not utilize all of its capabilities.

Design or Find a 3D Print Model

This is something that everyone struggles with, but it is often a good choice to do a quick search if the part/item you are searching for has the probability of being a common item.

Popular sites to search for 3D models:

- thingiverse.com (Free)
- thangs.com (Free)
- printables.com (Free & Paid)
- cults3d.com (Free & Paid)
- stlflix.com (Paid Subscription)

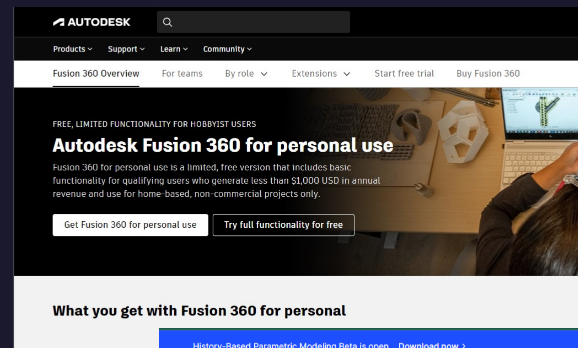


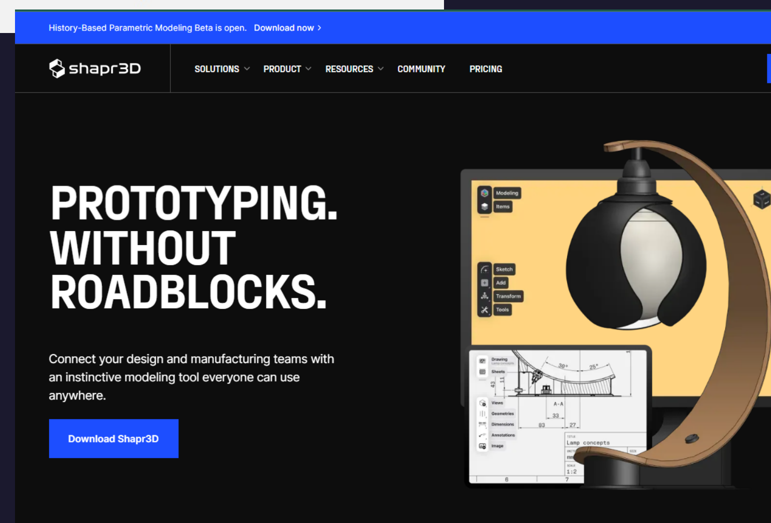
Design or Find a 3D Print Model

Popular 3D Modeling Software

- Autodesk Fusion 360 (Free & Subscription)
- Shapr3D (Free & Subscription)
- TinkerCAD (Free)
- FreeCAD (Free)

While free software can be a good place to start learning occasionally, it may not be your best option. This is due to free 3D modeling programs are often time limited in their capabilities and only offer the verry basics in design capabilities.





Common FDM Filament Types

PLA
(Polylactic Acid)

- PLA is a biodegradable and environmentally friendly thermoplastic.
- It's easy to print with, has low warping, and produces minimal fumes.
- PLA is commonly used for prototypes, hobbyist projects, and educational purposes.
- Glass Transition Temp: 55c-60c (130-140f)
- Print Head Temp: 190c-210c
- Bed Surface Temp: 0-40c

PETG
(Polyethylene Terephthalate Glycol)

- PETG offers a balance between the ease of printing like PLA and the strength of ABS.
- It is durable, chemically resistant, and has good layer adhesion.
- PETG is often used for mechanical parts, containers, and outdoor applications.
- Glass Transition Temp: 75-80c
- Print Head Temp: 230-250c
- Bed Surface Temp: 60-80c

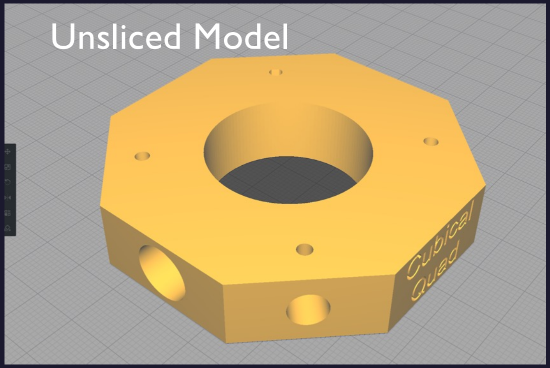
ABS
(Acrylonitrile Butadiene Styrene)

- ABS is known for its durability and impact resistance.
- It has a higher melting point than PLA and can withstand higher temperatures.
- ABS is commonly used for functional parts, automotive components, and consumer goods.
- Glass Transition Temp: 95-105c
- Print Head Temp: 220-250c
- Bed Surface Temp: 90-110c

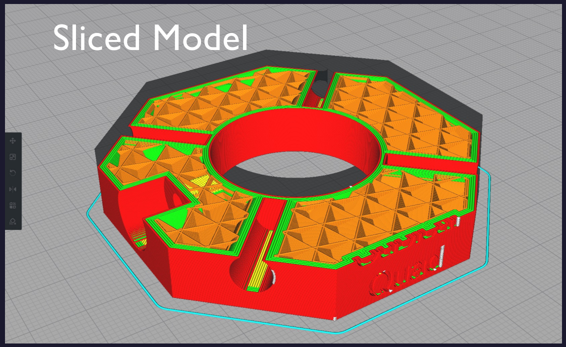
ASA
(Acrylonitrile Styrene Acrylate)

