INKSMITH

3D PRINTING GUIDE

EVERYTHING YOU NEED TO KNOW TO START 3D PRINTING

ABOUT INKSMITH

IT'S NOT ABOUT TEACHING TECHNOLOGY, IT'S ABOUT TEACHING WITH TECHNOLOGY.

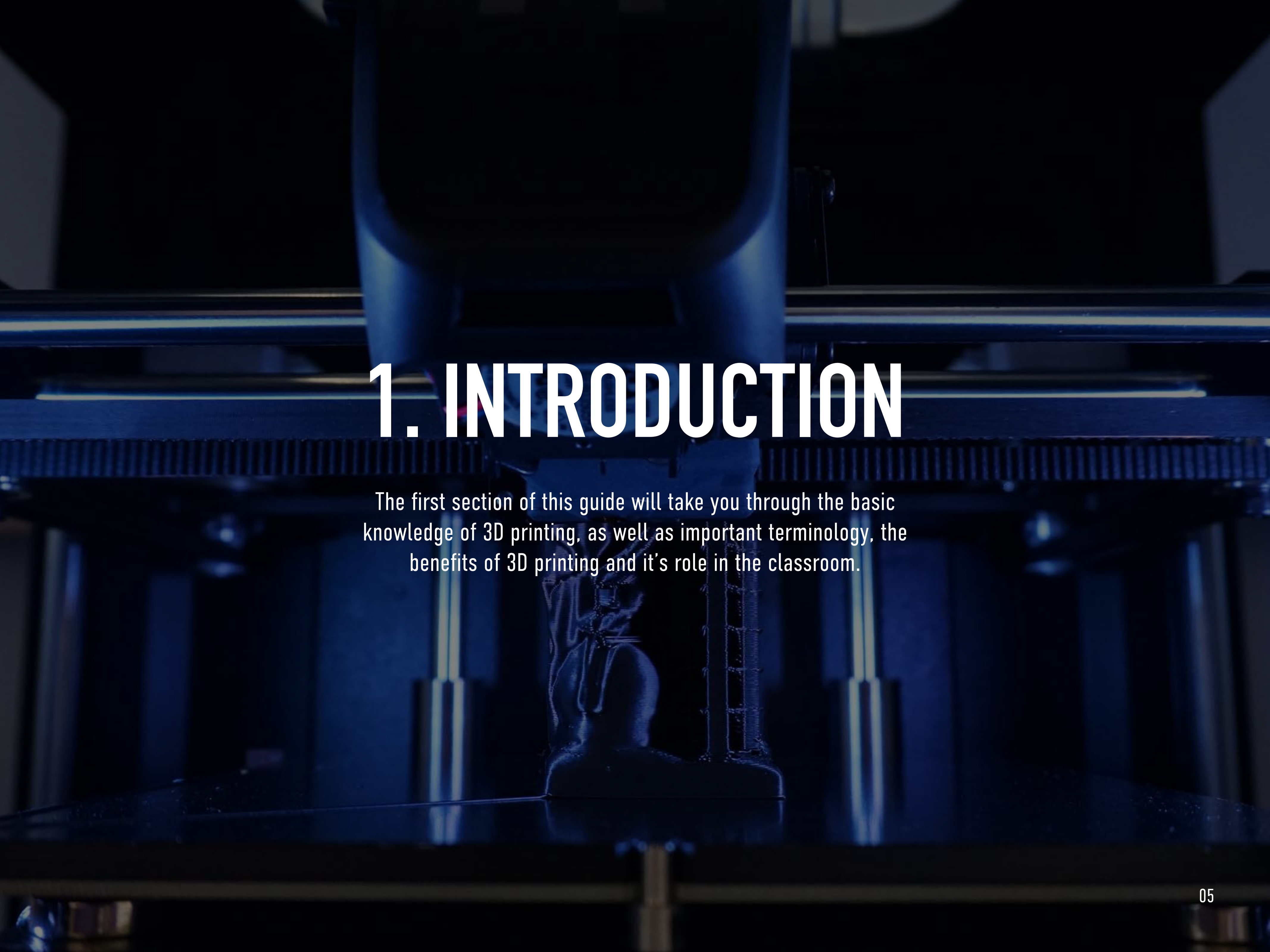
Our mission at InkSmith is to help educators across Canada build hands-on, experiential learning opportunities to introduce their students to STEM/STEAM competencies and design thinking principles. We focus on creating immersive education solutions designed for 21st-century learning.

We've put together this document as a resource for educators to help get you started with 3D printing. If you have any questions or need further assistance, don't hesitate to contact our technical support team:

tech@inksmith.ca
1-844-465-7684 ext. 1

TABLE OF CONTENTS

TABLE OF CONTENTS



1. INTRODUCTION

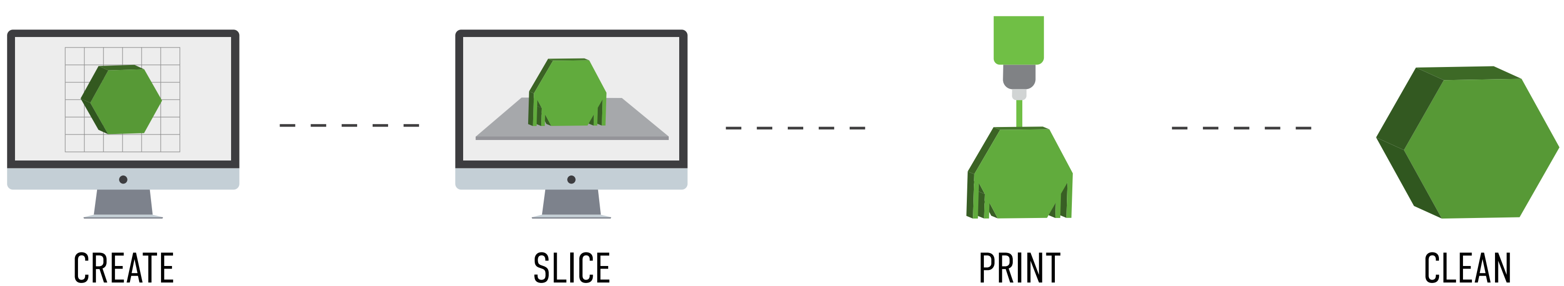
The first section of this guide will take you through the basic knowledge of 3D printing, as well as important terminology, the benefits of 3D printing and it's role in the classroom.

1.1 WHAT IS 3D PRINTING?

In some ways, 3D printing can be a bit of a misnomer as there are more than 20 different technologies that fall under the umbrella. What we use in the classroom is called Fused Deposition Modeling or FDM; it's essentially a hot glue gun on 3 axes that uses a variety of plastic strings called filament (which looks just like weed wacker wire) to create anything you or your students imagine!

These machines are used extensively in industry for prototyping, but are the natural choice for classrooms because they are safe, affordable, simple to operate, and still offer the same teachable skills and operating principals as the other technologies.

THE 3D PRINTING PROCESS



1.2 3D PRINTING TERMINOLOGY

AUTO-LEVEL: Auto-leveling is the process in which the printer mechanically adjusts the print bed before each operation to ensure there is a level surface to print on.

BED: The bed is the surface on which the model is printed. It moves up and down (Z-axis) in increments that are fractions of a millimetre based on chosen print resolution. When the print is done it will still be attached to the bed surface and must be removed.

FILAMENT: Filament is the basic input material and is stored on large spools in a plastic "wire". The printer pulls the filament into the machine and pushes it into the extruder so it can be melted to make the layers of your model.

