Designing for Big Area Additive Manufacturing

Alex Roschli[1]*, Katherine T. Gaul[1], Alex M. Boulger[1], Brian K. Post[1], Phillip C. Chesser[1], Lonnie J. Love[1], and Fletcher Blue[1]

[1]Manufacturing Demonstration Facility
Oak Ridge National Laboratory
2350 Cherahala Blvd
Knoxville, TN 37932

ABSTRACT

Additive manufacturing (AM), more commonly referred to as 3D printing, is revolutionizing the manufacturing industry. With any new technology comes new rules and guidelines for the optimal use of said technology. Big Area Additive Manufacturing (BAAM), developed by Cincinnati Incorporated and Oak Ridge National Laboratory's Manufacturing Demonstration Facility, requires a host of new design parameters compared to small-scale 3D printing to create large-scale parts. However, BAAM also creates new possibilities in material testing and various applications in the manufacturing industry. Most of the design constraints of small-scale polymer 3D printers still apply to BAAM. Beyond those constraints, new rules and limitations exist because BAAM's large-scale system significantly changes the thermal properties associated with small-scale AM. This work details both physical and software-related design considerations for additive manufacturing. After reading this guide, one will have a better understanding of slicing software's capabilities and limitations, different physical characteristics of design and how to apply them appropriately for AM, and how to take the inherent nature of AM into consideration during the design process.

Keywords: Big Area Additive Manufacturing; 3D printing; additive manufacturing; design; tutorial;

1. INTRODUCTION

Big Area Additive Manufacturing (BAAM), developed by Cincinnati Incorporated and Oak Ridge National Laboratory's Manufacturing Demonstration Facility, is a large-scale Fused Filament Fabrication (FFF) system. FFF is the process of building a part by extruding material in layers. FFF started with smaller machines (approximately 8"x8"x8"), but BAAM was the first foray into building large parts with FFF. Because of this, BAAM requires a host of new design parameters compared to small-scale printing to create large-scale, additively manufactured (AM) parts. According to Gibson et al.[1], the unique capabilities of AM include: shape complexity, in that it is possible to build virtually any shape; hierarchical complexity, in that hierarchical multiscale structures can be designed and fabricated from the microstructure through geometric mesostructured to the part-scale macrostructure; material complexity, in that materials can be processed one player at a time; and functional complexity, in that fully functional assemblies and mechanisms can be fabricated directly using AM processes. Most of the design constraints of small-scale polymer 3D printers still apply to BAAM and will be outlined below. Beyond those

*Corresponding author: Alex Roschli: roschliac@ornl.gov

constraints, new design rules and limitations must be taken into consideration because of the nature of the BAAM process.

BAAM is a gantry-driven, extrusion-based system. It functions by depositing polymer pellets into a heated barrel that houses a single screw. The heated barrel melts the pellet feedstock, and the screw pushes the melted material through a nozzle resulting in extrusion. Small-scale AM systems typically use a plastic filament as feedstock, but the use of polymer pellets and a single screw have enabled much faster and cost-effective printing capabilities on the BAAM system. Carbon-fiber reinforced acrylonitrile butadiene styrene (CFABS) pellets have been found to work well as feedstock for fabricating large AM parts, and the majority of the parts discussed in this work were fabricated using CFABS.

Additionally, the BAAM system enables the fabrication of large-scale parts because it eliminates the use of an oven. According to Love et al. [2], although building parts in an oven minimizes thermal distortion, it is very difficult to accurately control temperatures over large areas, which has limited the build volumes of the first 3D printing systems. This is not an issue for BAAM; therefore, possible part sizes have increased significantly. As stated, creating larger parts with a new technique changes the way one approaches design; the authors aim to outline those changes and present possible solutions and alternatives.

2. PREPARING A PRINT – FROM DESIGN TO SLICE

It is important to understand the process in which a computer-aided design (CAD) model ultimately becomes G-Code. The software used in this process, called slicing, has many capabilities and limitations that influence design considerations. Section 1 aims to explain slicing to make designing for additive manufacturing more straightforward.

2.1 Design

The first step in preparing a print is to design the part. This is typically done with CAD software such as Solidworks, Fusion 360, or Rhino. In this software, the user can create the part to be printed with many tools that allow for creating complex structures as well as assemblies of multiple parts. A raw CAD file stores all the original data from the part design including curvature such as arcs and splines (Figure 1). However, to slice the part, which is the next step, the part must be exported from the CAD software as an STL file. A stereolithography (STL) file is a stereolithography file that uses triangles to approximate the surfaces of the CAD file (Figure 1). This means that all complex curvature data is lost and replaced with straight lines as triangles. This process is illustrated in Figure 2.

2.2 Slicing

Once the part has been designed and exported as an STL, the file can be loaded into a slicing program to create G-Code. A slicing program is a standalone program that divides the STL file into layers, known as slicing, and then creates tool paths accordingly, which are stored as G-Code. G-Code is the output of the slicing software that is loaded into a 3D printer to instruct it how to build the part.

Slicing works by intersecting a horizontal plane with the STL file and continuously moving the plane vertically by a single layer height until the entire part has been sliced. A horizontal plane intersecting an STL file can be seen in Figure 3; the resulting cross section from that intersection viewed from the top can be seen in Figure 4. Each time the plane meets the edge of a triangle in the STL file, a point is created. All the points together form a polygon, or polygons, that act as the boundary, or boundaries, for that layer. From there, the slicer fits toolpaths to each polygon to form the layer. The size of the toolpaths and how they are generated is determined based on the user settings.

Perimeter moves are the first to be generated during the slicing process, and they are created by offsetting the boundary polygon inward by one bead width. Following the perimeters are the insets, which are also created by offsetting the original polygon, minus the perimeter region, inward by one bead width. After all perimeters and insets are generated, the remaining area, which sometimes consists of multiple regions, is designated as infill. The infill region can be filled using various methods. One of which is to keep offsetting the boundary polygon inward until nothing is left; This is called concentric infill. Other infill patterns rely on overlaying a grid of points on the infill region and clipping the grid of points to fit. The points can then be connected to form different geometric patterns such as parallel lines or honeycombs. An example of one layer of a part with perimeters, insets, and infill is shown in Figure 5.

The process of generating toolpaths happens for every single layer until there are no layers left. Once all the paths have been created, they are translated into G-Code. The G-Code includes commands for the speed of the machine, positions for each axis, and speed of the extruder, to name a few. G-Code is stored as a text file and exported from the slicer so that it can be loaded into the printer. An example of a G-Code text file is shown in Figure 6.

2.3 Software Limitations

When using CAD to design a model for BAAM, it is important to keep the desired bead, or extruded material, width in mind. For the best results, the CAD model must not contain sections less than two bead widths thick. Sections thinner than approximately four inches should be designed as an exact multiple of the bead width. Larger sections have more room for error because infill will be used to fill the area between perimeters. The slicing software uses operator-input bead width settings to fit toolpaths into the constraints of the CAD file. This means a 0.3-inch diameter nozzle has an input bead width of 0.34 inches. Therefore, if the desired wall thickness is 4 inches, the wall thickness must be changed to either 3.74 inches or 4.08 inches to account for the input bead width value. If the wall thickness was kept at 4 inches, the software would place six closed-loop paths in the space, and the center most beads would overlap by 0.08 inches to keep the part at 4 inches. This overlap in the middle can cause overfilling that will build up layer after layer potentially causing the part to fail.

3. GENERAL 3D PRINTING DESIGN GUIDELINES

Section 1 provided an overview of how software is used to create physical objects. Software capabilities and limitations must be considered when designing. However, there are physical design considerations to factor in as well. The sections below detail the most important design consideration for additive manufacturing.

3.1 Bridging

Bridging occurs during a print when the machine prints a horizontal overhang between two towers (Figure 7). In contrast with a traditional overhang, which protrudes into free space, bridges are connected to the body of the part at both ends. The possible size of the gap under a bridge is heavily dependent on the printer, material, and environment. Small printers can bridge gaps of several inches because the large surface area-to-volume ratio of the extruded plastic allows the bead to solidify almost instantly. Small printers often include part cooling fans that are directed near the nozzle to help cool the material immediately after extrusion, which also helps with bridging. However, on the BAAM system, the surface area-to-volume ratio is vastly smaller causing the plastic to cool much slower. Therefore, as hot plastic is extruded over a large gap, it will sag and break off due to the increased weight and temperature. Material type and environment temperature will also influence bridging. Different materials cool at different rates, so a material that cools quickly is more likely to bridge than a material that cools slowly. A warm environment, such as the environment found in BAAM, keeps each bead of plastic at a higher temperature and makes it more likely to sag and fail during bridging.

To test the maximum bridging distance, a part is designed with a V-shaped cut in the bottom surface (Figure 8). The part is built up a few layers tall with this cut before starting to build over it. Starting at the narrow end of the V, successive beads are continually laid down spanning the gap until it fails. A failure is determined as a bead that breaks off or sags down more than ½ of a layer height. The maximum distance for successful birding using a 0.3" in diameter nozzle was measured at 1.85"; the maximum distance for successful bridging using a 0.2" nozzle was measure at 2.25". An example of an additively manufactured bridge test article is shown in Figure 8.

3.2 Cavities

A cavity is a hollow space within a solid body. Cavities are feasible with AM, but whether a cavity is printable using BAAM depends on geometry and design of the part. Assuming a user wants a cavity on the inside of a part, its geometry must be self-supporting to avoid the requirement of support material. If the cavity is necessary but not self-supporting, the gap must be covered by bridging, or support material must be used to support the roof of the cavity. If support material is used, an opening must be left in the cavity so that the support material can be removed once the print is complete. The hole must be large enough to access all the support material with tools to remove it. Support material should not be used if the cavity is completely enclosed by the part. Examples of cavities are shown in Figures 9 and 10.

In the instance of the part shown above in Figure 10, the cavity would not be printable on its own. The roof of the cavity would not be supported and would ultimately droop into the part and fail. Support material could be placed in the cavity, but it would be totally enclosed and never removable without cutting the object. One solution is to grow the internal walls toward the center of the part at a 45-degree angle so that they come together just in time to support the roof (Figure 11). This will still leave a cavity, but prevents the roof from caving in.

3.3 Improving Strength in the Z-Direction

There is a strength reduction across most AM processes in the Z-direction because of the inherent lag between layer deposition and the cooling that occurs during that time. Layers are deposited in the X- and Y-directions causing them to have the highest strength. The X- and Y-directions have the highest strength because layers are formed with continuous extrusions and the plastic within a layer is often molten; this bonds the plastic before cooling occurs. The bond between each layer, which forms the Z-direction, is not as strong because a new layer of molten plastic is extruded onto a solidifying layer of plastic. The bond between a molten layer and a solidified layer is much weaker than the bond between molten beads within the same layer. X- and Y-strength can be further improved by adding fiber reinforcement, such as carbon fiber, to the neat material. The fibers align during printing and increase strength. The addition of fiber reinforcement does not improve Z-strength because fiber does not improve the adhesion between a molten layer of plastic and a solid layer of plastic. Therefore, a part must be designed in a way that it will not see any loading in line with the layer bonds that could separate the layers. Figure 12 illustrates this concept.

3.4 Delamination

Delamination occurs when the layers of a part separate from each other or when the bottom layer of a part separates from the build surface. Both types of separation cause print failure and run the risk of damaging the machine. Temperature is the root cause of both failures.

Layers of the part separating from one another is the result of inconsistent cooling of the part or too much cooling of the part. The problem occurs because after a hot layer of plastic is deposited, it begins to cool, and when plastic cools, it contracts. How much the plastic contracts during cooling (or how much it expands when heated) is related to each material's coefficient of thermal expansion. As the layer is cooling, a new layer of hot plastic is deposited on top of the previous layer. This process repeats itself for however many layers tall a part is. As a part gets taller, the bottom layers continually cool to room temperature, while the top layers stay hot from the extrusion process. The cool bottom of the part contracts but is constrained by the expanded hot top, which results in curling.

This issue is more apparent on BAAM because the large build volume of the machine leads to longer layer times and taller parts, which results in a larger thermal gradient across the part. The areas that are most likely to delaminate are the areas that cool the quickest. Thin wall sections with low volume and high surface area cool quickest, such as the fenders in Figure 13. Thicker areas hold heat much longer and are less likely to delaminate. The best way to avoid delamination is to minimize the layer time by printing faster, printing less parts at once, or orienting the part to minimize cross sectional area. However, there is a limit to how fast you can print. This is discussed in more detail in section 5.

The bottom layer of the part separating from the build surface is the result of thermal expansion differences between the part and the substrate. If the substrate, or the surface the part is printed on, has a much larger thermal conductivity than that of the printed material, then the substrate will rapidly pull heat from the printed material. If the difference is large enough, such as when printing ABS on cold aluminum, then the printed part will not stick to the build surface and will instantly warp. If an aluminum substrate is heated, then the part will stick but may delaminate if the aluminum cools or if the plastic becomes too cool. The bond between substrate and part can be improved by applying a tacky glue-like substance on top of the substrate before printing. Another solution is to have a heated build plate, which decreases the difference between the substrate and the part. The BAAM system uses a heated plate covered with a build sheet made from ABS that is held in place by vacuum.

The final step of printing is removing the part from the machine. When taking the part off the build sheet, there is risk of warping. The risk is dependent on the geometry and temperature of the part. Long flat sections, like beams or plates, are at most risk of warping. The best solution is to leave the part on the bed and allow it to slowly cool down uniformly while being held in place by vacuumed down build sheet. This ensures that the bottom of the part is not introduced to a cold surface that would cause the part to go into thermal shock, which causes stress in the part and results in curling.

3.5 Surface Finish or Print Resolution

Fused Filament Fabrication (FFF), the process of building a part by layering extruded material, creates a layered surface finish on parts regardless of orientation. This is because the beads that are extruded in a cylindrical shape due to the circular nozzle. Each layer is another cylindrical bead stacked on top of the previous one. Larger nozzles, like those used on BAAM, often mean larger layer heights meaning larger cylinders creating an even more noticeable corduroy-like surface on the side of the part. Small printers use smaller nozzles which results in smaller cylinders and thus a smoother finish. These rougher finishes seen on BAAM can be sanded smooth by hand, finished with a CNC router, or smoothed with chemicals such as acetone. The in-plane surface finish, seen on the top of the part as opposed to the sides, is often much smoother and only shows variations where there is over or underfilling. Overfilling causes ridges to build up and underfilling leaves voids or holes.