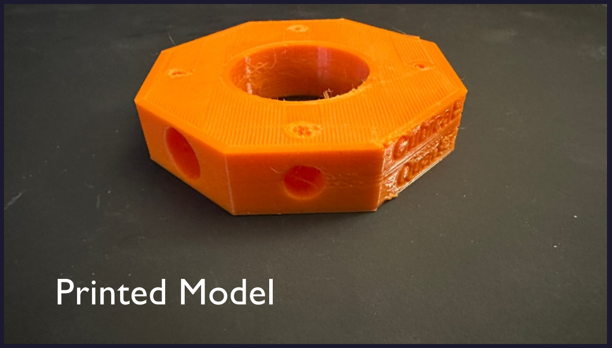
- ASA is similar to ABS but has improved UV resistance and weatherability.
- It is suitable for outdoor applications where exposure to sunlight and weather is a concern.
- Glass Transition Temp: 95-105c
- Print Head Temp: 240-260c
- Bed Surface Temp: 80-110c

Slicing Your 3D File



Once you find or create the 3D model you will then need to slice the object for printing. Slicing is just how it sounds; it takes the solid object and converts it into layers and converts it to g-code which the printer can read and understand.





Slicing Your 3D File

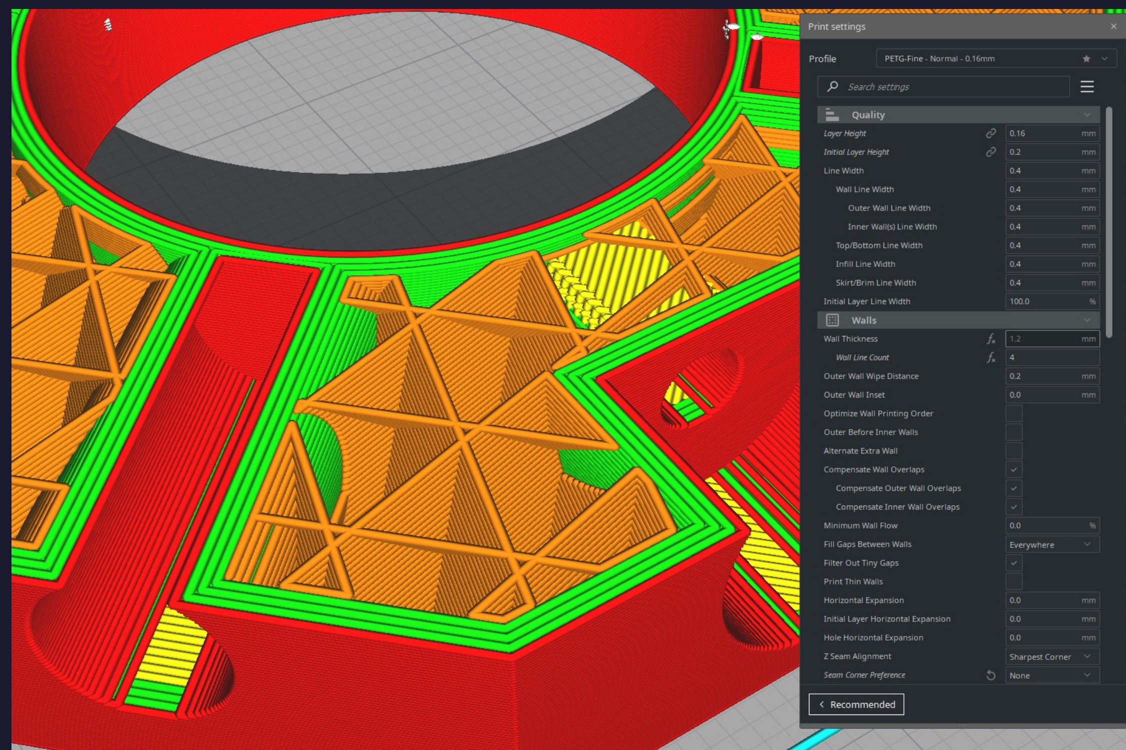
Slicing software is generally free and opensource with only a few paid and/or proprietary iterations out there. During the slicing is where you will set the layer line height (resolution), temperatures of the nozzle and bed to fit the requirements of the filament type and many other parameters.