EXTRUDER: The extruder is the component that melts the plastic filament at a very specific rate. The printer moves the extruder left-to-right (X-axis) and front-to-back (Y-axis) to lay down layers of plastic.

G-CODE FILE: This is the output from the slicing software. It's a long series of coordinates and instructions about how to move the motors in the printer.

LOADING: Loading is the process where the printer filament is loaded into the printer and fed to the extruder. Similarly, "unloading" is the process of taking the filament out of the printer and flushing out the extruder.

NOZZLE: The nozzle is the component of a 3D printer that deposits the molten filament into the build area.

PRINT RESOLUTION: Print resolution refers to the thickness of the layer height. Models with thicker layers will print faster but have more visable surface layers. Models with thinner layers will have a more seemless print surface but will take longer print.

SLICING: Slicing refers to the process of taking your exported .STL file and translating it into a language the printer can understand. You can also add any necessary supports and change the model orientation in this step.

STL FILE: .STL is the file format that is usually generated by a computer-aided design program. This file format can then be imported to most slicing softwares.

1.3 BENEFITS OF 3D PRINTING

Today, 3D printers are being used as tactile learning tools which allow students to bring their own designs and creations to life. With wider adoption, ongoing professional development for educators, and the availability of affordable machines, 3D printers are becoming increasily popular in education.

From abstract 3D models to teach surface area and volume, to anatomy, topographical maps, engine blocks and historical artifacts - 3D printing adds a new dimension to your students learning across a wide spectrum of class subjects.

There are endless ways to use 3D printing in the classroom to teach otherwise rudimentary topics. It's not about teaching students how to 3D print, but rather using 3D printing to get students interested and engaged in what they're learning. From math to history, and every subject in between, there are hundreds of ways to integrate 3D printing into everyday curriculum.

3D printing allows educators to challenge their students' design thinking and problem solving skills. It allows students to solve real-world issues with tangible solutions. Through these design challenges, students will be equipped with the experience and skills to approach solving problems with design thinking; anticipating failures, intuition for efficiency, and an understanding of the need for iteration.

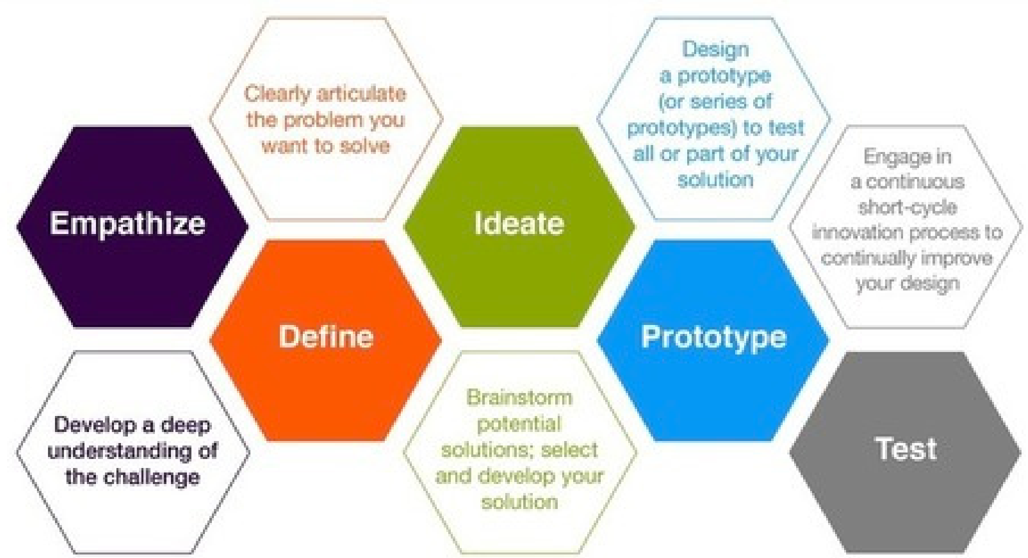
Most important of all, you'll instill the confidence to succeed through 'failure'.

1.4 3D PRINTING AND DESIGN THINKING

When it comes to teaching and implementing design thinking, it's important to introduce challenges that allow students to create multiple iterations of their solution. 'Failure' is an integral part of the process as this experience influences future iterations and builds a mental "what if" framework that guides their efforts. More than just a process, design thinking is a tool that can be utilized in a wide range of activities, both mental and physical.

The process of design thinking requires users to understand the challenge, define the problem, ideate, prototype and then test their solution. All of these steps play an important role when it comes to 3D printing. The design thinking framework can be applied to multiple subjects from history, to math and everything in between. Whatever problem you're trying to solve, using the design thinking framework and 3D printing will help you get there.

Not only can 3D printing be used in the practice of design thinking, but the practice of 3D printing can be used to teach design thinking.



1.5 3D PRINTING BY SUBJECT

MATH: In math, 3D printing can be used to model equations or print objects on which to calculate surface area and volume. You could also use 3D printing to teach fractions, shapes, and equations by 3D printing physical models which can be used to demonstrate these concepts to students.

SCIENCE: The possibilities of 3D printing to teach scientific concepts are endless. You could 3D print the parts of a cell, teach the periodic table with 3D printed molecules, or build a 3D printed car to teach speed, velocity and other laws of physics.

HISTORY: Help students understand different eras of history by 3D printing replica medieval technology or ancient artifacts. 3D print a model of an ancient Egyptian pyramid or a replica of the Mayflower ship - either of which is sure to get students interested and engaged.

GEOGRPAHY: What better way to learn about topography or tectonic plates than to have 3D printed versions the students can hold and analyze. Teach students about the depths of the ocean by 3D printing the deepest ocean trenches or underwater volcanoes.

SOCIAL STUDIES: Teach students about important life skills by 3D printing the braille alphabet, or challenging students to create 3D printed assistive devices to help those with a disability. You could even teach students about safety concepts by 3D printing a whistle that really works - the possibilities are endless!

2. FILE SOURCING AND CREATION

In order to start 3D printing you will need actual printable files. 3D printing files can be designed using a CAD software or be download from one of the many open-source platforms online.

2.1 FILE SOURCING PLATFORMS

The easiest way to obtain 3D printing files is to download them from one of the many online open-source file networks. There are several open-source file networks from which you and your students can access millions of free 3D models across a wide range of subjects. Below are the three most common open-source file networks:

THINGIVERSE: Thingiverse is a website dedicated to the sharing of user-created digital design files. As the world's largest 3D printing community, Thingiverse believes that everyone should be encouraged to create and remix 3D things, no matter their technical expertise or previous experience. In the spirit of maintaining an open platform, all designs are encouraged to be licensed under a Creative Commons license, meaning that anyone can use or alter any design.

PINSHAPE: Pinshape is a marketplace that showcases the digital work of 3D designers from all over the world. 3D print designers set their own prices for their design files, and also choose which license to offer their work under (Creative Commons or other). People with 3D printers can browse the selection of designs and then either get the file for free to print themselves, or pay the designer for access to the file before printing.

GRABCAD

GRABCAD: GrabCAD is an online, cloud-based platform that allows professional designers, engineers, manufacturers, and students to download and upload to the largest collection of professional CAD models anywhere on the internet. GrabCAD also has online tutorials to learn from other designers as well as challenges to show off your skills and win prizes.

