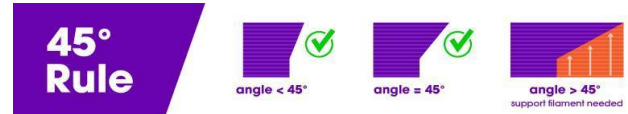


DESIGN TIP #1: DESIGN TO AVOID SUPPORTS

Supports are an extra material that acts as “scaffolding” to hold a design in place if there is nothing beneath it to build on. Depending on your design, supports may be necessary to prevent filament from drooping when it’s printed. However, supports can be difficult to remove and will add time to your print job. It is recommended to avoid them as much as possible. As a general rule, only use them for objects with an overhang beyond 45°, as without the support your object might sag and break.



How to Reduce the Need for Supports?

The **YHT** Rule- A good way to remember what shapes are safe to design without supports and which to avoid.



Anything in a “Y” shape is safe to print without support because it’s a gradual slope which still has enough material beneath it to keep it from drooping. Even though the arms of the “Y” are outstretched, because they extend at 45 degrees or less, they do not require support.



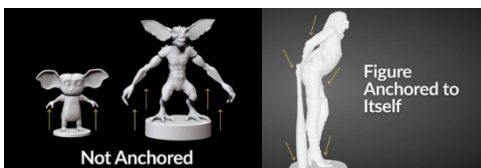
Designs that take the form of an “H”, where the middle overhang connects to either side is called bridging. Bridging allows material to be printed without support and with minimal or any sagging. However, if a bridge is over 5 mm long support generally is required to give an accurate surface finish. For this example, the center bridge is over 5mm and support was needed.



Anything with a “T” shaped overhang will not have any support and will almost certainly create errors in your print. There is nothing for the outer arms to be printed on and the material will just fall down without support for the arms of the letter.

Split the Model into Multiple Parts So They Can Each Print Flat

After you print the parts, you can bond them with an adhesive like cyanoacrylate (super glue) or a solvent. This can be especially useful for complex designs and larger prints.

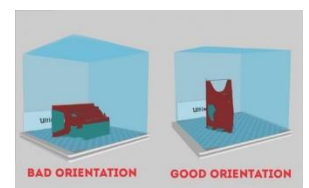


Anchor Your Print

Anchor parts of your design that would go beyond 45 degrees and attach them to a different part of the model to avoid overhang.

Orient Your Model Properly

When an object is imported into a 3D conversion software, you need to ensure that it is sitting directly on the build plate. Supports can sometimes be reduced simply by rotating your design into the appropriate orientation for printing. For example, in the case of the ‘T’ print above, the need for supports can be completely removed by laying it flat on its back.



Dissolvable Support Option

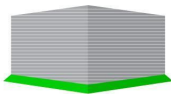
On finely tuned printers like our Ultimaker 3 at Aaniin Library with two print heads, the support material can be printed with a dissolvable material that does not tear away from the part but instead dissolves away in a manner that does not affect the main material of the printed model. This will result in a better surface finish

where the support is in contact with the main material but can be an expensive and a time-consuming solution.

DESIGN TIP #2: UTILIZE RAFTS, BRIMS AND SKIRTS

Rafts, brims (sidewalks) and skirts are an easy way to get better results when you print. One of the easiest ways to prevent poor bed/build plate adhesion, along with print warping, is to utilize rafts, brims and skirts.

RAFT

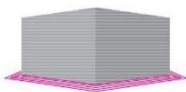


Rafts are a throwaway horizontal surface that sits under your object and extends a specific distance away from the sides of your object. Due to its larger surface area, the raft



greatly improves adhesion and reduces warping in the object itself, since the edges of the raft are much more likely to warp, leaving the bottom of the object level and square. A raft can also be used to provide greater stability to objects that have small footprints or objects that may tend to be top heavy or larger.

BRIM

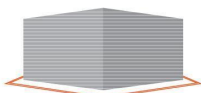


Brims (also known as sidewalks) are a close relative of rafts. A brim only extends outward from the perimeter of an object (think of the brim of a hat), it has no contact with the



object's underside. For smaller or more delicate objects, a brim may be preferable to a raft due to the fact that its only contact with an object is along the outside edge in a very thin layer. Support structures, that tend to be narrower and have a low surface area on the bed, can easily pop off the bed, so it's wise to print supports with Brims.

SKIRT



A skirt is nothing more than a brim that doesn't touch the edges of the object that you're printing. Skirts serve a useful purpose because they help prime the extruder and



establish a smooth flow of filament. Most 3D print slicing software will automatically add a skirt to the beginning of each print. Skirts are usually just a couple of layers thick and can be used in multiple ways (i.e. preview of how a 3D printer is working/how material is flowing, thermal barriers etc.)