Popular Slicing Programs

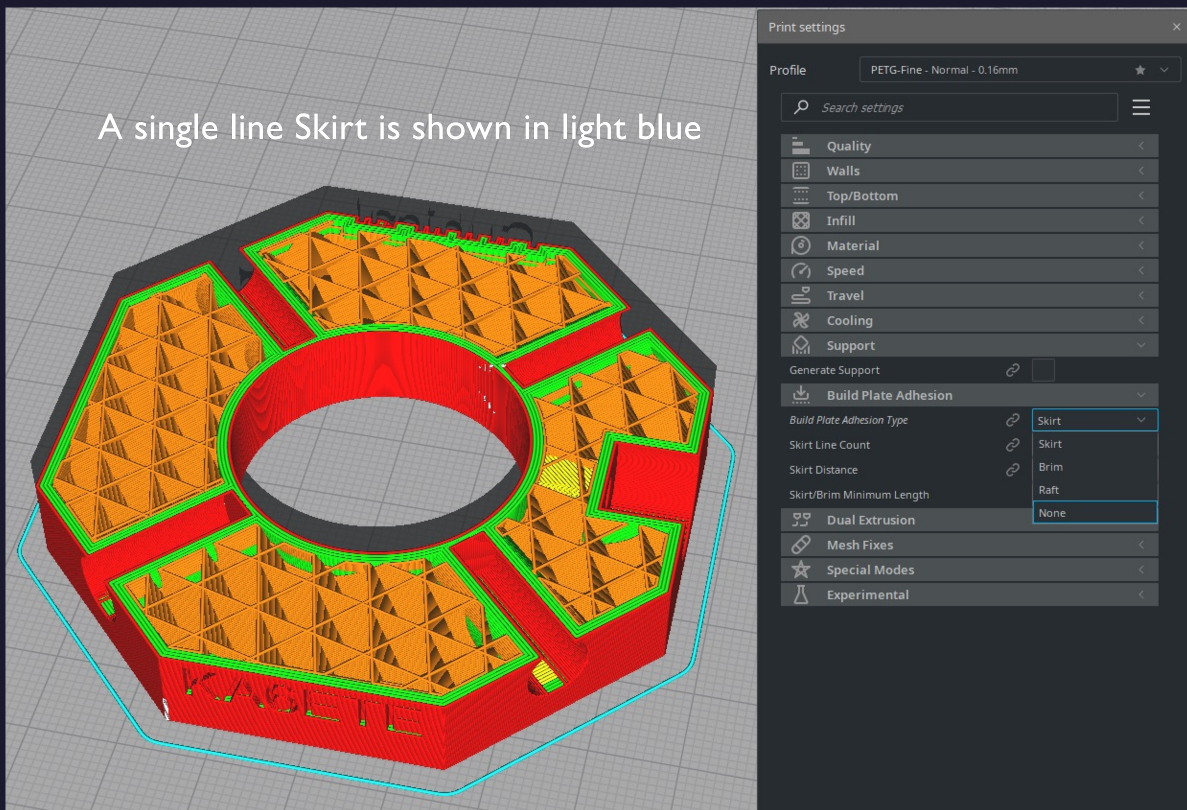
- Ultimaker Cura
- Prusaslicer
- Simplify3D
- Superslicer
- Chitubox (Resin)
- Lychee slicer (Resin)

Slicing Your 3D File

Sample of the slicer is taken from Ultimaker Cura and shows some of the settings that will need to be made. A good baseline to start from is to select the default setting for the material you are using from the program itself and adjust to your needs and machine. This can only be done with trial and error as EVERY printer will be slightly different, even if they are the same model.



Slicing Your 3D File

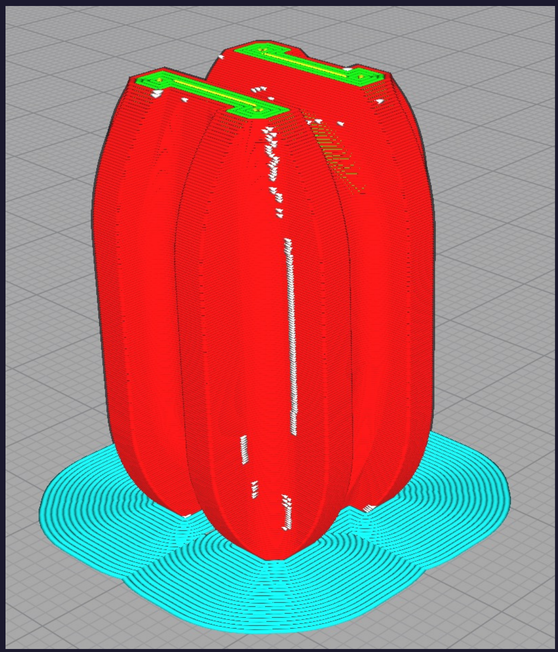


Another important setting to use is "Bed Adhesion", these consist of; None, Skirt, Brim and Raft.