2.2 CAD SOFTWARE

If you wish to create and design your own 3D printing files you'll need to use a CAD software. The abbreviation CAD stands for Computer Aided Design. CAD software will allow you to create and modify 3D objects. There are multiple CAD software platforms available, below is a list of platforms we suggest:

TINKERCAD: Tinkercad is an easy-to-use, browser-based 3D design and modeling tool. Tinkercad allows users to imagine anything and then design it in minutes.

SKETCHUP: SketchUp, formerly Google Sketchup, is a 3D modeling computer program for a wide range of drawing applications such as architectural, interior design, landscape architecture, civil and mechanical engineering, film and video game design.

VECTARY: Vectary allows users to create 3D models with a drag and drop 3D modeling tool. You can also collaborate with friends and colleagues through their online platform.

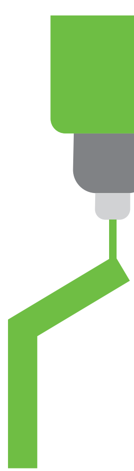
ONSHAPE: Onshape's modern CAD system frees engineers to focus on doing their best work. Unlike old CAD systems, Onshape unites modeling tools and design data management in a secure cloud workspace that is accessible on any device, never loses data, and eliminates design gridlock.

2.3 DESIGN CONSIDERATIONS

Whether you're designing a 3D model yourself or downloading an open-source file, there are some important design aspects you should consider before printing. Keeping these design considerations in mind will help to improve your 3D printing results.

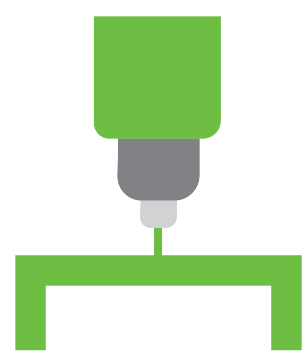
OVERHANG

Overhang occurs when any part of the print extend outward, beyond the previous layer, without any support. Overhang can lead to drooping and/or poor surface quality.



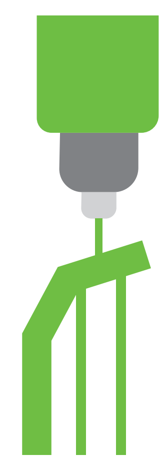
BRIDGING

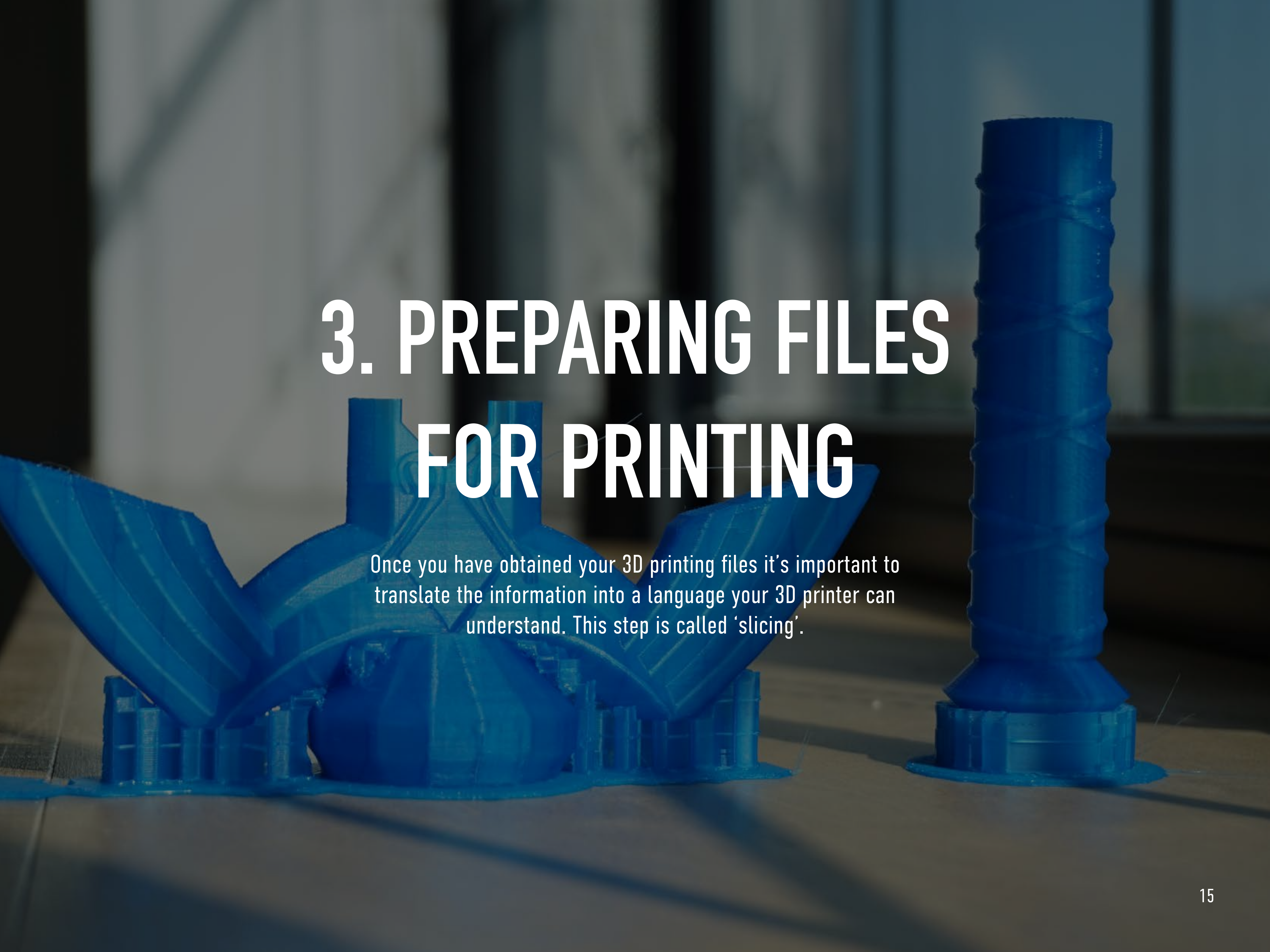
Bridging is when any part of the print extends horizontally, in mid air between two points. The bigger the gap between the two bridging points, the more likely the structure is going to be compromised.



SUPPORTS

Supports are temporary structures that are added during the slicing process to support any overhang, bridging, or weaker parts of the object. Supports are usually broken or peeled off after printing.





3. PREPARING FILES FOR PRINTING

Once you have obtained your 3D printing files it's important to translate the information into a language your 3D printer can understand. This step is called 'slicing'.

3.1 OBJECT ORIENTATION

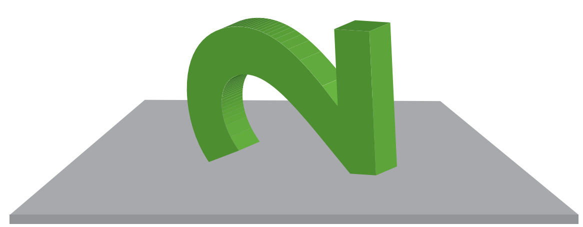
Object orientation is an extemely important part of preparing your file for print. Object orientation refers to the position in which your object will be printed on the bed. Object orientation can be adjusted using the slicing software prior to printing. The goal during this step is to orient your object so that you are printing with the most possible flat surface area touching the print bed. Doing this will allow for the most success during printing. Here are some examples of incorrect and correct object orientation.

EXCESSIVE OVERHANG

This object orientation creates an excess overhang at the top portion of the number 2. This excessive overhang could lead to drooping or warping during printing and/or poor surface quality.