To measure the surface finish, two parts were printed. The first was a tall wall that was cut into a long vertical strip for measuring the layer to layer surface finish (Figure 14). The second was an 8-bead wall where the top surface was cut off for measuring the in-plane surface finish (Figure 15). Both parts were printed from CF-ABS using a 0.3” diameter nozzle. The first piece had an average deviation of 0.03025” with max deviation of 0.059”. The top piece had a lower average deviation of 0.02864” with a max deviation of 0.18”.

3.6 Assemblies and Tolerances

Assemblies are multiple pieces that are put together to form a single part. Assemblies can be printed as separate pieces and later joined, or they can be printed as one part with pieces already joined together. Small-scale 3D printers can easily print assemblies with the parts already joined because assemblies require tight tolerances, which is achievable on small-scale machines that have a high print resolution. BAAM is meant to print quickly, which is achieved by sacrificing print resolution. The low resolution of BAAM is not currently suitable for printing assemblies with tight tolerances. For BAAM, assemblies must be designed and printed as separate pieces. When making 3D printed parts that must be assembled post-fabrication, tolerancing the design is typically rounded up one bead width. This slight oversizing of the part allows for each part to be machined to the exact tolerances of the assembly.

3.7 Overhang Angle

An overhang angle (Figure 16) is the angle between the part being printed and a parallel line in plane with the build surface, which is also called being measured from the horizontal plane. For example, the overhang angle of a square box being printed would be 90 degrees. Designing without support material introduces a new design challenge. Parts with an overhang angle less than 45 degrees from the horizontal build surface plane typically cannot be printed. They must either be redesigned, or support material must be used. Generally, an overhang angle less than 45 degrees causes the layer being deposited to not make enough contact with the previous layer to bond properly. This causes that layer to be unsupported in the Z-direction.

Unsupported material can fall off or sag over the edge, leading to a failed build or loss of dimensional accuracy. With BAAM, beads of plastic are thicker and take more time to cool. A bead that stays hot for too long remains malleable and is prone to sagging or collapsing if an additional layer is printed on top of the still malleable layer too quickly. To achieve steep overhangs, the layer must be given sufficient time to cool and solidify before printing on top of it. Through a topology optimization study performed by Gaynor and Guest [3], they found that by designing components and structures whose features rise in the build direction at an angle that is greater than a process-specific minimum allowable self-supporting angle, support material is eliminated from the design and fabrication process, which saves material, build time, and post-processing time.

According to a study performed by Additive Engineering Solutions [4], the starting and stopping point of the extrusion nozzle after each layer can greatly impact overhang angle printing. During the first test performed in this study, four circular parts were printed with overhang angles of 25, 30, 35, and 40 degrees from vertical. Surprisingly, the 25- and 40-degree overhang angles printed successfully, but the 30- and 35-degree angle parts failed. In the first test of the study, the start/stop location was set to "random". This means that the starting and stopping point of each bead printed was different after every layer. This also led to inconsistent layer time across the part, which impacts bead adherence.

In the second test of the study, the start/stop location was set to "optimized," which means the extruder starts and stops in the same X- and Y-locations on every layer. This allows for more consistent path planning and layer time. This also causes material to "pool" slightly at the start/stop location creating a seam. The seam would normally be viewed as a print defect. However, the extra material may have created additional surface area for each layer to adhere to enable successful overhang angles. All other parameters of the test were kept the same as the first test, and overhang angles 25, 30, 35, and 40 all printed successfully. Therefore, it is suggested that the extrusion nozzle start/stop location be in the same location layer to layer.

To further test the overhang angle capabilities of BAAM, and compare convex and concave overhangs, a new test was devised. For the test, the overhang angle started at 90 degrees, a vertical wall, and gradually started building out towards being perfectly horizonal, or parallel with the build plate. The path for the layer went around a series of convex and concave circles with straight walls in between. The changing convex and concave was done to demonstrate the different angle capabilities based on whether the bead was being pulled in, convex, or pulled out, concave. The concave tests (Figure 17) were run five times each and the convex twice. The failures angles from the horizontal can be seen below in Table 1.

Table 1 – Overhang failure angle tests. X represents a data point not collected.

		Failure Angle from Horizontal (Degrees)				
	Circle Diameter (in)	**Test 1**	**Test 2**	**Test 3**	**Test 4**	**Test 5**
Concave	3	43	43	40	37	37
	3.5	43	40	40	37	37
	4	43	40	40	37	39
	4.5	49	43	40	37	37
	5	52	46	40	37	37
Convex	3	X	X	X	29	35
	3.5	X	X	X	31	35
	4	X	X	X	33	35
	4.5	X	X	X	33	35
	5	X	X	X	33	35

4. BEAD WIDTH CONSTRAINTS

The BAAM extruder is capable of printing at multiple bead widths. The bead width is typically determined by the nozzle diameter, but it can be varied by adjusting print or extruder speed. BAAM has four common nozzle sizes: 0.1", 0.2", 0.3", and 0.4". Once the bead is extruded onto the printer bed, the top surface of the bead is flattened by the tamper. The tamper (Figure 18) is a device used to level the top surface of the bead to maximize surface area to strengthen the bond of the next layer. The tamper also helps keep material in plane by spreading out areas of overfill. Table 2 shows the bead width and layer height for each standard nozzle size after being tamped.

Table 2 – Common nozzle diameters and their associated bead widths and heights

Nozzle Diameter (in)	Bead Width (in)	Bead Height (in)
0.1	0.11	0.05
0.2	0.22	0.1
0.3	0.34	0.15
0.4	0.5	0.2

4.1 Open Loop vs. Closed Loop

For a part to be printed as accurately as possible, it must be designed with the machine and slicer, or the software that divides a part into layers and tells the machine the tool paths to follow to print those layers, capabilities in mind. One such consideration is the requirement of closed-loop paths. All perimeter and inset moves created must be based on a closed-loop path. In other words, the extruder will start and stop at the same location (see Figure 19(b)). This means that a one bead thick extension of the part where the machine starts at one end and ends at the other is not possible (see Figure 19(a)). This section must be designed to be two bead widths thick so that a closed-loop path is created. The typical design process revolves around this two-bead width minimum thickness for all sections.

4.2 Hollow Shapes

To design hollow parts on the BAAM printer, it is important to understand the difference between inset, infill, and perimeter. Based on the slicer software, the perimeter bead is the first inset bead. An inset is an offset of the outer perimeter toward the center of the part. The infill is described as the volume filled in between the inner and outer perimeters. By adding more insets to a part, less volume is needed to fill with infill. Figure 20 illustrates a hollow part with perimeters, insets, and infill.

In Figure 20(a), the blue, gray, and green lines can all be defined as insets. However, the blue and green lines receive the distinction of 'perimeter' because they are the inner- and outer-most beads. In Figure 20(b), the blue, gray, and green lines are still classified as insets. The blue and green lines still receive the distinction of 'perimeter' because they are the inner- and outer-most beads. However, the orange grid represents infill, which is an area of a part that is quickly filled in, normally following a grid pattern. The infill is completed quickly because it will not be seen and mainly functions to fill a void.

When printing a hollow shape, it is easiest to design the part as a solid in CAD, and then allow the slicer to do the hollowing by adjusting the infill and skins settings. Skins are the number of solid infill layers on the top and bottom of the part. They completely cover the exterior of a part just as skin covers the exterior of your body. This will give the slicer software a more accurate way of creating the desired wall thickness. When slicing the solid part to be hollow, it is important to turn off infill and skins in the slicer settings. This will allow the part to be hollow, and the desired perimeter and inset values can be chosen. This will also ensure there are no gaps between the inner and outer perimeters. A benefit of designing hollow parts in this fashion is that a one-bead outer wall can be achieved, which is difficult as discussed in section 3.1.

If a hollow part needs to be constructed with only one perimeter (i.e. one bead width), one cannot design a part in CAD that is one-bead width thick because the software used to slice the part interprets parts with an inner and outer perimeter as described above. Since only a one-bead width thick perimeter exists in this example, both an inner and outer perimeter do not exist simultaneously. Therefore, the slicing software will delete the part entirely. It should be noted that even if one inset is selected, the slicing software will program the part to have a two-bead thick wall. This is because the part technically has an inner and an outer perimeter, meaning the inset count is one. Because of this feature, there is only one way to print the hollow part with a one-bead thick wall. To do this, a solid model of the part needs to be created in CAD. When choosing the options for the slicing software, one inset needs to be specified. Additionally, as mentioned above, infill and skins need to be turned off. With these settings, the slicer will remove the solid core and only print the outer perimeter, which yields the desired one-bead thick walled part because a solid part has no inner perimeter. An example of this is displayed in Figure 21.

5. DESIGNING WITHOUT SUPPORT MATERIAL

One of the most significant differences between small-scale and large-scale AM is the use of support material. Small-scale 3D printers have build volumes ranging from 11.6 x 7.6 x 6.5 inches (MakerBot's Replicator+) to 36 x 24 x 36 inches (Stratasys' Fortus 900mc). When using small-scale machines, geometric constraints are not as much of an issue because of the use of support material. Support material is used to support portions of the design that could be affected by gravity, which can deform the part before it is complete. Generally, support material is removed from the part by breaking it away, or the support material is dissolved in a solvent bath. The significantly larger build volume of the BAAM (20 x 8 x 6 feet: X, Y, and Z) does not allow for the use of support material because the two traditional methods of removing support material become complicated by the size of the parts. Breakaway support material cannot be used with the BAAM process because the large extrusion size increases the surface area with which the support material bonds to the model. This makes the breakaway process tedious and will leave large imperfections on the part. A solvent bath is also an unacceptable solution because the bath would need to be near the size of a swimming pool to fit the large BAAM parts. Because of the inability to print support material, the designer must take this into account when modeling parts.

To design a part without support material, the part should be designed based on the capabilities of the model material. Bridging gaps should be eliminated or minimized with overhang supports based on material capability as discussed in sections 2.1 and 2. 2. Holes in the side of a part should be left out because they will result in unsupported cavities; these holes can be drilled out after the printing process. Overhang angles should be increased so that they don't need support, or they may be removed and printed as separate pieces. Often the best solution to printing a large part that needs support is to print the parts that need support separately and bond the pieces together after printing.

6. LAYER TIME LIMITATIONS

The thermal properties of a material dictate layer time limitations and some aspects of design. As explored by Dinwiddie et al. [5], the use of thermography to examine printed parts allows for the detection of thermal gradients within the part, which can lead to thermal stresses and geometric distortions. The detection of horizontal temperature gradients on printed parts motivates designers to consider more carefully how each layer is deposited. If a layer is not given sufficient time to cool, it will not be rigid enough to support the next layer during printing. This is called the minimum layer time. The hot plastic will deform and be pushed outward, which causes a loss of dimensional accuracy and often a failure of the part as subsequent layers will sag and fall off (see Figure 22). It is important to notice in Figure 22 that only a section of this part failed. In this specific incident, two walls were very close to each other, which stored heat in that location. Because of this, the two walls could not cool down quickly enough causing the part to sag. This is very common on small parts, and in most cases, the part will collapse when an uneven distribution of heat is displayed. A similar type of part failure is caused by using steep overhang angles (see Figure 23). As the angle increases, the layer gets smaller and less material is needed for that layer. This causes the layer time to decrease and, in turn, decreases cooling time. The part may not completely fail, but the surface will sag and ruin the flat surface.

A simple fix to any of these problems would be to print an additional part during the same build. The extra part will take up time, or it will slow down the speed of the printer, which allows each layer more cooling time. It is important to understand that there is a constant cooling rate for each material, and if this rate is not maintained throughout the build, the part will start to develop defects. Each material has its own cooling rate based on the specific rheology for that grade of polymer. Keeping the layer time in mind when getting ready to print is essential and can save a great deal of time in the long run.

An opposing issue occurs when a layer cools too much. A layer that has cooled too long will not allow the next layer to adhere properly. This is called the maximum layer time, and this issue is more complicated than overheating because it depends on different features of a layer and how a part cools as it is being printed. For a section with infill, becoming too cool is typically not an issue because the infill has a high density and holds heat longer. However, on large parts with a feature that is just an inner and outer perimeter, too much time between layer deposition causes too much cooling and results in the layers bonding improperly. The solution to this issue is typically a redesign of the part to optimize layer times.

To demonstrate the slow cooling time, bars were printed as shown in Figure 24. The bars varied in width from two beads wide to eight beads wide to see how heat was held in thicker parts. Each part was printed 40 layers tall, which results in parts approximately 6” tall. At the end of the print, a thermocouple was placed in the top center of the part, and the cooling time was measured. Data points were recorded for time to cool to table temperature (110C) and time to cool to ambient temperature (24C) (Table 3). As expected, the thicker parts took longer to cool.

Table 3 – Time for walls to cool with a varying number of beads

	2 Beads Table at 0C	**2 Beads** Table at 110C	**4 Beads** Table at 110C	**6 Beads** Table at 110C	**8 Beads** Table at 110C
Time to Cool to 110C	7:30	8:03	12:35	13:05	16:56
Time to Cool to 24C	2:07:39	N/A	N/A	N/A	N/A

7. MAXIMIZING THROUGHPUT

7.1 Path Dependent Optimization

Throughput refers to the processing speed of the machine or how much material can be printed in an hour (lb/hr). Generally, the BAAM’s throughput is approximately 100 lb/hr when printing with a 0.3” nozzle and carbon-fiber reinforced ABS, which is the most commonly used material on the BAAM system, currently. Although the machine is capable of 100 lb/hr, the throughput of an average part ranges between 50-65 lb/hr. This decrease is caused by the extruder accelerating and decelerating – a function of a part’s geometry. For example, to make a circle, the extruder makes tens or sometimes hundreds of moves, whereas a square can be printed in four moves. This discrepancy occurs because an STL surface is represented as triangles (as seen in Figure 1(b)) approximating the curvature. The number of moves the printer must make to print a curve is determined by the resolution of the STL. An STL with a higher resolution will have smaller triangles, which means more points are created during slicing.