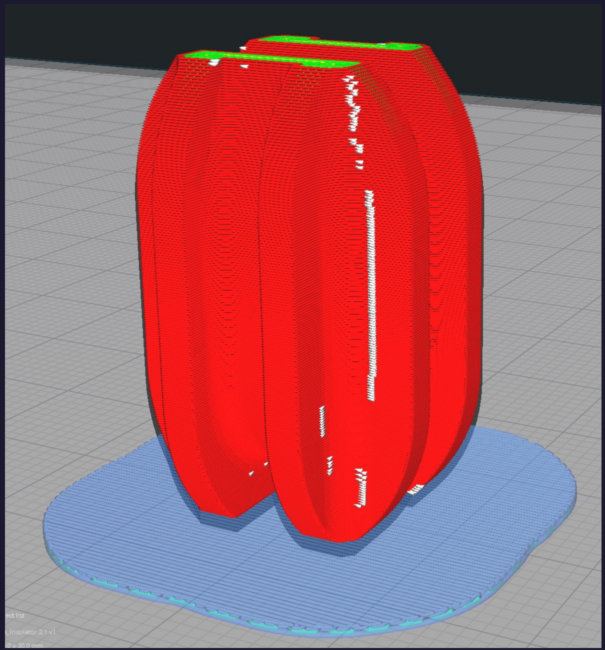
- Skirt – A skirt is a simple line around the object. This is useful for larger objects as it aids in priming the nozzle to get rid of any air that may have collected in the extruder.
- Brim – A Brim can help with slightly smaller items as it will increase the base surface layer to adhere to the bed.
- Raft – A raft is a multilayer subsurface that is printed for small parts that may otherwise become dislodged from the bed surface.

Slicing Your 3D File

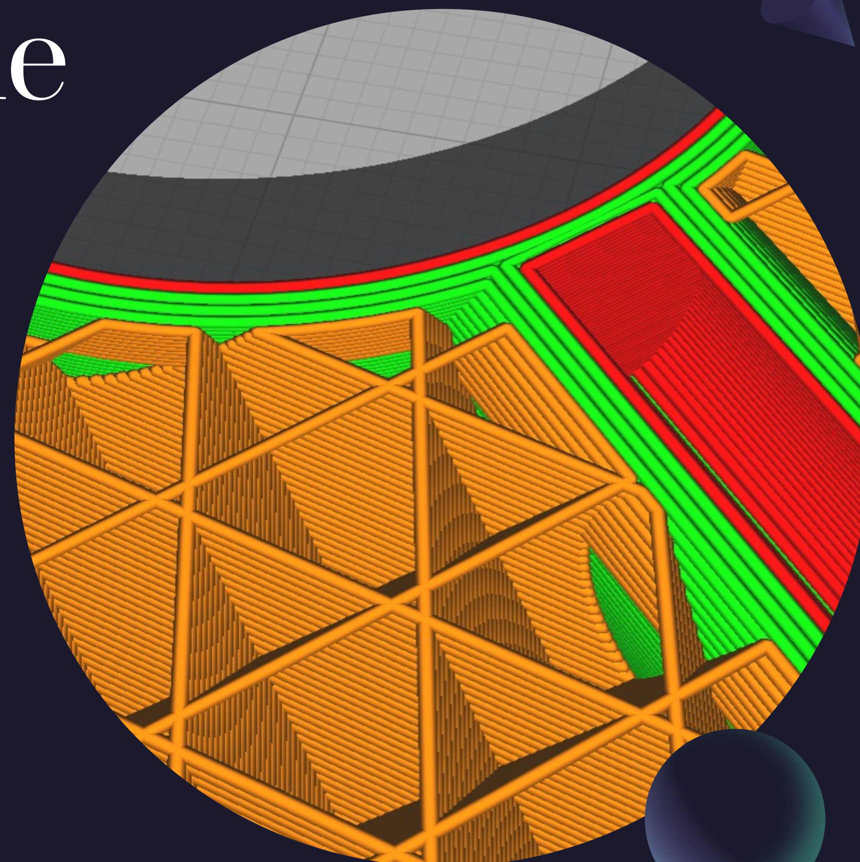
Brim



Raft



Slicing Your 3D File



Slicing Tip:

Contrary to most thinking adding more infill density is not always the best way to add strength to your finished product. I have found that adding one or two additional wall lines significantly increases the overall strength while using less filament and time to print.

Adding an extra wall line can also get rid of any small void that the slicer would try to fill in with a small 1mm or less infill pattern.

FDM Printing

Main parts of an FDM printer

Print Head/Extruder:

Single Extruder:

Description: The most basic type, featuring a single nozzle for depositing one type of filament at a time. It is commonly used for simple prints with a single material.

Dual Extruder:

Description: Equipped with two nozzles, allowing the printer to use two different filament types or colors during a single print job. This enables the creation of multi-material or multi-colored objects.

Bowden Extruder:

Description: In this design, the extruder motor is located remotely from the hotend and is connected by a Bowden tube. This configuration reduces the moving mass on the print head, enabling faster movements and potentially higher printing speeds.

Direct Drive Extruder:

Description: The extruder motor is mounted directly on the print head, and it feeds the filament directly into the hotend. This design is advantageous for flexible filaments and provides more precise control over filament extrusion.

Hot End/Nozzle:

The hotend is a critical component of an FDM 3D printer responsible for heating and melting the filament. It consists of a nozzle through which the molten filament is extruded onto the build platform. The hotend typically has a heating element (such as a resistor or cartridge heater) and a temperature sensor (usually a thermistor or thermocouple) to control and monitor the temperature.