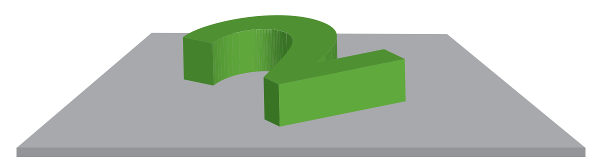
POOR BED ADHESION

This object orientation has very few points of contact on the printing bed. Poor bed adhesion could lead to prints becoming unstable or slipping off the bed during printing.



CORRECT ORIENTATION

This object orientation is the best possible option for this print. This orientation avoids any overhangs and allows for the most possible surface area to be in contact with the print bed.



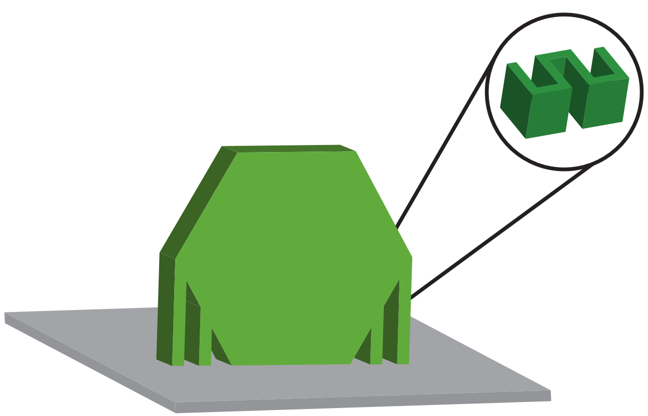
3.2 ADDING SUPPORTS

Along with object orientation, adding supports is another important step achieved with the slicing software. Supports are temporary structures that are printed along with your object to ensure the integrity and quality of the print. Most slicing softwares have a built in support feature. If selected, this feature will automatically add supports throughout your model as required.

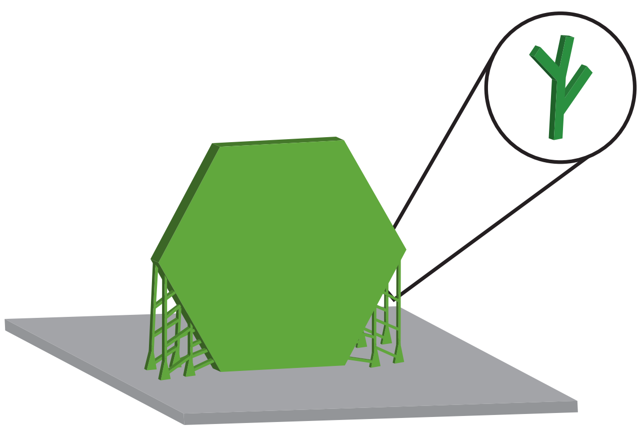
Supports are used whenever there is an overhang or bridge in order to ensure the structural integrity of the print. The temporary support structures can be peeled off after the print is complete. The downside however, is that depending on the size of the support, the surface quality can sometimes be compromised after the material is peeled off.

There are generally two types of support structures used in FMD 3D printing, accordion or lattice, and tree supports. Accordion or lattice supports are the most common, they're easy to add in and work well with most 3D models. Tree supports are just like they sound, a series of trunks that branch out to support bridges and overhangs.

Accordion/Lattice Supports



Tree Supports



3.3 BRIMS, RAFTS & SKIRTS

Brims, rafts, and skirts are three different types of base layers which can be added to 3D models in order to improve the bed adhesion and/or model stability.

BRIMS

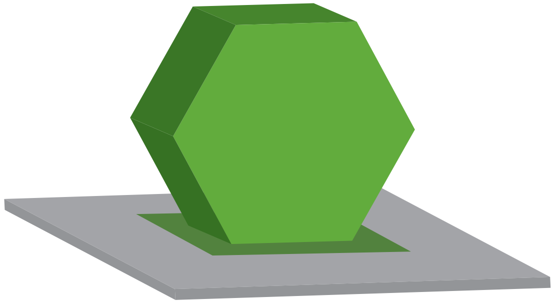
A brim is a thin layer of material that extends beyond the edge of the object in order to increase the base surface area. Brims help to improve bed adhesion and hold down the edges of a print which helps to prevent warping.

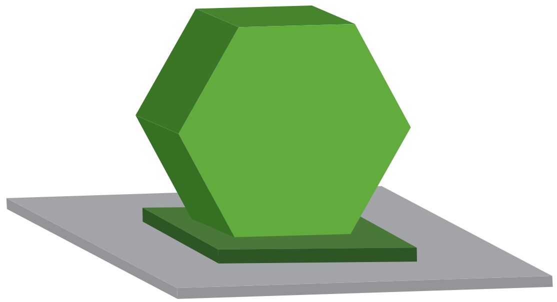
RAFTS

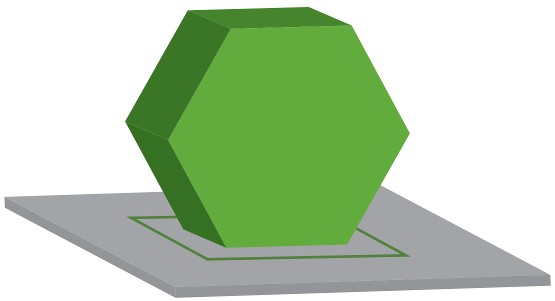
Rafts are similar to brims in that they provide greater surface area for improved bed ahesion. Rafts however are thicker than brims and are used to provide stability to objects with smaller footprints or to create a strong foundation for upper layers.

SKIRTS

A skirt is a thin line of material surrounding the printed object. Skirts help to prime your extruder and establish a smooth flow of filament. Observing the skirt also allows you to detect and adjust any leveling or adhesion issues before the actual model begins printing.