Therefore, 90-degree edges should be used, and curved edges should be avoided when possible. Although this may not always be realistic because the design is tailored to the part's purpose, the methodology of avoiding curves should be applied when possible to maximize throughput.

8. TOPOLOGY

8.1 Topology Manipulation

Although it may seem illogical, decreasing material used in a print does not necessarily decrease the time of the print. Making a part smaller overall will often save time and material, but adding holes, such as for mounting features, often adds time. This is because adding holes adds more tool paths. Each added path adds a start/stop sequence as well as dwells and travel moves. An example of the complexity involved in printing a hole is shown in Figure 25. For this reason, it's recommended that small holes are not printed and are drilled in after printing.

To demonstrate this, data was collected for the print time of a 36" square plate with a circular hole in the center. The size of the hole was varied from 0-25" in diameter. The print was done with one perimeter bead, two inset beads, and concentric solid infill. Print times and weights can be seen in Table 4 for a variety of nozzles. All 26 versions of the STL files of this part were sliced using ORNL-developed slicing software.

Table 4 – Time and weight calculations for different size holes in a 36”x36” plate.

Hole Diameter (in)	0.1" Nozzle Layer Time (mm:ss)	0.1" Nozzle Weight (lb)	0.2" Nozzle Layer Time (mm:ss)	0.2" Nozzle Weight (lb)	0.3" Nozzle Layer Time (mm:ss)	0.3" Nozzle Weight (lb)	0.4" Nozzle Layer Time (mm:ss)	0.4" Nozzle Weight (lb)
0	26:41	2.71	12:43	5.42	08:10	8.13	12:13	12.06
1	40:20	2.7	16:12	5.38	10:32	8.04	13:14	11.87
2	40:55	2.7	16:12	5.37	10:19	8.05	13:04	11.84
3	42:52	2.69	16:19	5.35	10:41	8.01	13:05	11.8
4	42:19	2.68	15:58	5.33	10:18	7.96	12:49	11.71
5	44:47	2.66	16:18	5.3	10:40	7.91	12:56	11.67
6	44:23	2.65	15:40	5.27	10:03	7.87	12:35	11.53
7	44:45	2.62	15:14	5.23	09:33	7.79	12:37	11.49
8	44:03	2.6	14:36	5.17	09:29	7.64	12:15	11.31
9	43:19	2.57	14:29	5.11	09:19	7.63	12:13	11.25
10	42:32	2.54	14:20	5.04	09:19	7.55	11:54	11.03
11	41:50	2.5	13:56	4.98	09:06	7.44	11:59	10.87
12	40:59	2.47	13:45	4.9	08:49	7.31	11:32	10.69
13	40:10	2.43	13:34	4.82	08:55	7.22	11:31	10.48
14	38:57	2.38	13:26	4.74	08:48	7.07	11:27	10.33
15	37:43	2.33	13:19	4.63	08:43	6.9	11:05	10.05
16	36:14	2.28	13:22	4.53	08:53	6.67	11:03	9.87
17	34:27	2.23	13:30	4.42	08:33	6.62	10:33	9.56
18	32:35	2.17	12:56	4.3	08:29	6.43	10:29	9.29
19	31:15	2.11	12:52	4.18	08:11	6.23	09:55	9.02
20	29:24	2.05	12:31	4.06	07:53	6.02	09:53	8.7
21	28:13	1.98	12:12	3.93	07:59	5.86	09:20	8.42
22	26:26	1.91	12:02	3.79	07:30	5.63	09:10	8.05
23	24:33	1.83	11:37	3.63	07:47	5.4	08:37	7.78
24	22:46	1.76	11:13	3.47	07:15	5.18	08:30	7.35
25	22:31	1.68	10:50	3.31	06:56	4.94	07:59	6.97

Of the four, the 0.1” nozzle takes the most amount of time. This is because it lays down a narrower bead and thus needs to make many more passes during printing. Smaller nozzles increase the amount of movements and dwells to make a part. The 0.4” nozzle has to be printed with a slower gantry speed so that the extruder can keep up with the large volume of material that has to be extruded for a 0.5” bead. The larger nozzle deposits a taller and wider bead which increases the volume of material deposited and thus making the weight higher.

Plotting the data from Table 4 (Figure 26) shows how the 0.2” and 0.3” nozzles have a sharp increase from the plate with no hole to the plate with a 0.1” hole. This is because the addition of a hole adds several new moves to trace the perimeter of the hole. With each move, a start and stop are added. The 0.4” sees less of an increase due to the very large bead width of 0.5”, which enables fewer overall moves. The 0.2” and 0.3” layer times slowly trend downwards until reaching a 20” hole. At the 20” hole the time is the same as with no hole. For the 0.4”, this break-even point occurs much sooner at a 9” hole.

8.2 Topology Optimization

Topology optimization refers to using a mathematical approach to reduce material used to create an object while adhering to specific loading and boundary conditions. Since topology optimization aims to reduce material use, in theory, it should simultaneously reduce the overall weight of the object and, possibly, the time needed to create it. Although technology in topology optimization is advancing [6], especially in various CAD packages, it is currently not ideal for large-scale AM. Regarding the Z-dimension, organic structures that lack traditional geometry are unfavorable as they need support material. An example of a small part, a GE engine bracket, that has been topologically optimized is shown in Figure 27 [7].

9. Conclusions

The development of large-scale AM has created many opportunities for innovation in the manufacturing industry; it has also created some challenges, especially in the way parts are designed. This guide has detailed the main obstacles and possible solutions and alternatives.

Overall, the capabilities and limitations of the software used in both design and slicing must be considered. Bridges can be printed, but the width the bridge can span is dictated by material properties. Cavities can be incorporated into parts by the correct use of bridges or the use of support material. However, a hole in the part must be left for later removal of that support material. Additionally, support material should be avoided in large-scale additive manufacturing as it leaves large defects and is very difficult to remove. Delamination may occur if proper layer times are not used, and even if proper layer times are used, the weakest direction of an additively manufactured part is the Z-direction. Keep in mind where a part will bear a load during the design process, and ensure parts bear loads in the X- or Y-direction when possible. Due to AM's layer by layer nature, a corduroy-like surface finish will occur on a printed part. The in-plane surface is much smoother than the layer to layer surface, but this surface can be smoothed by hand sanding, CNC machining, or the use of a chemical such as acetone. Ensure that overhang angles are more than 45 degrees to avoid sagging, drooping, or even part failure during the printing process. Extrusion nozzle diameter does not necessarily equal the bead width; this can affect geometric accuracy. An important limitation of slicing software to consider is that parts must be printed using a closed loop path. This means that the extruder must start and stop in the same location on each layer. It is also difficult to print a single bead walled structure with the current slicing capabilities. However, printing a single bead walled structure can be achieved by designing the part as a solid structure and adjusting the infill and skin settings in the slicing software. The software will essentially hollow out the part. As mentioned, layer times are very important. If a layer time is too short, the layers of a part will sag and droop as more material is deposited on top of the still malleable material. If a layer time is too long, the bottom layer will solidify too much for proper adhesion with the next layer, and delamination is likely to occur. Avoiding curves in a design is preferable because they affect the extruder's throughput causing inconsistencies in a part. Lastly, topology optimization capabilities in CAD software are improving and can help with the light-weighting of parts, which saves print time, material, and cost.

Through research and thorough testing on the BAAM system, these design guidelines have been found to be true. New developments will continue to alter and improve the guidelines, but this is the foundation of design for additive manufacturing.

10. ACKNOWLEDGMENTS

This material is based upon work supported by the U.S. Department of Energy, Office of Advanced Manufacturing under contract number DE-AC05-00OR22725.

The authors would like to acknowledge the contributions of Breanna Rhyne, Sierra Palmer, Rowan Palmer, Alex Riestenberg, Georgiana Blue, Kaylee Kiefaber, and Madison Dittner to this work. Breanna Rhyne, Sierra Palmer, and Georgiana Blue contributed topic suggestions and edits from a mechanical engineering perspective. Alex Riestenberg provided suggested edits and Cincinnati Incorporated's perspective. Rowan Palmer designed several graphics shown throughout the work. Kaylee Kiefaber and Madison Dittner ran the tests and collected the resulting data shown in Tables 3 and 4 and Figure 26.

11. REFERENCES

[1] I. Gibson, D. Rosen, B. Stucker, Additive Manufacturing Technologies: 3D Printing, Rapid Prototyping, and Direct Digital Manufacturing, Springer Link. 2nd Ed (2015). https://doi.org/10.1007/978-1-4939-2113-3.

[2] L.J. Love, C.E. Duty, B.K. Post, R.F. Lind, P.D. Lloyd, V. Kunc, W.H. Peter, and C.A. Blue, Breaking Barriers in Polymer Additive Manufacturing, U.S. Department of Energy Office of Scientific and Technical Information. (2015). https://www.osti.gov/servlets/purl/1185467.

[3] A. Gaynor, T. Guest, Topology optimization considering overhang constraints: Eliminating sacrificial support material in additive manufacturing through design, Structural and Multidisciplinary Optimization. 54(5), (2016) 1157-1172. https://doi.org/10.1007/s00158-016-1551-x.

[4] A. Schmidt, Self-Supporting Angles in Large Scale Additive Manufacturing: AES Originals, Large Scale 3D Printing, Additive Engineering Solutions. (2017). https://www.additiveeng.com/self-supporting-angles-in-large-scale-additive-manufacturing/.

[5] R. Dinwiddie, L. Love, J. Rowe, Real-time process monitoring and temperature mapping of a 3D polymer printing process, SPIE, Thermosense: Thermal Infrared Applications. (2013). https://doi.org/10.1117/12.1518454.

[6] T. Zegard, G. Paulino, Bridging Topology Optimization and Additive Manufacturing, Structural and Multidisciplinary Optimization. 53.1 (2016) 175-92. https://doi.org/10.1007/s00158-015-1274-4.

[7] Metrology News, Topology Optimization and DMP Combine to Meet GE Aircraft Engine Bracket Challenge. E-Zine Media. http://metrology.news/topology-optimization-dmp-meet-aircraft-challenge, 2016 (accessed 12 September 2018).

FIGURE CAPTIONS

Figure 1 – A cup designed using CAD software (left) and the STL approximation of the same cup (right).

Figure 2 - The triangulation process to go from CAD curvature to STL straight lines.

Figure 3 – A horizontal plane intersecting the STL file as done in slicing.

Figure 4 – Resulting cross section after creating a slice.

Figure 5 – In this figure, the red lines represent the perimeter, the cyan lines represent insets, and the green grid is the infill region.

Figure 6 – A G-Code sample showing the start of a print, through the end of the first bead.

Figure 7 – A part that bridges the gap between two towers at layer five.

Figure 8 – A CAD model of the bridging part used for testing (left) and the as-printed version (right).

Figure 9 – Section-view of a part with an internal cavity and a hole in the side for removing support (left), and a solid view of the model with roof shown (right).

Figure 10 – A part showing a hidden internal cavity. The cavity can only be seen when the part is cut in half as shown on the right.

Figure 11 – A design solution that maintains the cavity but supports the roof by growing the walls outward at 45-degrees.

Figure 12 – A printed part with layers stacked vertically (left), a load applied in line with the layer bonds (center), and the result of the load (right).

Figure 13 – A closeup photo of Strati, the World's First 3D Printed Car, showing the delaminations in the thin walled fenders.

Figure 14 – A section of a wall cut for measuring the layer to layer surface finish.

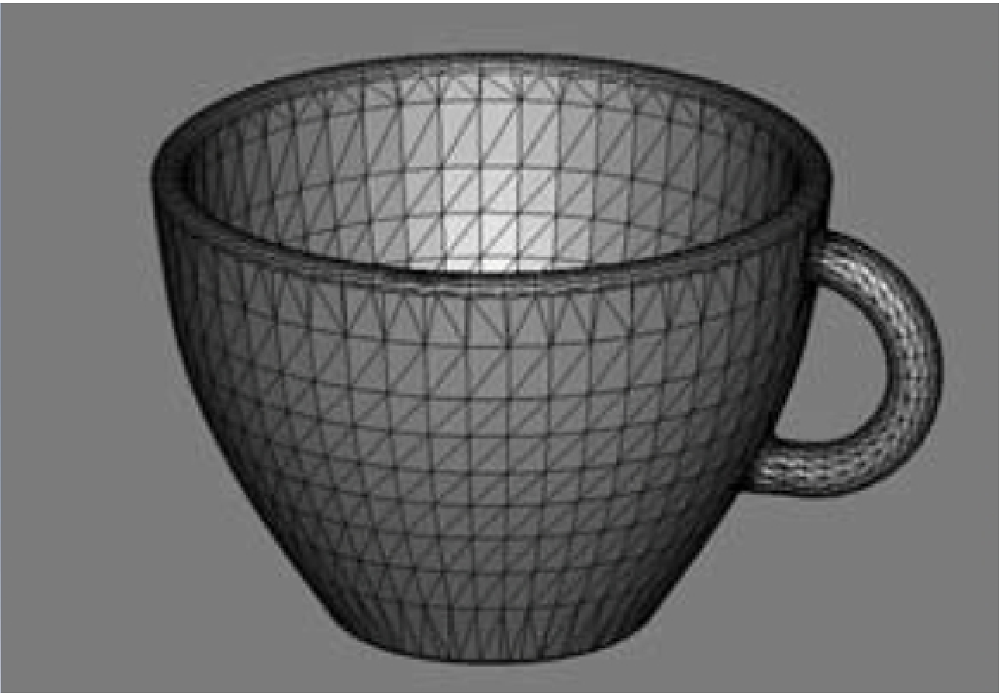
Figure 15 – The top of an 8-bead wall used to measure the in-plane surface finish.

Figure 16 – An illustration of a 45-degree and a 20-degree overhang from the horizontal plane.