1. **Functionality:** The filament is fed into the hotend, where it undergoes controlled melting. The molten material is then deposited layer by layer to build the 3D-printed object.
2. **Types of FDM Hotends:** FDM printers can have various types of hotends, including all-metal hotends (suitable for high-temperature filaments), PTFE-lined hotends (suitable for standard filaments), and more. Some hotends are designed for specific applications, such as those compatible with flexible filaments.
3. **Features:** Many hotends allow for nozzle replacement, enabling users to switch between different nozzle sizes for varying layer resolutions. Additionally, some hotends are equipped with cooling systems to regulate the temperature and prevent heat from traveling too far up the filament, particularly during intricate prints.

FDM Printing

Main parts of an FDM printer

Print Bed Types:

BuildTak:

Description: BuildTak is a brand of 3D printing surface material that provides strong adhesion during printing and easy removal of prints once completed. It is known for its durability and compatibility with a variety of filament types.

PEI (Polyetherimide):

Description: PEI sheets are transparent, high-temperature-resistant surfaces that offer excellent adhesion when heated. They are popular for printing with materials like ABS and PETG. Prints usually adhere well when the bed is heated, and they tend to release easily when the bed cools down.

Glass:

Description: Glass beds are flat, rigid surfaces that can be coated with various materials for improved adhesion. Some users apply glue sticks, hairspray, or special coatings like "nano-polymer" solutions to enhance adhesion. Glass beds are known for providing a smooth and flat printing surface.

Magnetic Flexible Build Plates:

Description: These systems often consist of a flexible, magnetic base and a removable, flexible build surface. Print adhesion is facilitated by the flexibility of the surface, making it easier to remove prints after completion. Popular brands include PEI-coated spring steel sheets on a magnetic base.

Textured PEI or BuildTak Surfaces:

Description: Some printing surfaces come with textures or patterns that promote better adhesion. These textured surfaces can be PEI-based or similar to BuildTak and help prints stick more effectively.

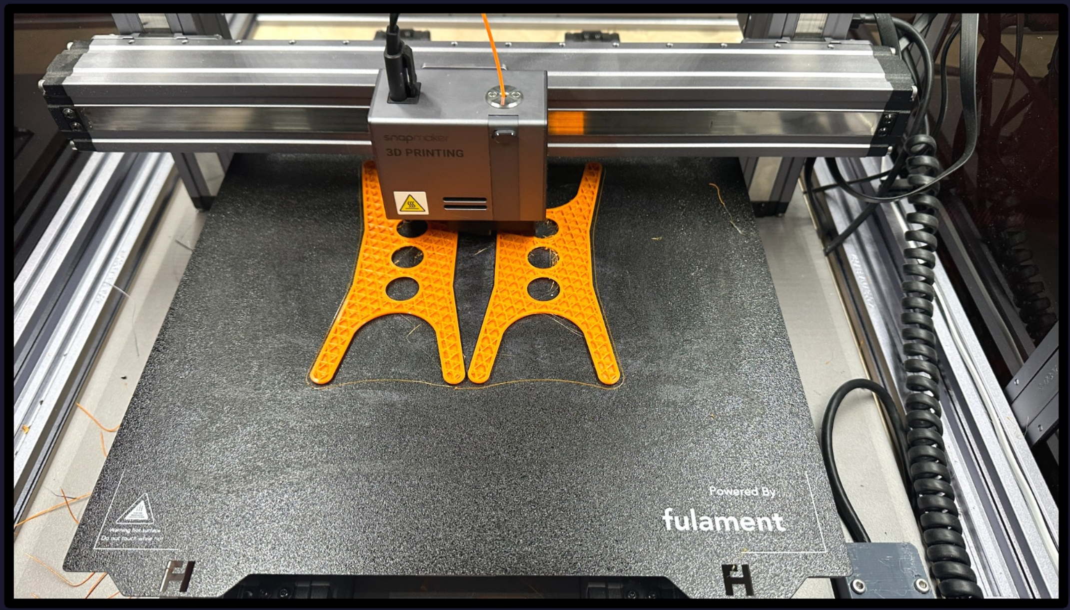
Aluminum with Adhesive Surfaces:

Description: Aluminum beds with attached adhesive surfaces, such as polyimide tape, provide a flat and even printing surface. The adhesive layer helps with print adhesion and is replaceable when needed.

Mirror Tiles:

Description: Some users opt for mirror tiles as a printing surface. The reflective surface can be coated with adhesion-promoting substances, providing a smooth and even base for printing.

PEI Coated Magnetic Flexible Metal Build Plate



In my opinion this offers more than any other build plate available for FDM printing.

Preparing to Print with Your FDM Printer

Now that you have your object ready to print the next step is preparing the printer. This consists of:

- Bed Leveling
- Cleaning and prepping the bed surface
- Loading and priming the FDM filament if applicable
- Loading the print file to the printer

Fortunately, most new printers have automated bed leveling. And the only manual step is the final which sets the print head distance from the bed surface. This is generally between 0.1mm and 0.3mm depending on the make of the printer.