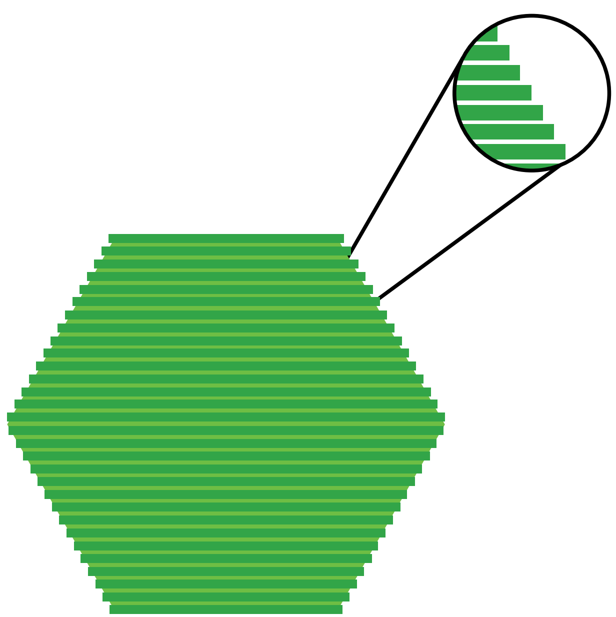




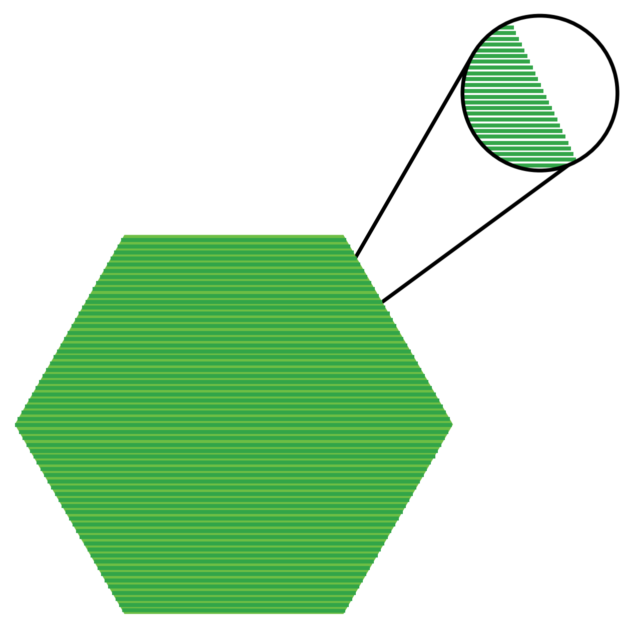
3.4 PRINT RESOLUTION

Print resolution refers to the layer height of of the 3D print. An object with a low print resolution will have thicker layers, often leading to a more noticable surface area. Printing with a low resolution often results in faster print times because the layer height is thicker and builds up the print object at a faster rate. On the other end of the spectrum, an object with a high print resolution will have much thinner layers, which will create a more seamless surface. Because the layers are much thinner, printing in a high resolution will often result in signficiantly longer print times. A balance between print quality and print time much be reached depending on the object being printed.

LOW RESOLUTION



HIGH RESOLUTION



3.5 TYPES OF INFILL

HOLLOW

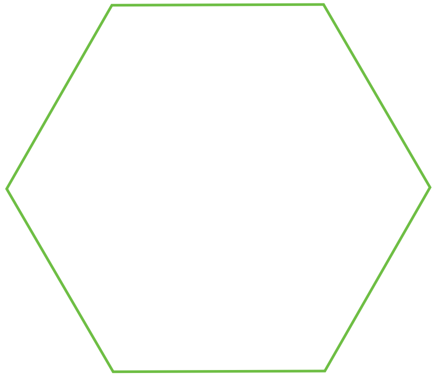
Hollow infill is just as it sounds, a model with zero infill. Hollow models work with things like cylinders, cups or objects that are designed to hold something inside of them. However, hollow infill will not work for models that have a top surface because without infill there is nothing for the top surface to be printed on.

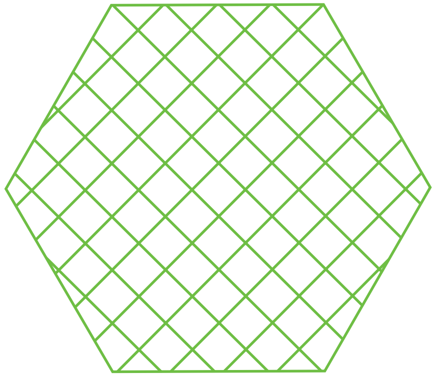
REGULAR

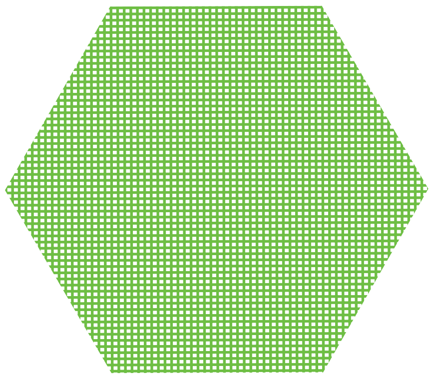
Regular infill is typically around 20% density and is printed in a pattern which maintains the integrity of the object and provides overall strength. Regular infill is suited best for objects that are enclosed but do not need to be completely solid.

SOLID

Solid infill is when an object is printed entirely solid from inside to outside. Printing with solid infill is not very common as it requires much more material and printing time. However solid infill may sometimes be used when density and mass is required.