DESIGN TIP #3: REDUCE PRINT TIME

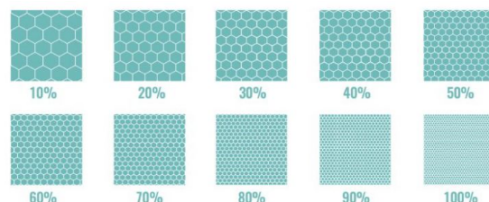
Orientation of Parts

When printing multiple parts, work to arrange them on the platform in a configuration that limits the stringing between them and limits that stringing to the corners. This can both reduce the time it takes to print and the time it takes to cleanup your print.



Infill Density

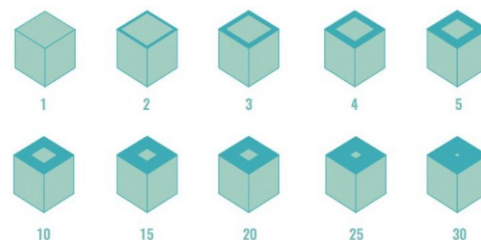
Infill is the material that comprises the interior of the 3D object between the outside shell/walls. Infill settings (measured in %) will affect print time and part strength. The higher the infill % the longer the print time will be. If you're creating an item for display, 10-20% infill is recommended. If you need something that is going to be more functional and sturdy, 75-100% infill is more appropriate. The secret to a successful use of infill is to find the optimal point where



sufficient strength is obtained for an object's designed purpose, with both cost and time being kept within acceptable parameters.

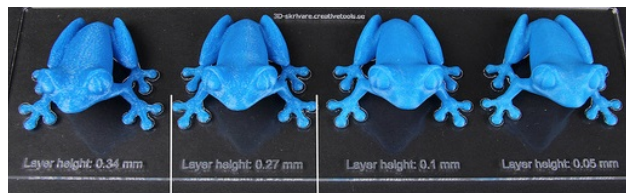
Shell Thickness

Shells refers to the number of times the outer walls of the design are traced by the 3D printer before starting the inner sections of your design. This means that shell thickness is intimately tied to infill percentage. When you increase the shell thickness of an object, you are also increasing its strength (capable of handling stress without the need for increasing the 3D print infill density). However, every increase in shell thickness will drive up print time. 3D print slicing software, such as Cura and Dremel, normally have a default shell thickness of 0.8-1mm. There shouldn't be any reason to change this for decorative prints however, if you are printing something that will need more durability, you may want to increase shell thickness.



Layer Height/Resolution

The height of each filament layer in your print directly affects the overall resolution of the finished print job. Prints made with thinner layers will create more detailed prints with a smoother surface however, it takes more time to print something, since there will be more layers that make up your object. Keep this in mind when designing your 3D print- If you want to print something with intricate details, you will get the best print with a thinner layer height. Prototyping? You can raise your layer height for a faster print job.



DESIGN TIP #4: EMBRACE FAILURE



There's a lot of failure in your future and that's okay as this is a necessary step you need to embrace during your 3D printing learning journey. Translating an object from a three dimensional model to real life is complicated. Even objects created by experienced designers fail. Even after using 3D Printers for quite a while, experienced 3D designers admit to a 70% success rate being a very good thing for users of any skill level. Designers often create, fail, edit and create again until they get it right. Try, fail, learn, and repeat is your best method to get good at this process!

RESOURCES USED

- How to Use 3D Print Infill Settings - Increase Strength, Save Filament <https://rigid.ink/blogs/news/optimum-infill>
- How to Print Faster <https://ultimaker.com/en/resources/21918-print-faster>
- 3D Slicer Settings for Beginners – 8 Things You Need to Know <https://pinshape.com/blog/3d-slicer-settings-5-things-you-need-to-know-about-3d-printing-software/>
- Rafts, Skirts and Brims! <https://www.simplify3d.com/support/articles/rafts-skirts-and-brims/>
- How to Print Overhangs, Bridges and Exceeding the 45° Rule <https://rigid.ink/blogs/news/how-to-print-overhangs-bridges-and-exceeding-the-45-rule>
- What is a 3D Printing Raft, Brim or Skirt - Why and How to Use Them <https://rigid.ink/blogs/news/3d-printed-raft-brim-skirt>
- The Innovation Station: Tips for Printing 3D Parts <https://innovationstation.utexas.edu/tipdesign/>
- Make: Top Ten Tips for Designing to Print <http://makezine.com/2013/12/11/top-ten-tipsdesigning-models-for-3d-printing/>
- Print Quality Troubleshooting Guide <https://www.simplify3d.com/support/print-qualitytroubleshooting/>
- 3D Printing Problems Overview <https://all3dp.com/common-3d-printing-problems-3d-printertroubleshooting-guide/>