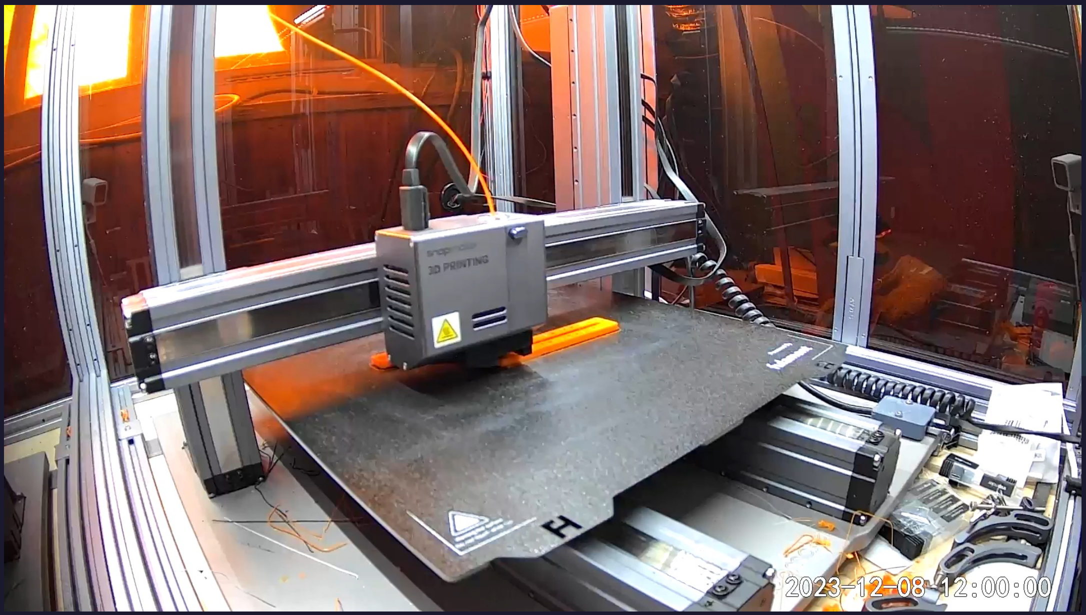
Cleaning and Adhesion Tip:

- Regardless of the type of bed your printer has, keeping a bottle of Isopropyl Alcohol near is helpful.
- Clean regularly to prevent any buildup of material on the bed surface.
- Painters Tape – This can be extremely helpful especially when dealing with a glass print bed, as it gives you a thin surface that is at time more easily removed than a print that is stuck to the surface.
- Elmers Washable Stick Glue or Hairspray – These can be extremely helpful in aiding in bed adhesion and even removal for some of the harder to remove filament types such as PETG.

Preparing to Print with Your FDM Printer

Now load your file and you are ready to start printing with your FDM printer.