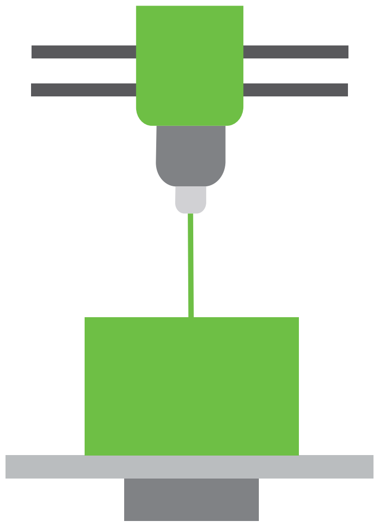






3.6 PRINT SPEED AND TEMPERATURE

PRINT SPEED

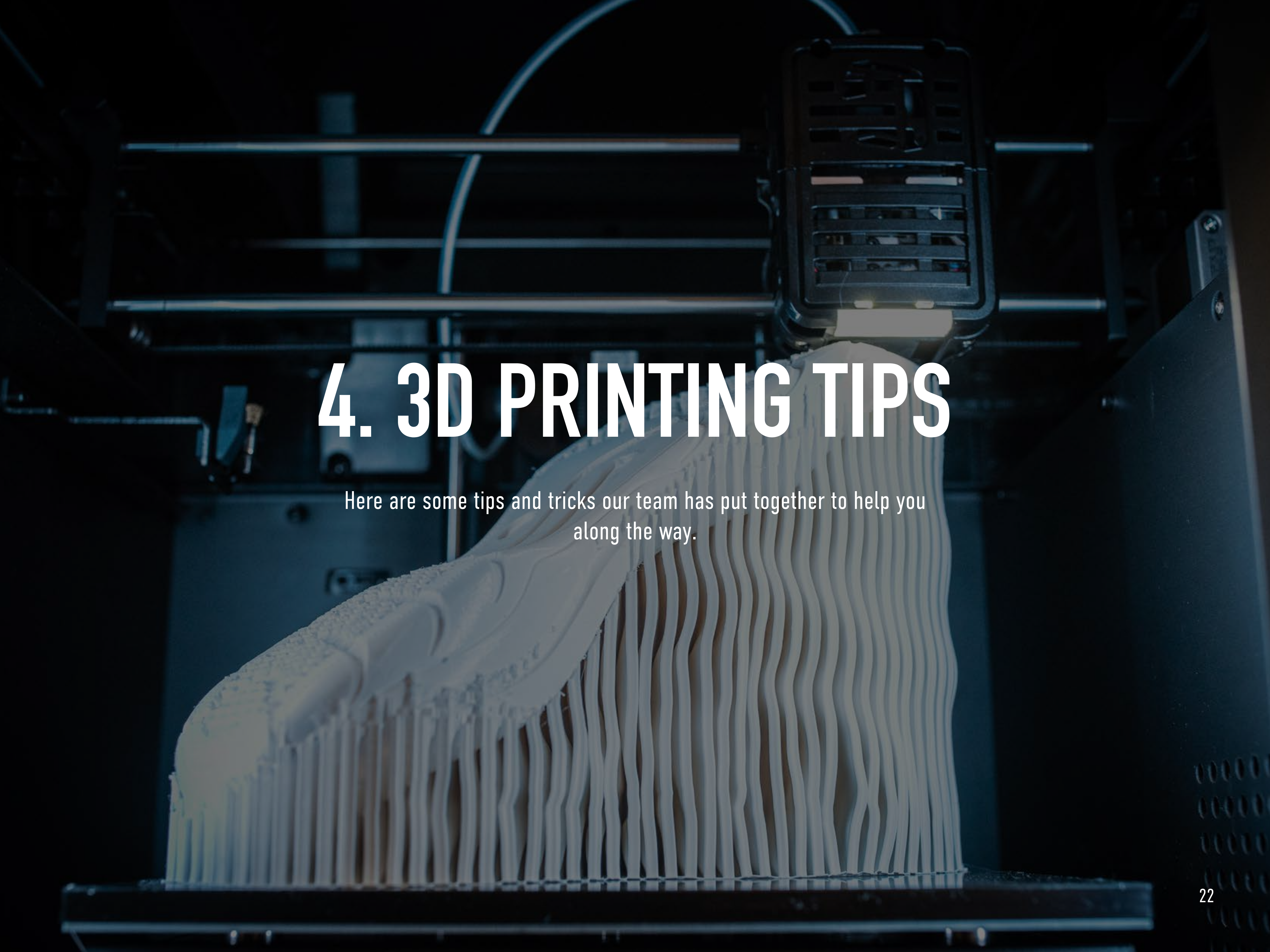


Finding the correct print speed for your projects is a fine balance between print quality and time. The faster you set a project to print, the lower quality the print will be. This is because the faster the extruder is moving, the more likely you are to have underextrusion. Underextrusion is when the extruder does not have adequate time to extrude enough filament to fufill that layer. Vise versa, if you wish to have a very high quality print you will need to allow significant time for the print to process. This is because the extruder is moving much slower to ensure proper extrusion and alignment with each layer. The balance between print speed and quality will vary depending on the object being printed, material and other requirements of the project.

PRINT TEMPERATURE

Print temperature is the temperature at which the plastic filament is extruded from the machine. Depending on which type of filament you are using, the print temperature will vary. Before every print you will need to ensure that the printer is set to extrude at the correct temperature. Most filaments will state the required temperature on their packaging however sometimes this process requires some trial and error to determine the exact printing temperature.

In addition to extrusion temperature, some 3D printers with heated beds will also require a print temperature to be set for the printing bed. Setting the correct temperature for the printing bed will help to ensure that the filament will stick during operation.



4. 3D PRINTING TIPS

Here are some tips and tricks our team has put together to help you along the way.

4.1 CALCULATING FILAMENT COST

Calculating the cost of filament is an important part of budgeting for the use of a 3D printer. To calculate the approximate cost of any print, we suggest using a slicing software to estimate the amount of filament material that will be used. Once you know this information you can calculate the cost per 1 gram of your filament to figure out the total cost of each print. Start by answering the questions below:

1. What is the weight of your filament reel in grams? ______________________________

2. How much does one filament reel cost? _______________________________________

Calculate the cost per 1g of filament using this equation:

cost of filament reel / weight of filament reel

3. My cost per 1g of filament? _______________________________________

Now that you have the cost per 1g of filament you can take that number and multiply it by the number of grams of material your print will use as determined by the slicing software.

5. 3D PRINTING CURRICULUM

Learning how to effectively integrate 3D printing into the classroom curriculum is just as important as learning how to use the technology. In this section you will find a number of examples of 3D printing lesson plans, ranging on variety of subjects and difficulties.

5.1 LESSON PLAN: DINOSAUR DIG

In this activity, students will work as a class to research paleontology and fossil excavation before creating a 3D printed Tyrannosaurus Rex skull. For the purposes of evaluation, students will break into four smaller groups. As fossils are unearthed, they tell a story about the dinosaur's life. Therefore, each group should research one aspect of the world as it was when the T-Rex roamed the planet (70 to 65 million years ago).

For example, groups can explore;

- Diet: what did the T-Rex eat? Describe what the T-Rex may have hunted/consumed.
- Environment: what was the environment like for the T-Rex? Describe what the T-Rex would have encountered on an average day, including other organisms and plant life.
- Survival: did the T-Rex give birth to live offspring or eggs? Describe the average life cycle of the dinosaur.
- Extinction: how/when did the T-Rex become extinct? Describe theories and evidence from scientific research.
- Fossilization: how does a dinosaur become a fossil? Describe the process and consequences of fossilization.

Following this research, groups will compile their information into written/presentable format. Each one of the four groups will be assigned a part of the T-Rex to model using software and print on the 3D printer.

5.2 LESSON PLAN: IT'S AN EMERGENCY

In this activity, students will work in groups of 4-5 to research how to respond in emergency situations (e.g., fire, flood, road accidents, suspicious vehicles) before creating their very own 3D printed whistle and emergency response plan. For the purposes of evaluation, students will research and create a brief oral presentation (15 minutes) about responding safely to emergency situations.

They should pick one particular situation, and describe a plan for how to respond. They should prepare their presentation with these points as guidelines:

- What are some common examples of emergency situations? Identify one to focus on for the presentation.
- What are the steps that should be taken when responding to emergencies?
- What barriers or challenges could there be to responding in emergencies? Identify these in relation to the emergency focused on for the presentation.
- How could sound play a role in emergency situations?
- What other tactics could be helpful to use when responding in emergencies?

Following this research, students will create their own emergency whistle using 3D printing CAD software.

5.3 LESSON PLAN: HANDLING HEAT

In this activity, groups of 4-5 students will collaboratively design, build, test and modify a series of 3D printed containers that minimize heat loss, or maximize cool temperatures for a given volume of liquid.

For the purposes of evaluation, the students will be given a set of design constraints which can either be prioritized by the instructor or in individual groups.

- Overall object height should be no taller than 25 cm.
- Final designs should have an overall capacity of at least 355 ml.
- Designs should consist of one main body and a removable cap.
- Group Bonus; Final design includes a component that can steep ingredients and infuse flavour, minimize print time, minimize filament usage.
- Individual Bonus; Final design includes a storage compartment of some sort.

Designs will be created using one of the following 3D modelling software packages; TinkerCAD, Fusion 360, Inventor, SolidWorks or Onshape.

Students may be allowed multiple iterations of their design for this project at the instructor's discretion, with the goal of allowing them to ideate, create, test and iterate according to their findings.

5.4 LESSON PLAN: HELPING HANDS

In this activity, 4-5 students will work in groups to create a prosthetic hand.

E-nable is a product from enabling the future, a company that makes open source, 3D printable prosthetic limbs. Students will create the hand as part of E-nable's Lend-a-hand initiative where the hands they create will be sent to those who lack the means of production to create them themselves.

Students will size a hand according to the sizing guide provided on the website and follow those instructions to print and subsequently assemble one of the open source designs from the website.

In the first session, students will conduct research on prosthetics, and in in the second session, they will print and assemble the hand. The files for this project will be created using Blender, an open source and free software.

5.5 LESSON PLAN: UNDER SIEGE!

This activity spans several sessions; the first focuses on testing out 3D printed catapults and trebuchets to achieve launching an object a specific distance. Then, students will gain experience communicating procedural thinking to peers by creating and providing instructions for other groups to replicate their launch.

The second session focuses on how other groups received instructions and any improvements that could be made to the design.

The third session focuses on creating and/or modifying 3D-printed models of a siege engines and reflecting on how the design functions or what could be improved.

Designs will be created using one of the following 3D modelling software packages; TinkerCAD, Fusion 360, Inventor, SolidWorks or Onshape.

Students may be allowed multiple iterations of their design for this project at the instructor's discretion, with the goal of allowing them to imagine, create, test and revise designs according to their findings.