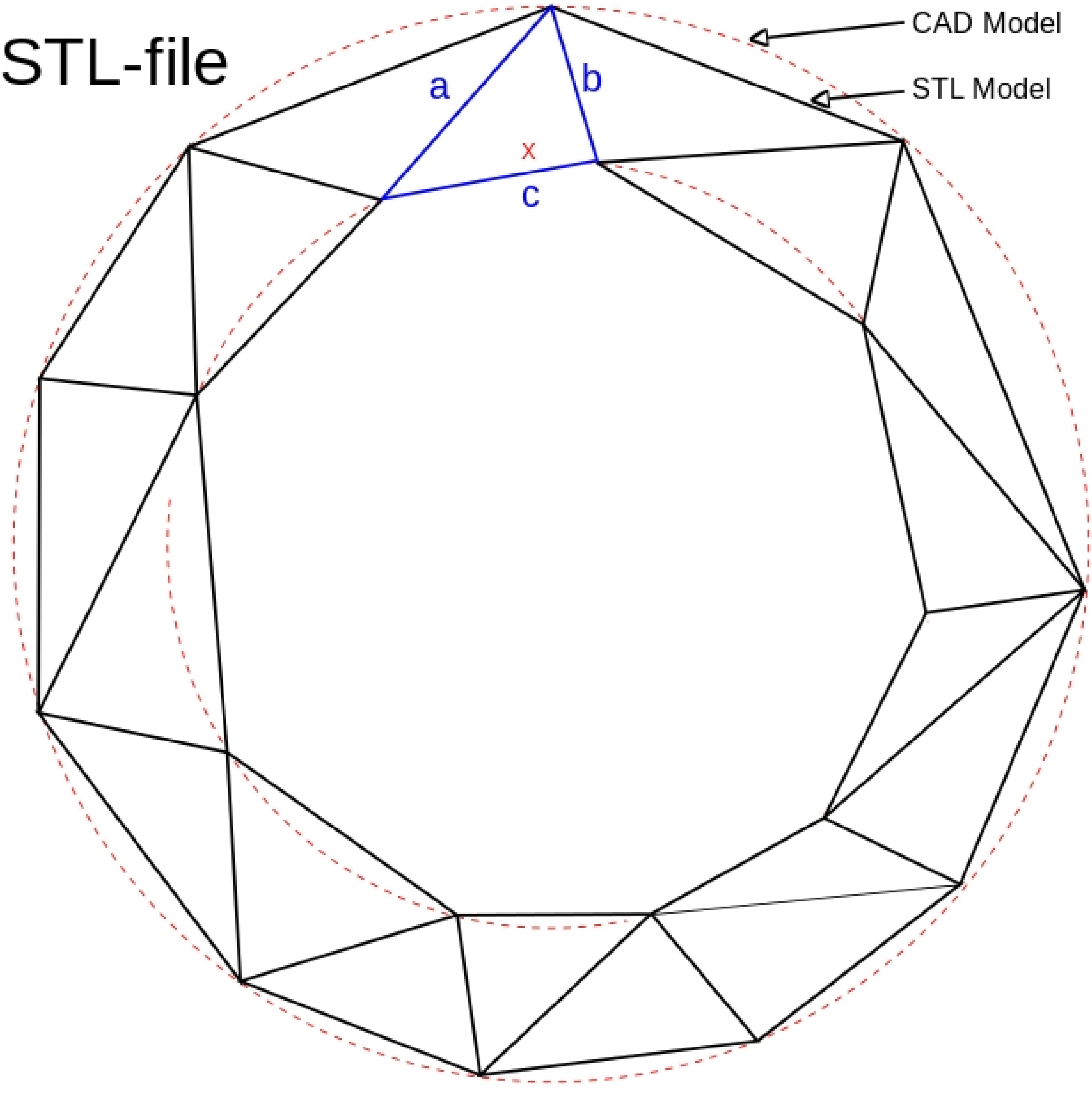
Figure 17 – Convex and concave overhang angle test pieces.

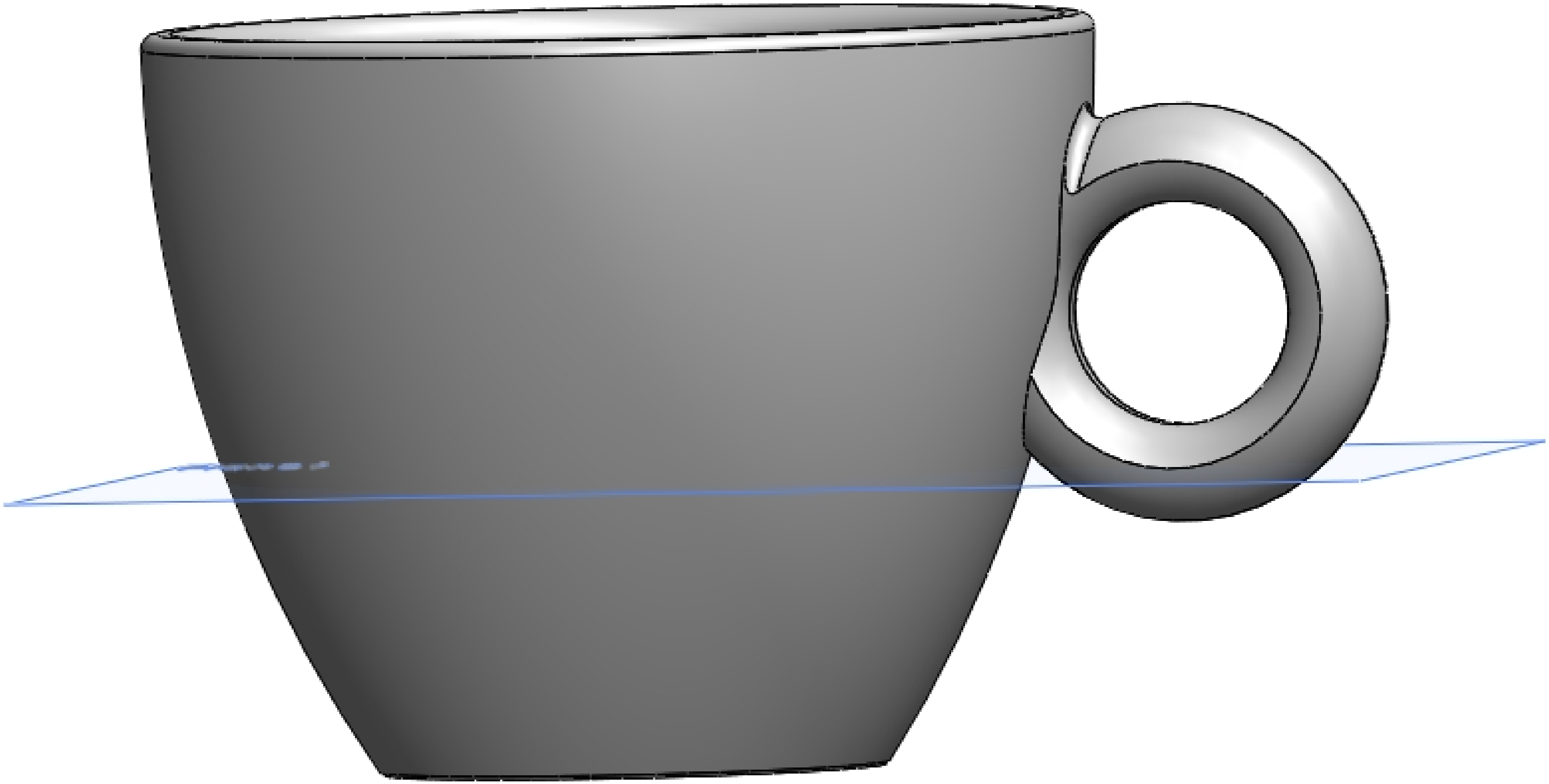
Figure 18 – Photograph of the tamper device. The round disk at the bottom moves up and down to tamp the bead of malleable material after extrusion.

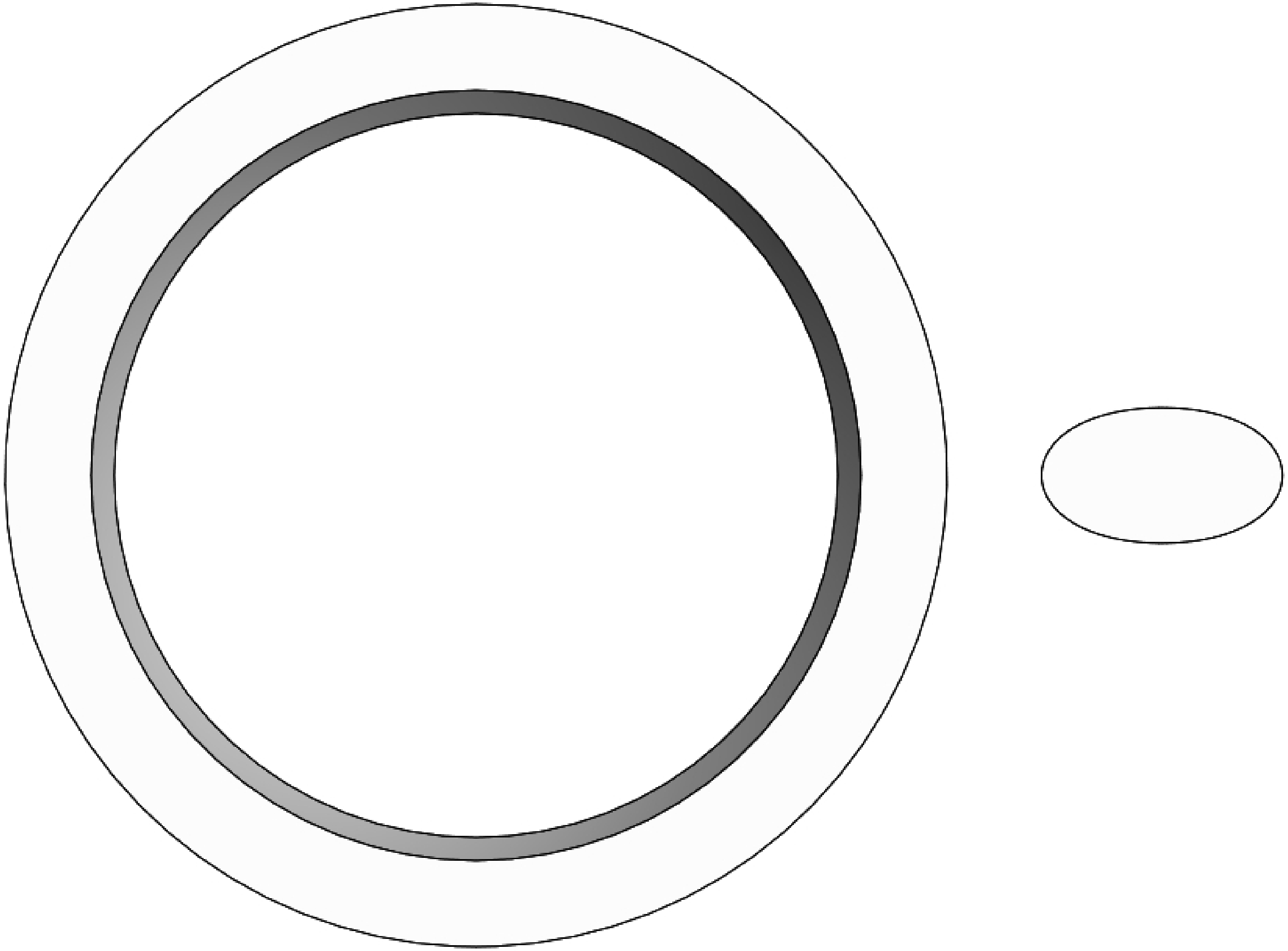
Figure 19 - Example 1 displays an open loop, and Example 2 displays a closed loop. Example 1 cannot be generated with current slicing software.