Video

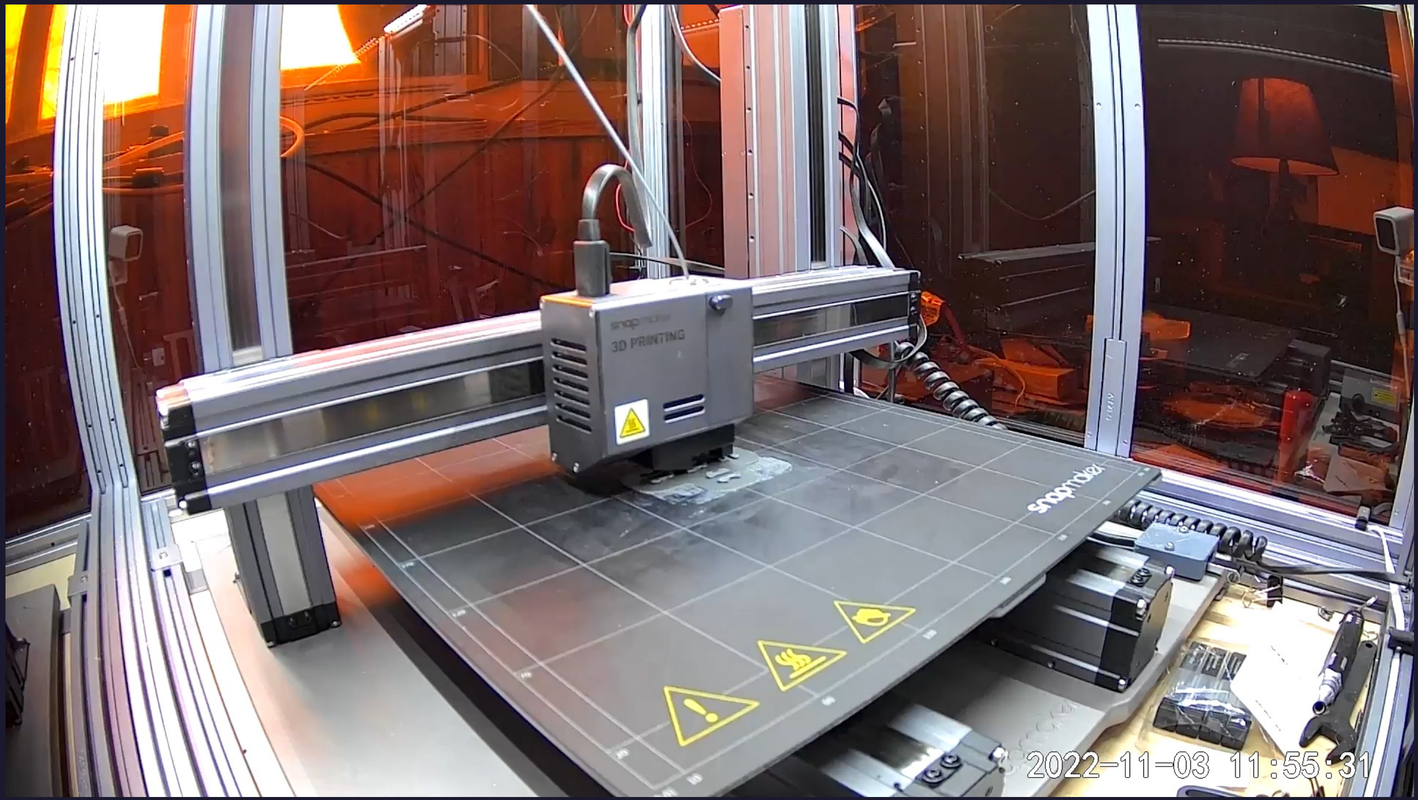


Tips for a successful first print:

- Start small and try to watch how your printer acts during the print before you attempt something that takes hours if not days.
- It is most important to keep an eye on the first few layers as that is when the majority of the issues happen.
- If your printer does not have a camera, it would be advised to get one before attempting a long print. It gives you the ability to check on the print periodically to catch any issues before they turn into problems.

FAILED Print

Video



FDM Filament Storage and Drying

1. Storage:

- **Cool and Dry Environment:** Store filament in a cool, dry place away from direct sunlight and moisture. Humidity can negatively impact filament quality.
- **Sealed Containers:** Keep filament in airtight containers or vacuum-sealed bags with desiccant packs to prevent exposure to air and moisture.
- **Dedicated Filament Storage:** Consider using dedicated filament storage solutions, such as dry boxes or filament storage cabinets, which can maintain controlled humidity levels.
- **Avoid Extreme Temperatures:** Avoid storing filament in extreme temperatures, both hot and cold, as this can affect its properties.

2. Desiccant Packs:

- **Silica Gel Packs:** Place silica gel desiccant packs inside the filament storage containers to absorb any moisture that may be present.
- **Replace Desiccants:** Regularly replace or recharge silica gel packs to ensure continued moisture absorption capabilities.

FDM Filament Storage and Drying

. Drying Filament:

- **Dehydrator or Filament Dryer:** Invest in a dedicated filament dehydrator or filament dryer. These devices are designed to remove moisture from filament effectively.

- **Oven Method:** If using a conventional oven, preheat it to a low temperature (usually around 50°C to 60°C or 122°F to 140°F), place the filament on a tray, and let it dry for a few hours. Be cautious not to exceed the recommended temperature to avoid damaging the filament.

- **Time and Temperature:** The drying time depends on the level of moisture and the specific filament material. Generally, a few hours at a low temperature should be sufficient.

Resin (SLA) Printing

SLA, or Stereolithography, is a type of 3D printing technology that uses a process called photopolymerization to create highly detailed and precise three-dimensional objects. It is a form of resin-based 3D printing that operates by selectively curing liquid photopolymer resin layer by layer.

Common Types of SLA Resin

1. **Standard Resins:**
   1. **Properties:** Standard resins provide a balance of properties, including strength, flexibility, and surface finish.
   2. **Applications:** General-purpose prototyping, concept models, and functional parts.
2. **Flexible Resins:**
   1. **Properties:** Flexible resins are characterized by their elasticity and rubber-like properties.
   2. **Applications:** Prototyping of flexible and bendable parts, such as gaskets, shoe insoles, and flexible prototypes.
3. **Tough Resins:**
   1. **Properties:** Tough resins offer enhanced impact resistance and durability.
   2. **Applications:** Functional prototypes and parts requiring high toughness and resistance to impact.
4. **High-Temperature Resins:**
   1. **Properties:** High-temperature resins are designed to withstand elevated temperatures.
   2. **Applications:** Prototyping for parts exposed to heat, such as under-the-hood automotive components.

Differences

- **Resolution:** Resin printers typically have much better resolution as they can cure the resin in as little an area as a single pixel from a 4K mask

- **Part strength:** Filament is still out ahead of resin in strength, but not by much. Newer resins are very strong.

- **Part flexibility:** Although this depends heavily on the resin (or filament) chosen, resin printed parts tend to be more rigid than filament printed parts

Resin Printing Steps

Printer Setup:

•**Resin Selection:** Choose the appropriate resin type based on the requirements of your project (e.g., standard resin, flexible resin, castable resin).

•**Printer Calibration:** Ensure that the SLA printer is properly calibrated for accurate layer deposition and curing.

•**Build Platform Leveling:** Level the build platform to ensure that the first layer adheres evenly to the build plate.

Resin Printing Steps

Printing:

•**Resin Pouring:** Fill the SLA printer's resin tank with the chosen resin material. Ensure that the resin is evenly spread across the bottom of the tank.

•**Printer Operation:** Start the SLA printer, which will begin the layer-by-layer printing process. The printer's UV light source selectively cures the liquid resin, solidifying it into the desired shape.