6. TROUBLESHOOTING AND MAINTENANCE

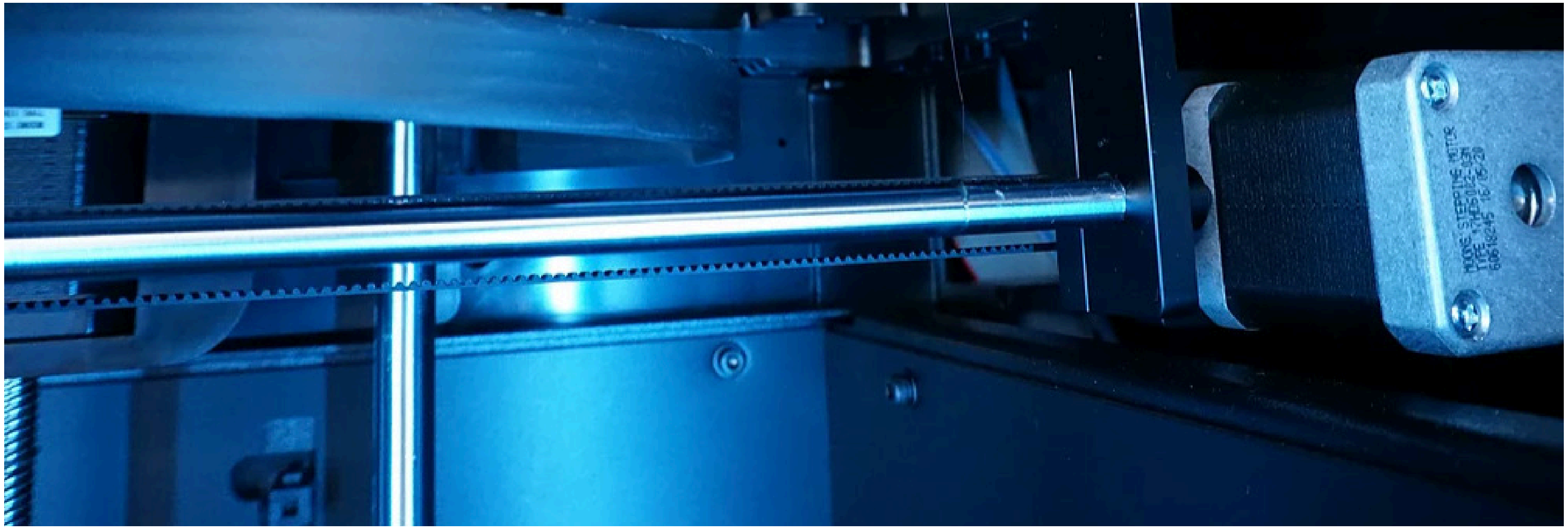
Troubleshooting and general maintenance is an important part of operating a 3D printer. In this section we'll cover some common troubleshooting issues and how to address them, as well as general maintenance tips to keep your printer running smoothly.

6.1 REGULAR MAINTENANCE

Keeping up with regular maintenance on your 3D printer will help keep it running for a long time. This includes checking mechanical components as well as looking to see if there is any damage to the nozzle or print bed. As the nozzle and bed are considered consumable parts they are not covered under Cubicon's warranty but may be covered under InkSmith's extended service plan.

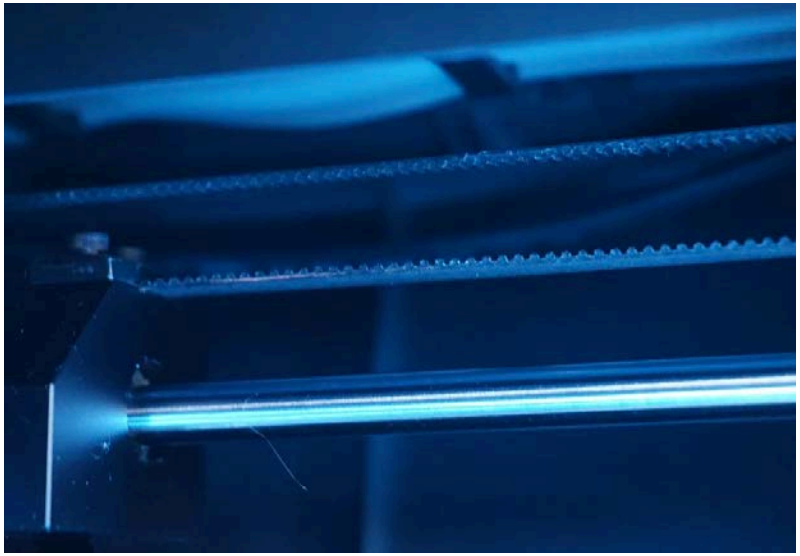
Check your rails, bearings and leadscrew

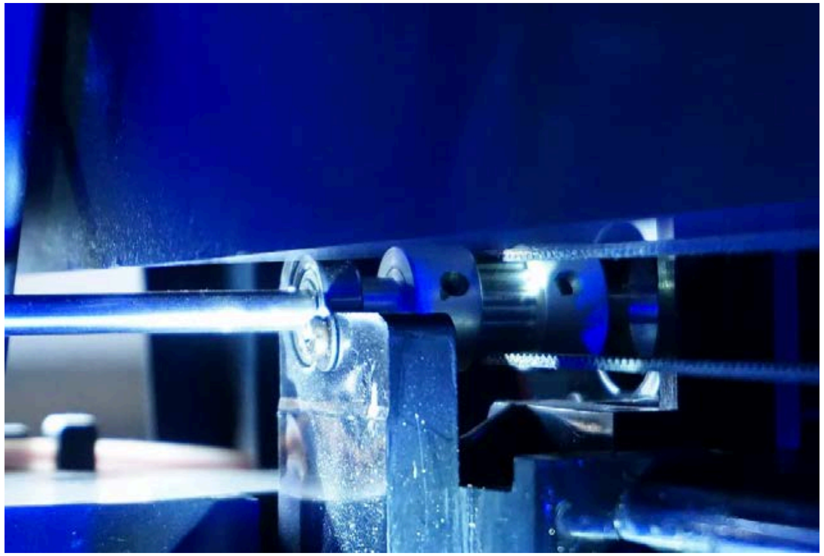
- Rails and linear bearings need to be greased in order to keep movement smooth and quiet. Multipurpose or almost any automotive grease or lubrication oil will work just fine. Just make sure it is not too thick. Avoid using WD40 as this is not intended to be used as a bearing lubricant and can cause bearings to seize.
- All of these parts are greased from the factory but over time this can wear away or become contaminated and new grease will need to be applied.



Check your belts and motor couplers

- If you're noticing a rippling effect in the z-axis it is likely that you have a disconnect between your motor and the axis that you're driving. This will be either loose belts or a loose motor coupler.
- A loose belt will result in large amounts of backlash when changing direction leading to corners that aren't sharp. From factory the belts are tight, but in the event that they become too loose for prints to work properly you will have to tighten them. Loosen off the plate holding the belt in place and pull the belt tight by hand. Tighten the plate back in place. The belt doesn't have to be extremely tight to give good prints and when plucked the free side of the belt should give a low bass note.
- A loose coupler will lead to poor prints that ripple back and forth in the affected direction. Make sure these are held rigidly to the motor shaft by tightening the set screws.