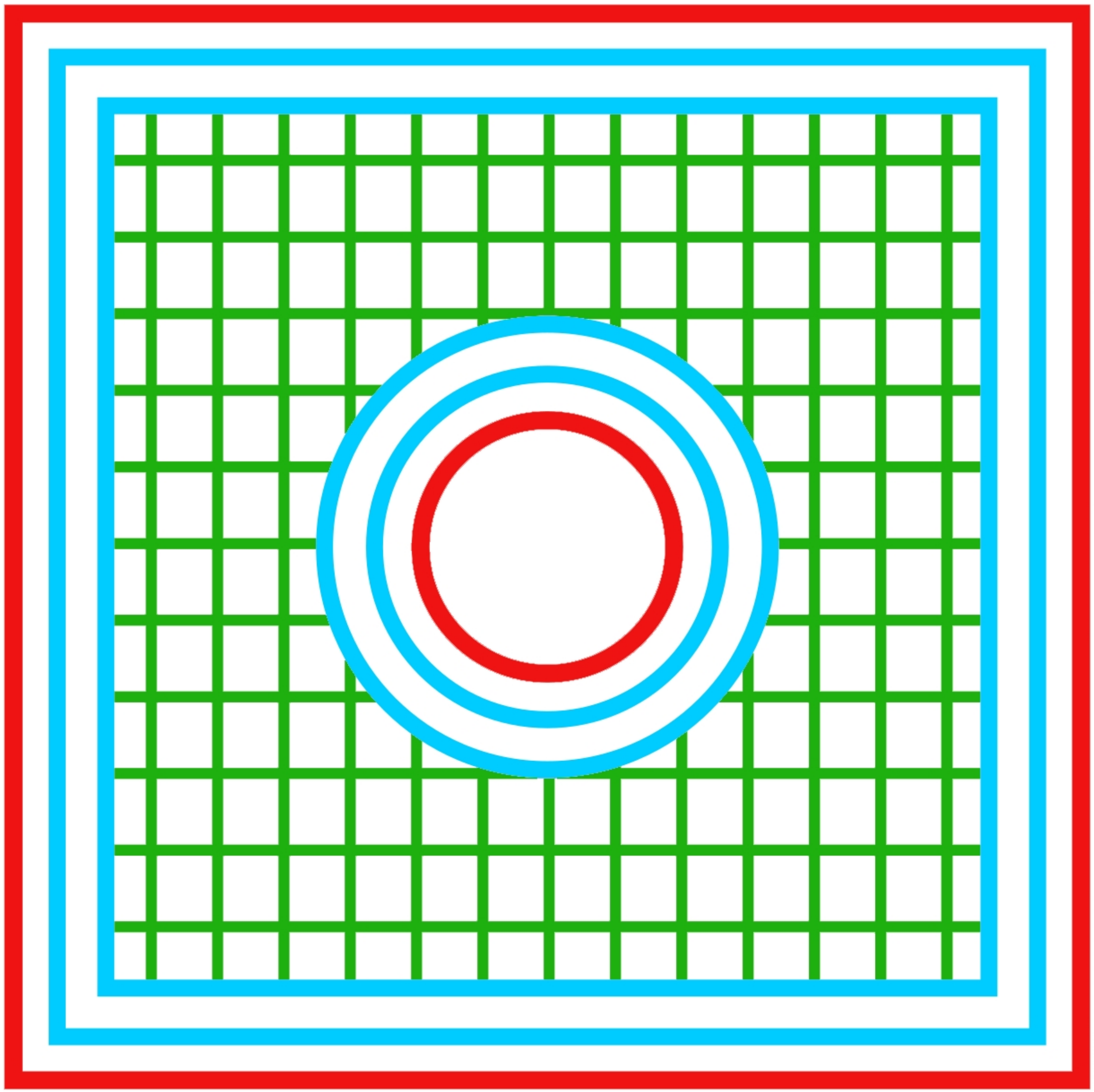



STL-file

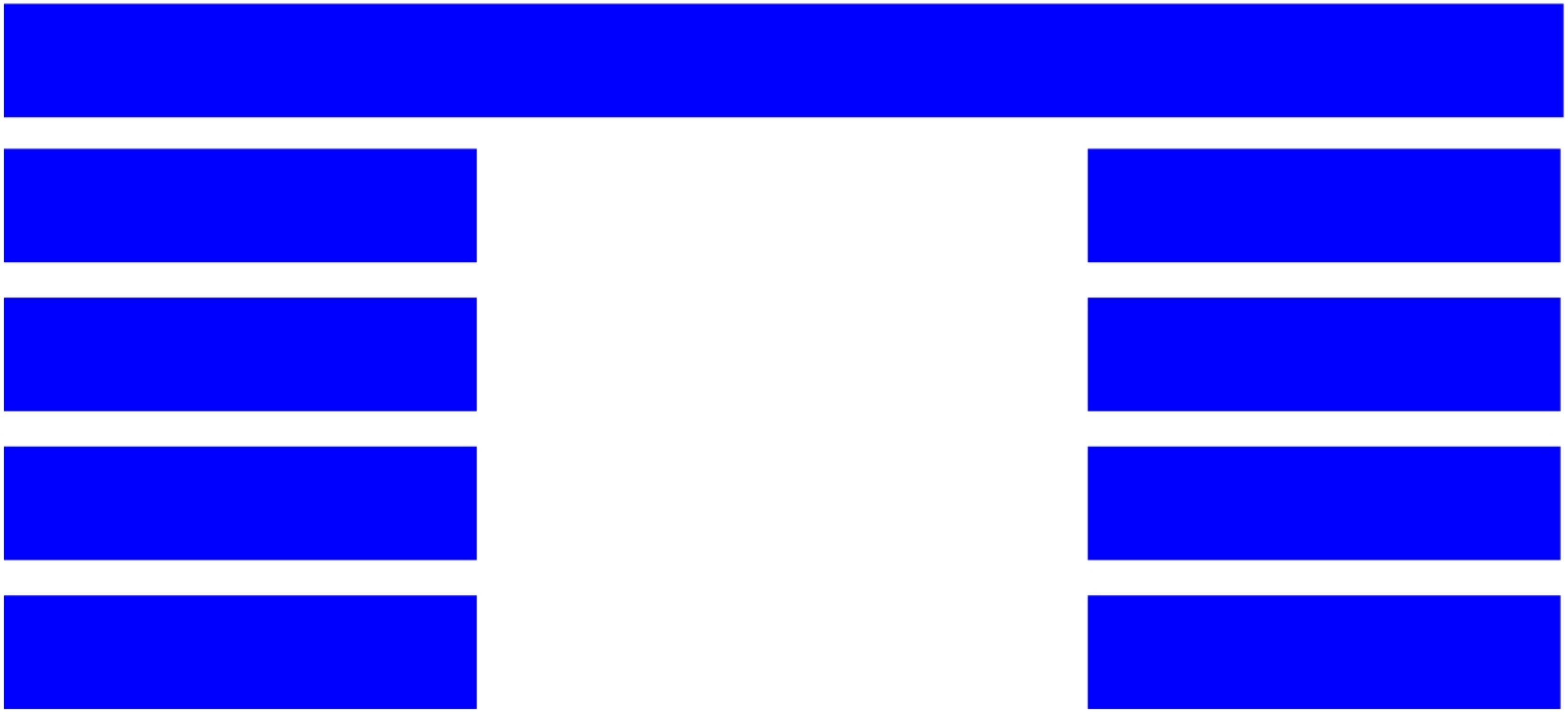


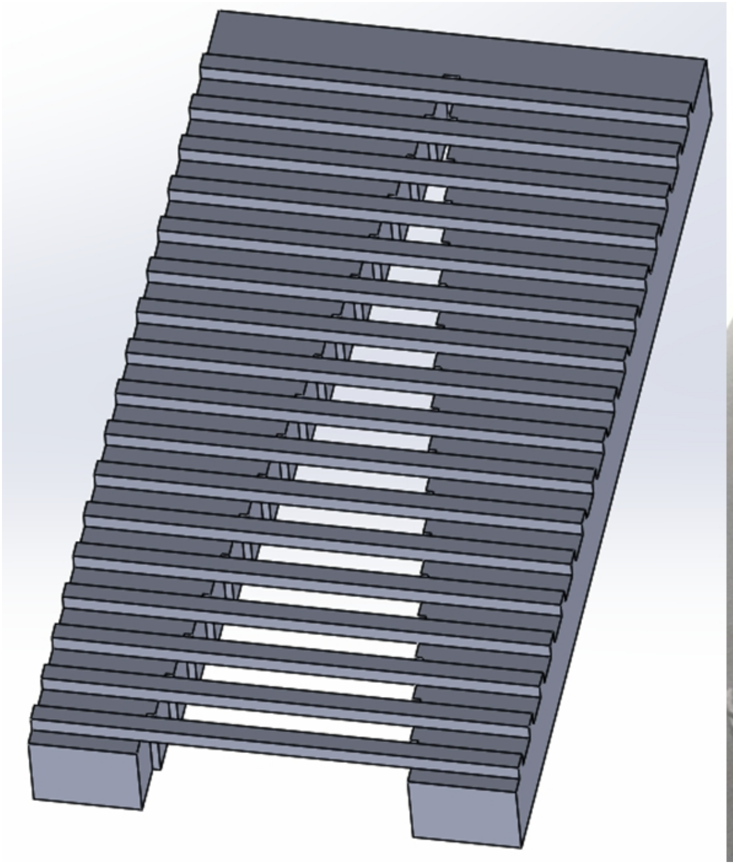


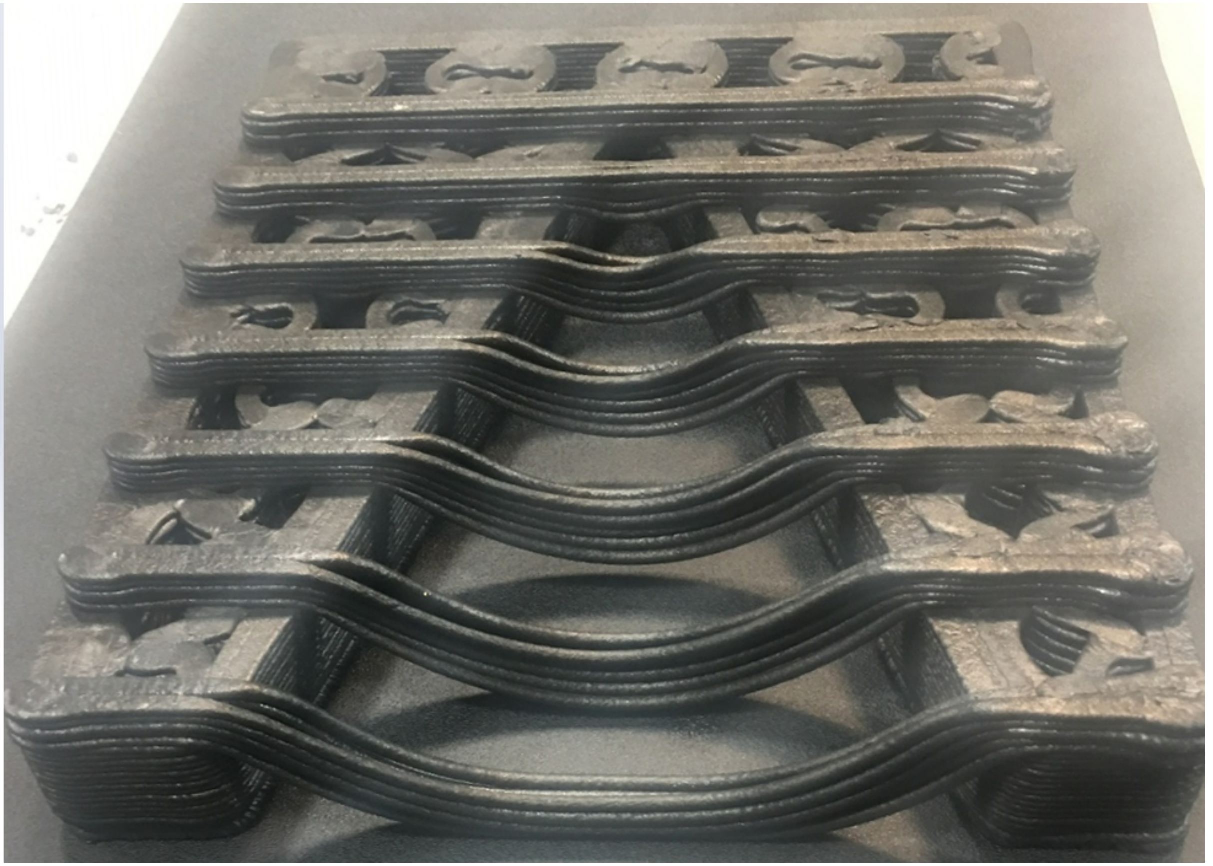


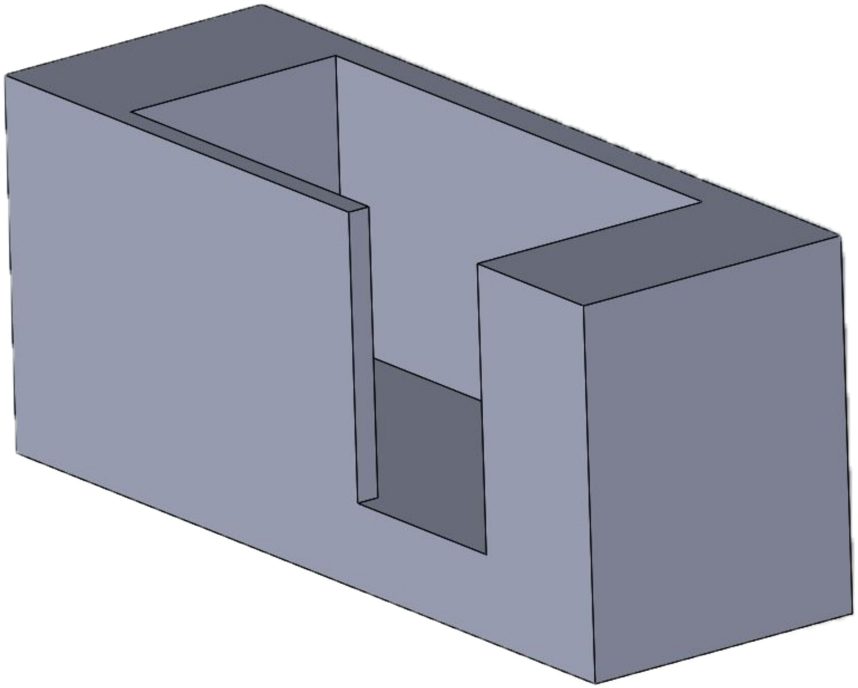


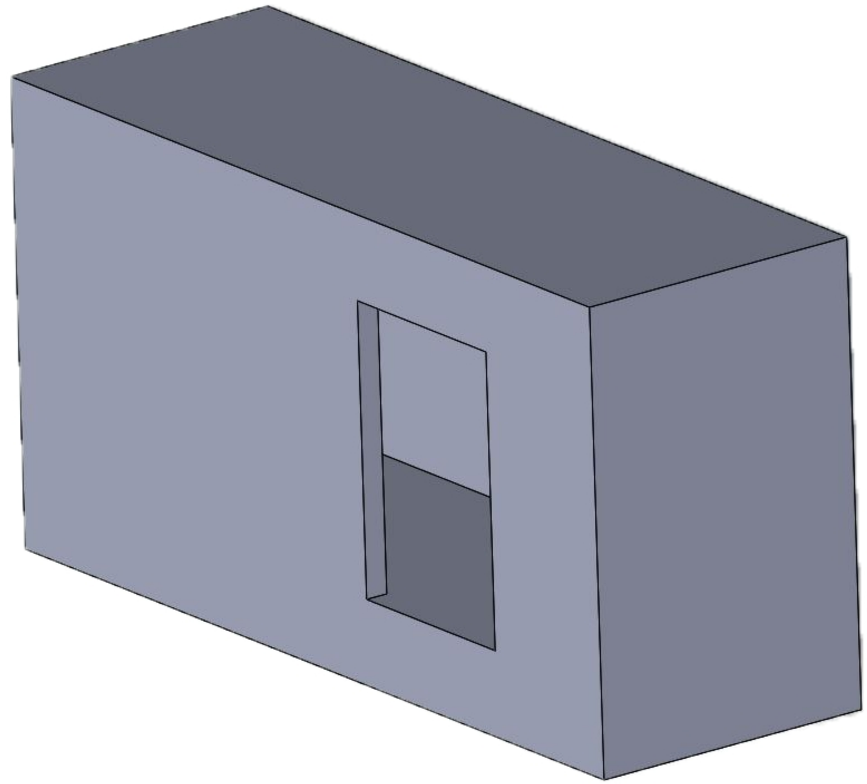
```
(Layer count: 27)
(BEGINNING LAYER: 1)
G1 F59.0551 Z0 (TRAVEL-Move Extruder off table)
G0 W-0.1500 (Moving Table)
M1 (OPTION STOP - START OF LAYER)
G0 X106.3950 Y25.1700 (TRAVEL)
G0 W-0.1500 (TRAVEL)
M66 L0.1593 (SET ACCELERATION VALUE)
G1 F59.0551 Z-14.2700 (TRAVEL-Lift Tip For Travel)
G0 X106.3950 Y25.1700 (TRAVEL)
G1 Z-14.7500 (TRAVEL-Lower Tip)
(TYPE:VOLUME)
M66 L0.2655 (SET ACCELERATION VALUE)
M12 (PERIMETER SPINDLE ADJUSTMENT)
M64 L0.70 (Turn Tamper ON)
M60 L1000 P10 (TURN FEED SHAKER ON)
M11 (Turn OFF Extruder Servoing)
M3 S300.0 (Turn Pump on)
G4 P0.05 (Dwell)
M3 S400 (Change Pump Speed)
M10 (Turn ON Extruder Servoing)
G1 F649.6063 X136.0550 Y25.1700 (VOLUME-WALL_OUTER)
G1 X136.0550 Y60.3300 (VOLUME-WALL_OUTER)
G1 X132.3950 Y60.3300 (VOLUME-WALL_OUTER)
G1 X132.3950 Y54.6700 (VOLUME-WALL_OUTER)
G1 X110.0550 Y54.6700 (VOLUME-WALL_OUTER)
G1 X110.0550 Y60.3300 (VOLUME-WALL_OUTER)
G1 X106.3950 Y60.3300 (VOLUME-WALL_OUTER)
G1 X106.3950 Y25.1700 (VOLUME-WALL_OUTER)
M5 (Turn Pump OFF)
```