•**Monitoring:** Regularly monitor the printing process to ensure that layers are curing correctly and that there are no issues.

Resin Printing Steps

Post-Processing:

•**Resin Drainage:** After printing is complete, the build platform is raised, and excess resin drips back into the resin tank.

•**Object Removal:** Carefully remove the printed object from the build platform. Wear gloves and use tools like spatulas to avoid direct contact with uncured resin.

•**Washing:** Wash the printed object in a solvent (usually isopropyl alcohol) to remove any remaining uncured resin. This helps improve the object's final properties.

•**Curing:** Cure the printed object using UV light. This final curing step ensures that the resin is fully polymerized, resulting in a hardened and stable final part.

Resin Printing Steps

Final Inspection:

•**Quality Check:** Inspect the printed object for any defects, layer adhesion issues, or areas that may require additional post-processing.

•**Support Removal:** If support structures were added during slicing, carefully remove them using appropriate tools.

Finishing:

•**Surface Finishing:** Depending on the desired finish, additional post-processing steps such as sanding, polishing, or coating may be performed.

•**Storage:** Store the printed object appropriately, taking into account the specific material properties.

Cleanup Tips

- **Wear gloves/mask:** The uncured resin is a skin and lung irritant. Use gloves in all cases, and either a VOC approved mask or operate in well ventilated area.

- **Use a small, stiff putty knife:** This will help you release the part from the plate, then will be used as a spatula to cleanup and recapture uncured resin.

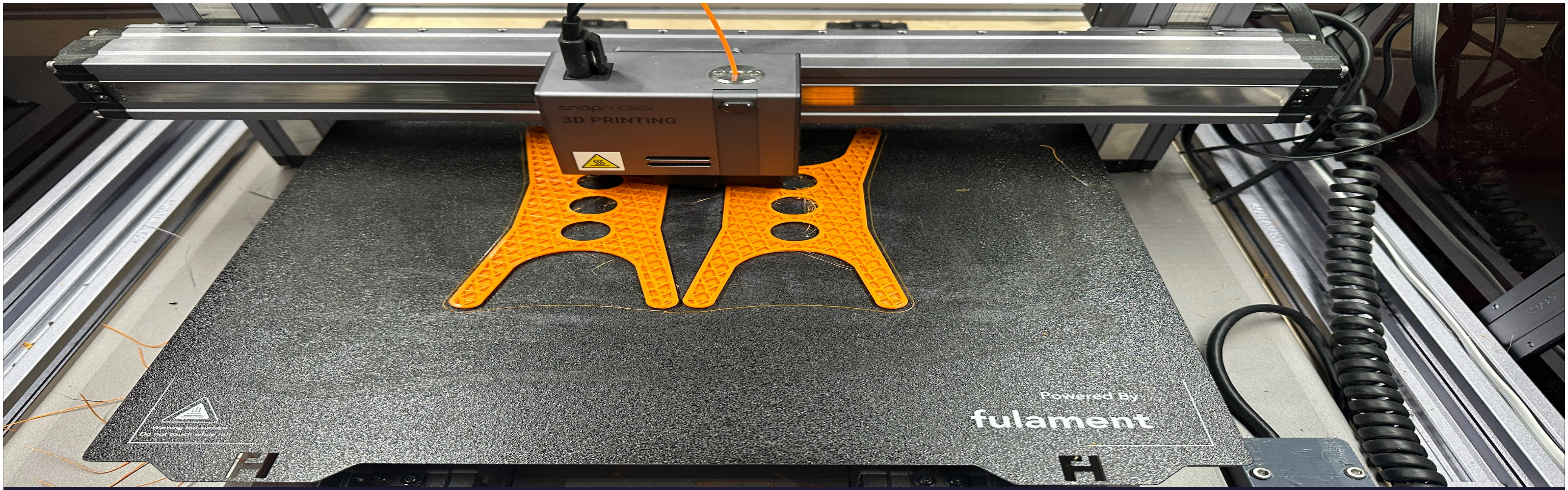
- **Use high percentage alcohol:** The higher percentage of isopropyl alcohol you can get, the easier the cleanup. 99% is ideal.

- **Draw shades/blinds during cleanup:** Any extraneous UV rich light (sunlight) will cure the resin very quickly and with unexpected results

- **Make sure to clean/empty your printer wash basin:** Washed resin will eventually settle and clog bearings

Troubleshooting

- Bed Adhesion: This is almost 100% of the time related to an unlevel bed. Too short of an exposure First layers is also a cause.
- Parts disconnecting: Make sure you pay attention to supports and use them as needed. Remember that you are printing upside down.
- Resolution: Make sure your slicer is set to the correct printer.
- Tools: Use the resin manufacturers settings in your slicer and you will likely have success on your first try



Closing

Remember that successful 3D printing is often a matter of trial and error, especially when working with different materials and printer configurations. It's a good practice to start with the manufacturer's recommended settings for your specific filament and adjust as needed based on your printing experience and the requirements of your project.

Thank You

James Cizek – KI0KN

Jim Dixon – KA6ETE