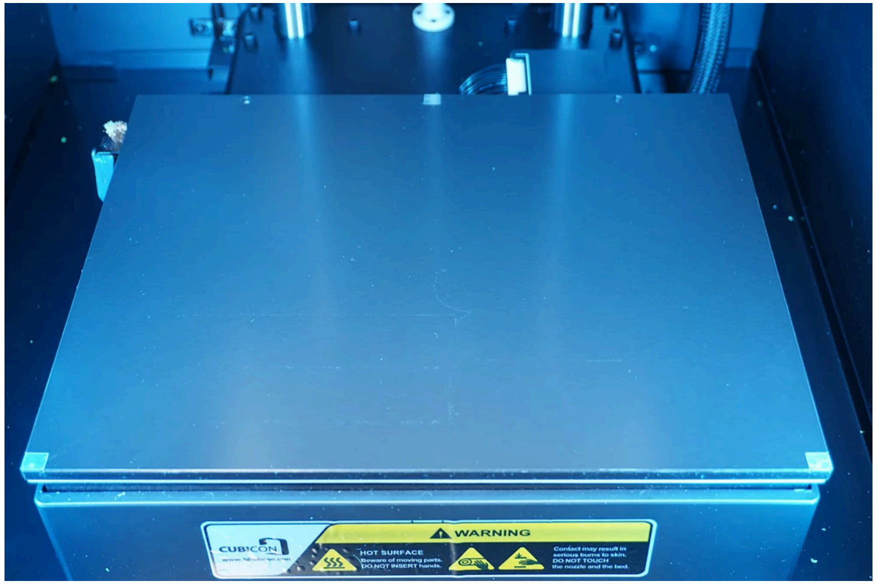




6.2 REMOVING A STUCK PRINT

The specialty coated bed on the Cubicon 3D Printers is great for providing a high quality finish to the bottom of your print bed. Under normal conditions, once the bed is cooled your part should just pop off. If your prints aren't detaching from the bed there are some methods that will help remove them.

- Wait for the bed to cool down sufficiently. The bed can be damaged if you are removing the print by force. The temperature at which the objects can be safely removed is dependant on the filament used and the model.
- If the bed has been allowed to cool to room temperature and the print has not detached then carefully push a flat object under the bottom of the printed object to take it off. You must be careful to not damage the coating of the bed.
- Any residue that remains must be fully removed with acetone prior to printing. Failure to do so may result in the print sticking to hard, a damaged bed or a damaged nozzle.
- Remove all acetone prior to printing by wiling with a wet cloth. Excess acetone can cause prints to stick too hard to the bed.
- If the coated bed becomes damaged it must be replaced.

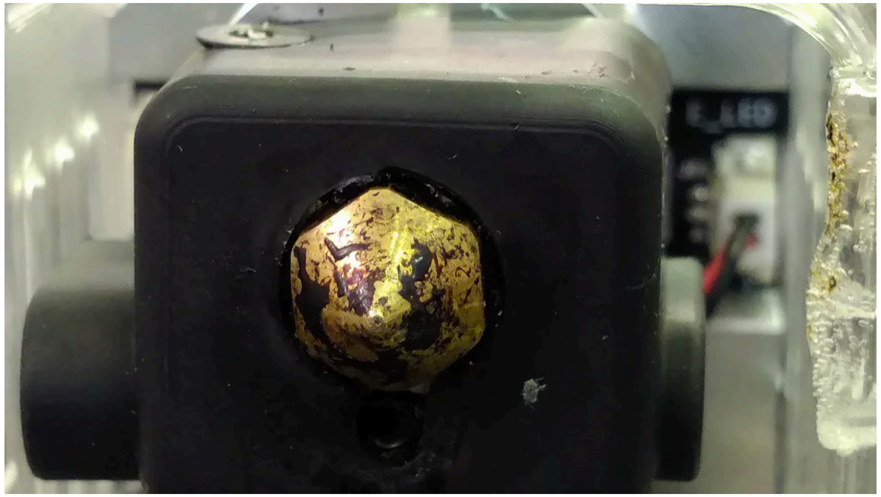


6.3 CLEANING THE NOZZLE

During printing melted filament can get caught on the outside of your nozzle. This can get caught on your print and affect print quality by causing blobs or discolouration. The outside of the nozzle should be cleaned in between prints.

1. Heat the nozzle to the recommended printing temperature of your loaded material.
2. Remove the residue with the tweezers or wire brush included in the Cubicon Tool Kit or with a cotton cloth. Be careful as the nozzle is very hot. Be careful to not damage the tip of the nozzle
3. If there is substantial build up that can not be removed completely, use the above method to remove as much as possible. Cool down the nozzle, shut down the printer and remove the extruder assembly. Wet a cotton cloth with acetone and wipe the nozzle.

Note: do not soak the entire head in acetone as the electronics and silicone cover can be damaged. Acetone should only be used with sufficient ventilation.



6.4 FIXING A CLOGGED NOZZLE

A clogged nozzle can be the result of many things but is typically the fault of the filament that is run through the machine. Filament that has inconsistent diameter (too large) can get lodged between the drive gear and the hotend. Dust, contaminants or even particles intended to be part of the filaments (such as wood or metal fill) can build up in the nozzle causing it to clog.

You'll know that your nozzle had become clogged when the filament being extruded is too thin, curling to the side, non-uniform or not extruding at all.

Clearing your nozzle

1. Heat your nozzle to the recommended printing temperature for the material being used.
2. In the menu select Load Filament. If the filament does not load properly and is still facing issues increase the temperature in 5° increments and try again.

Note: For PLA do not exceed 225°C as this will cause the PLA in the nozzle to carbonize further blocking the nozzle.

If this does not work then the clog is too severe to be removed by simple loading. You will need to use the Nozzle Management Pin that is included in your Cubicon Tool Kit.

Using the Nozzle Management Pin

1. Remove the filament through the unloading menu.
2. Turn off the power
3. Remove the extruder. Be careful as the nozzle is very hot. It is best to wear a heat resistant glove while doing this.
4. While the nozzle is still hot, press the Nozzle Management Pin down the extruder. Filament should come out the nozzle. Do not press too hard to avoid damaging the nozzle internally. Any damage caused by use of the nozzle management pin will not be covered by warranty.

6.5 COMMON PRINT FAILURES

No matter how experienced or inexperienced you are, 3D printing failures are bound to happen, it just comes with the territory. Some fails will come as a result of trial and error, machine error or sometimes just plain-old human-error. Either way, print failures should be taken as a learning opportunity in order to prevent the same mistake from happening again. Here are some of the most common problems that lead to failed prints and how to prevent them:

Problem: Print not sticking to the bed
Reason: One of the most common reasons that prints fail to stick to the printing bed is because the bed or extruder is not set to the correct temperature. When the correct temperatures are not set, the plastic filament will not be able to stick to the bed and the print will slide around with the extruder. If this problem goes unnoticed it can lead to major clogging issues and ultimately a failed print.

Problem: Under extrusion
Reason: Under extrusion or 'ghost printing' is a term used to describe a print that has very thin or weak layers. This scenario most often happens when the extruder is no longer able to extrude the proper amount of filament for each layer. This can occur when the print speed is too fast for the layer thickness or because the filament spool has become tangled and the printer is not able to pull in filament.

Problem: Poor print quality or ugly print
Reason: A print can come out under quality for a few different reasons. One reason is that there is a lack of support material. When there is a lack of support material, any overhangs or bridging can result in drooping or saggy filament layers. Another reason for poor print quality could be either of the previous problems listed above.

Problem: Print stops before completion
Reason: One common reason that a print will stop before completion is a result of the slicing process. During the slicing process, resizing a print can sometimes cause the print to have 0 wall thickness. As a result, the printer will stop when it reaches this point.


INKSMITH
IMMERSIVE EDUCATION SOLTUIONS
FOR 21ST CENTURY LEARNING
www.inksmith.ca
tech@inksmith.ca