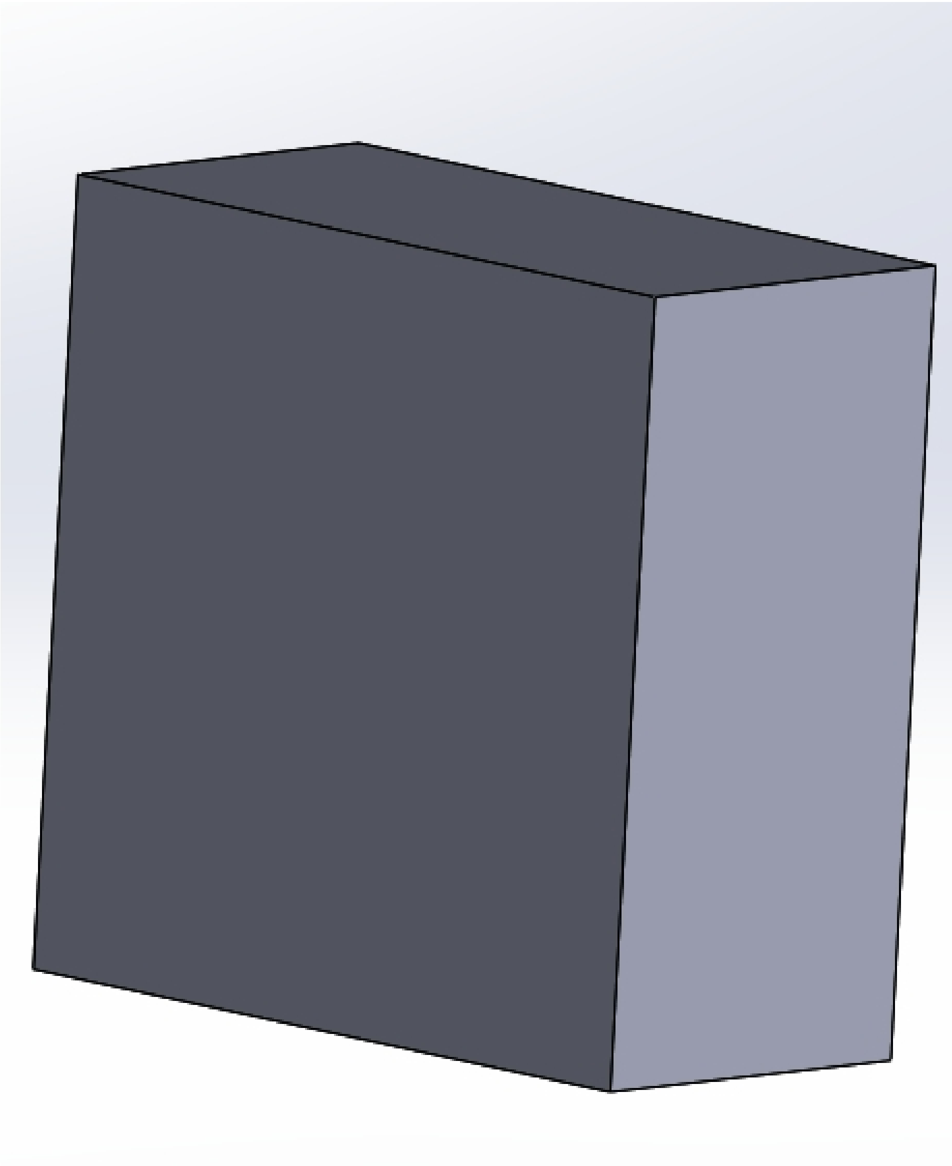


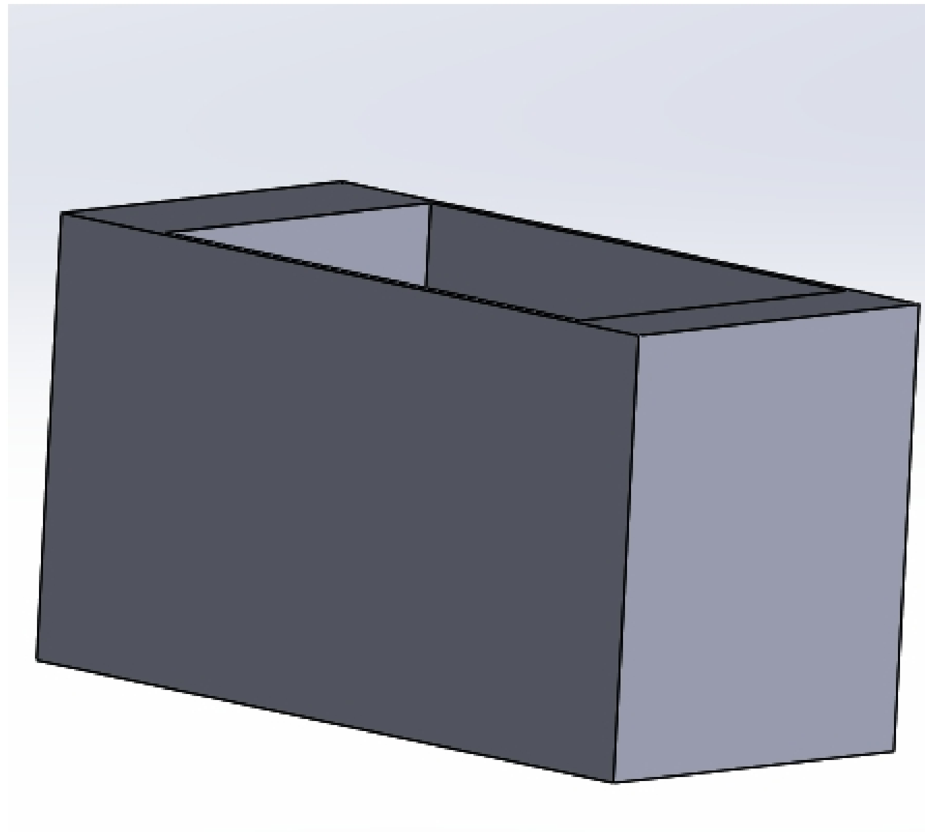


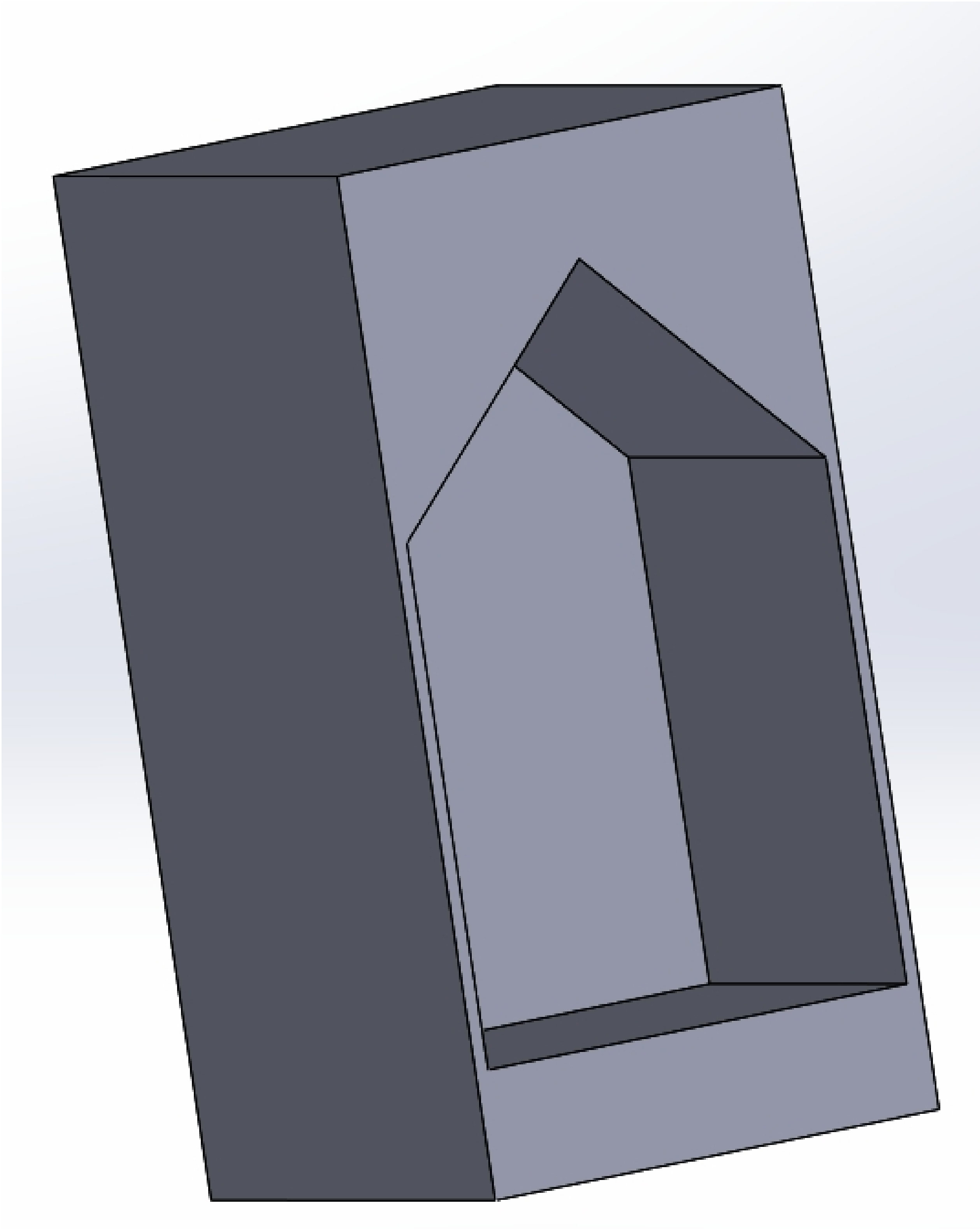


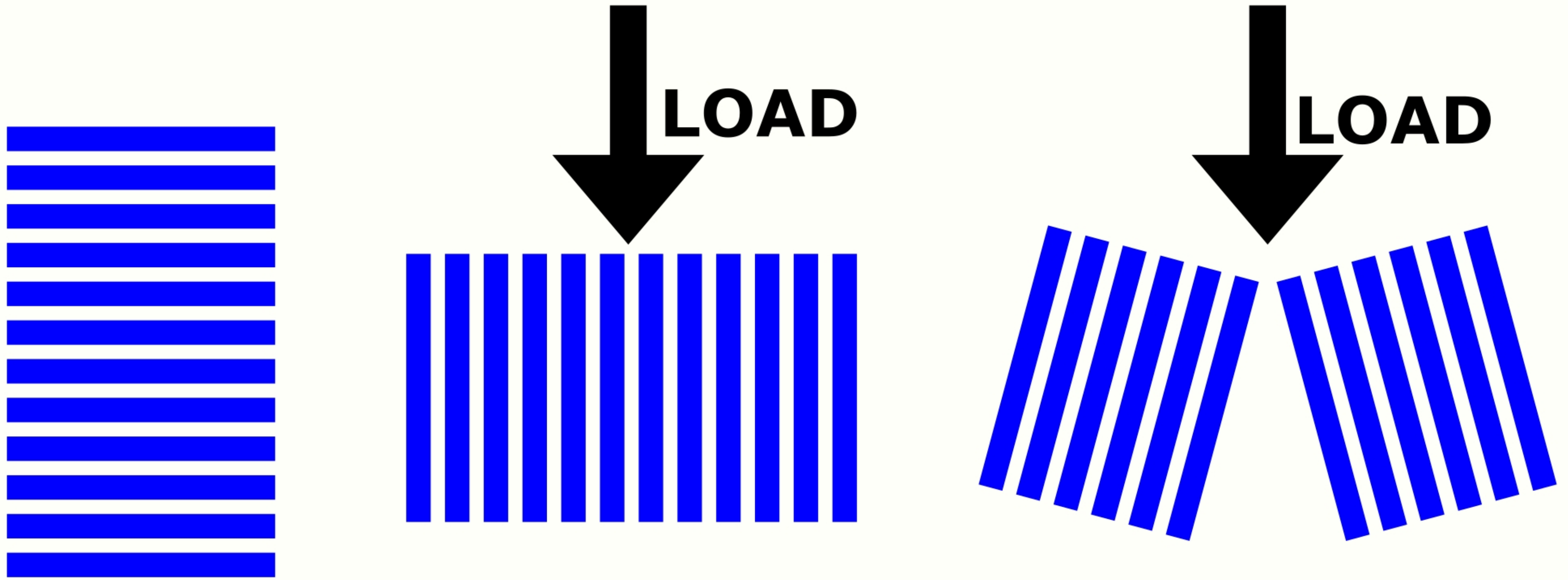


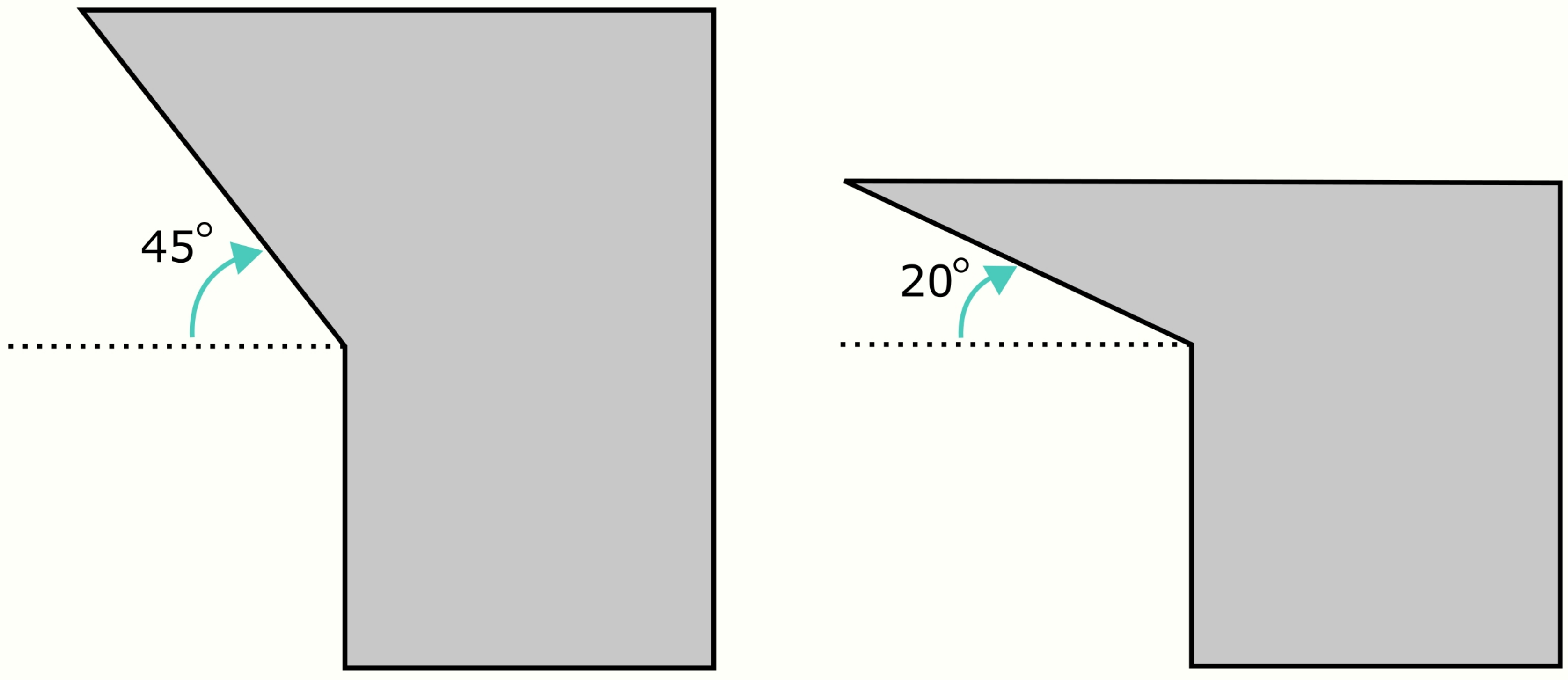


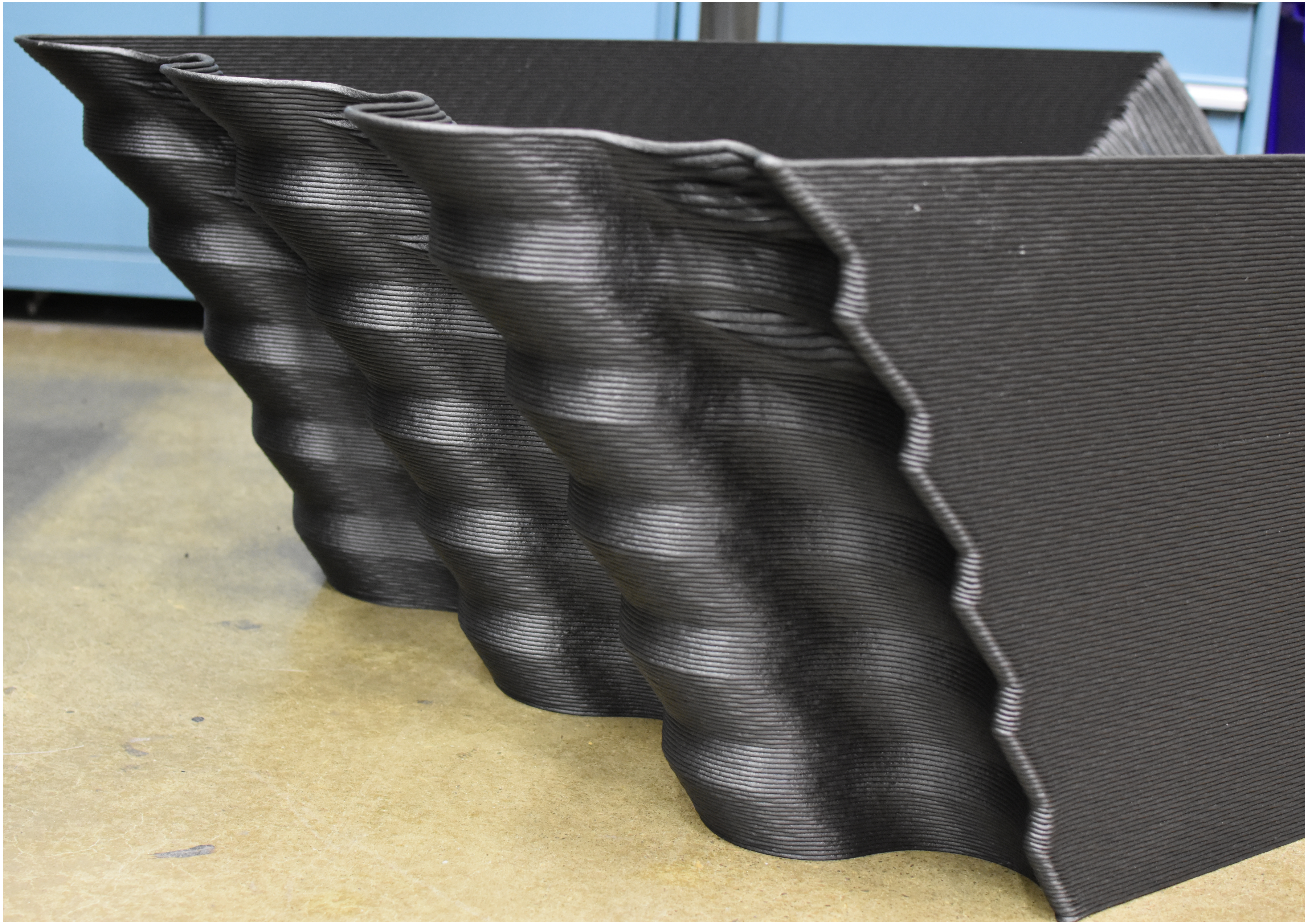


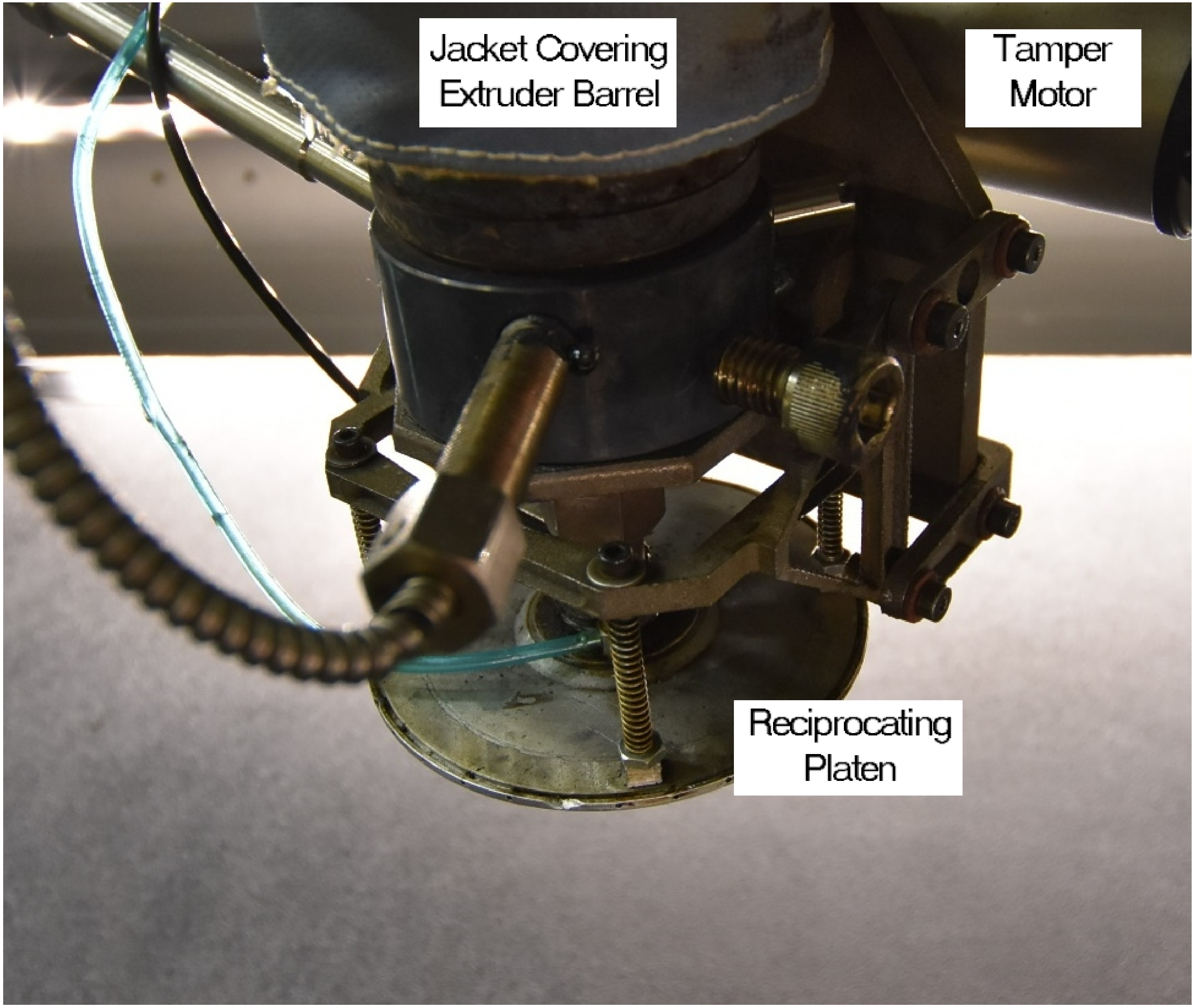




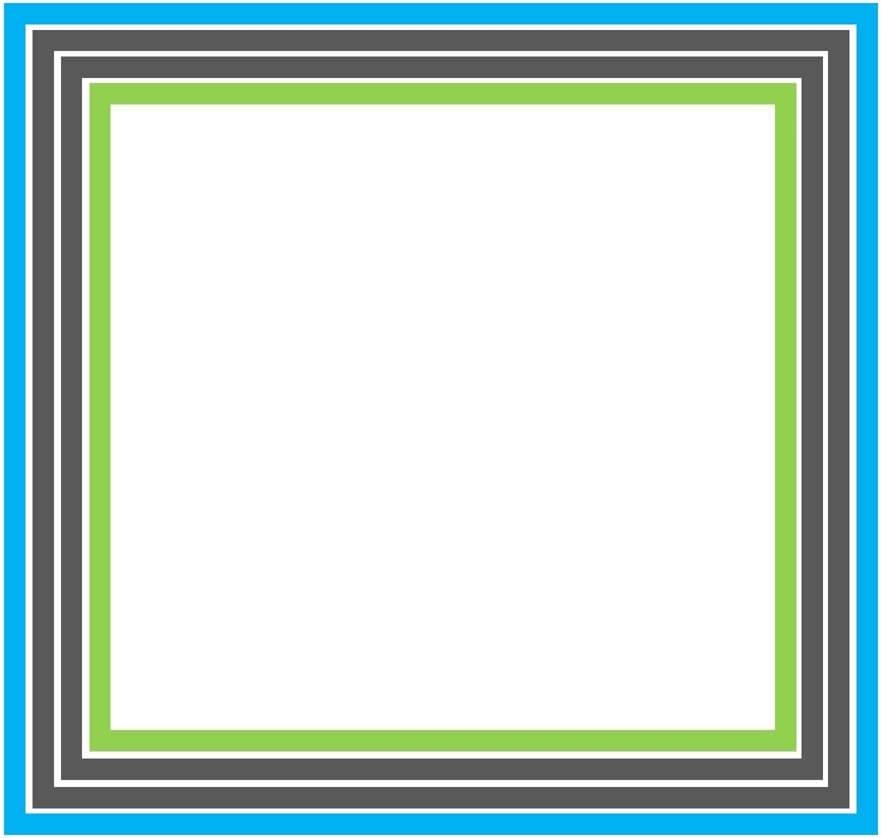

LOAD
LOAD

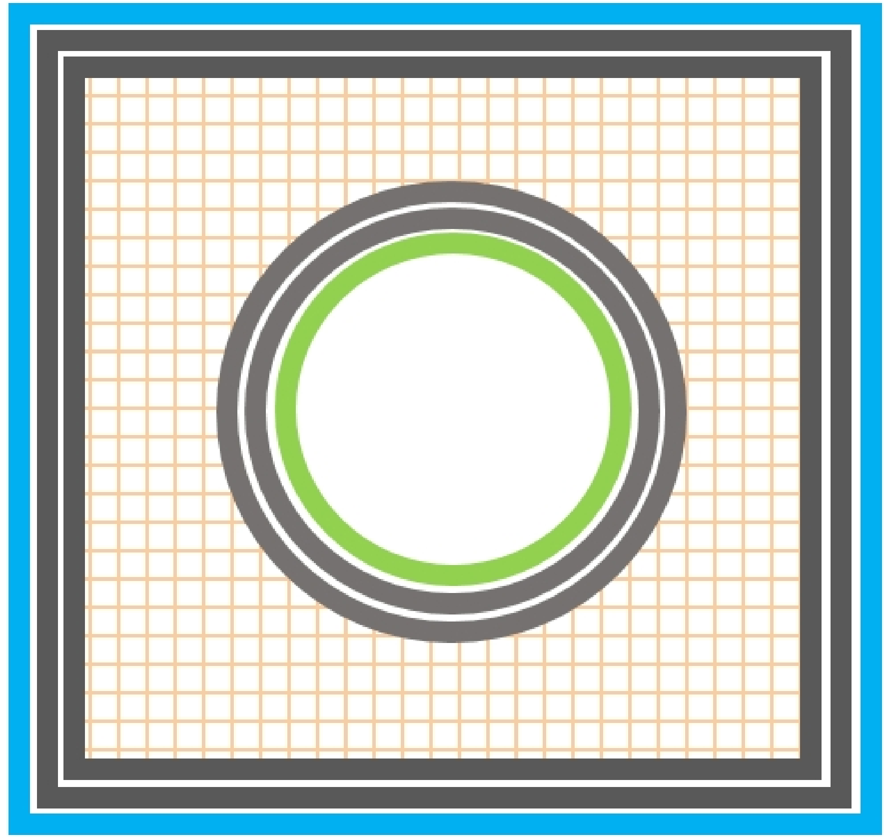

45°
20°

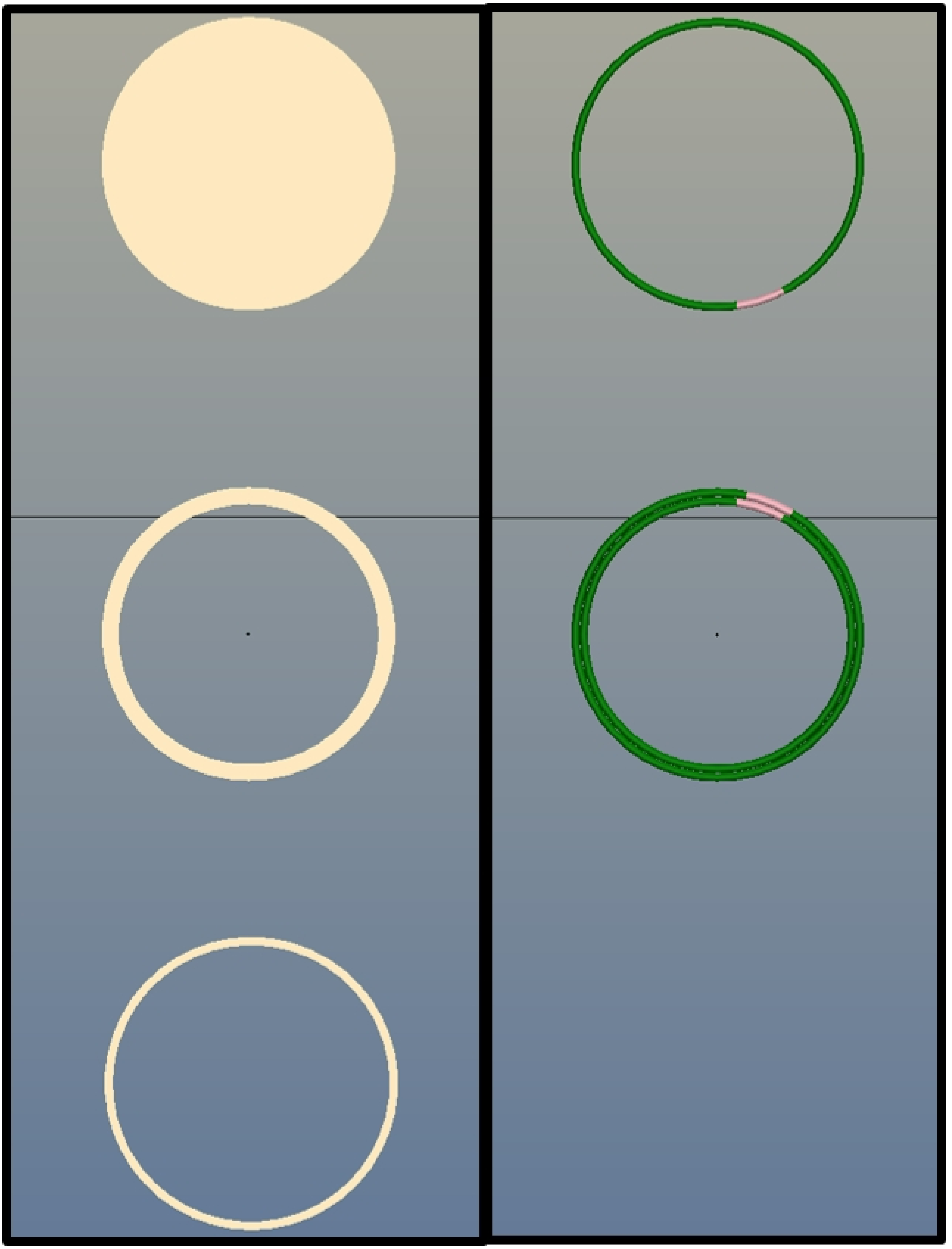


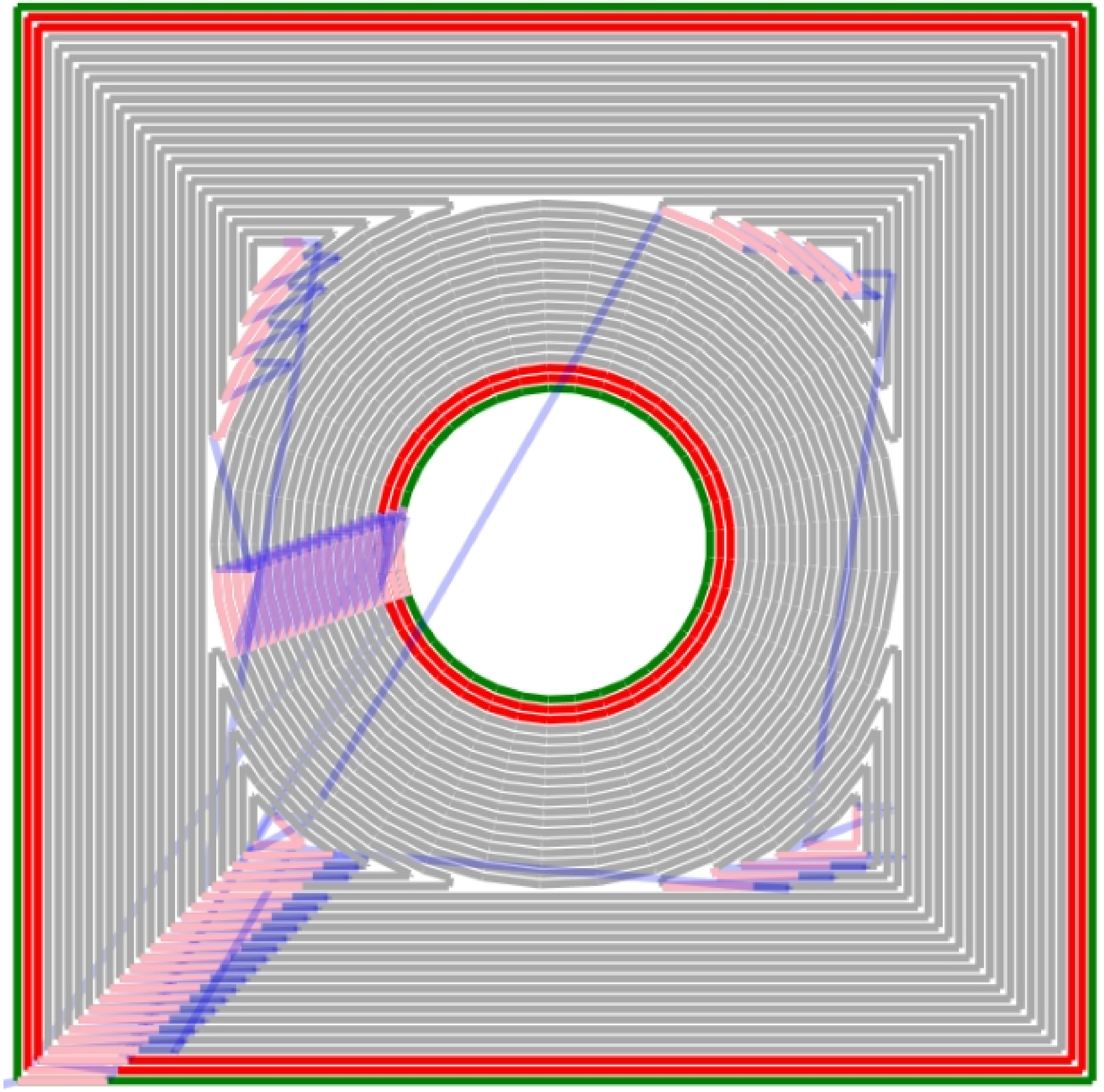

Jacket Covering
Extruder Barrel
Tamper
Motor
Reciprocating
Platen

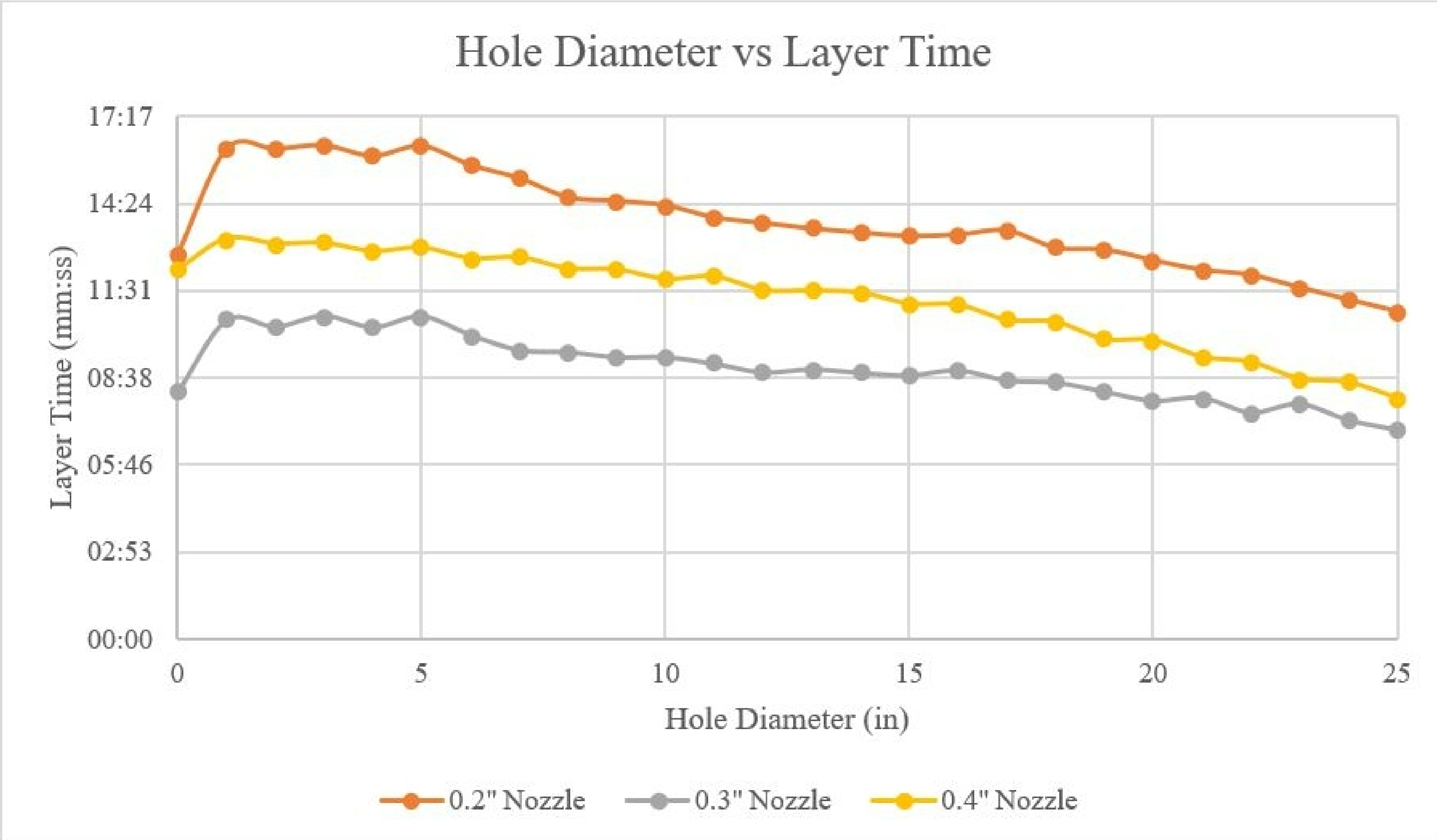

1.
START
STOP
2.
START
STOP










Hole Diameter vs Layer Time
Layer Time (mm:ss)
17:17
14:24
11:31
08:38
05:46
02:53
00:00
0
5
10
15
20
25
Hole Diameter (in)
0.2" Nozzle
0.3" Nozzle
0.4" Nozzle