Assignment 2

CURA Process planning assignment

Submitted By: -

Team Green

Utkarsh Dwivedi	**110135891**
Chirag Ahuja	**110121536**
Tej Kumar Patel	**110125676**
Mohamed Asif Mohamed Ismail	**110124148**

Additive Manufacturing

Submitted To: - Prof. Jill Urbanic

EXECUTIVE SUMMARY
Optimizing 3D Printing Parameters Using Cura Software

1. **Multi-face triangle**

Introduction: The study aims to optimize 3D printing parameters for an ABS material CAD model using Cura software. Usage of ABS material on Ender-3 pro is not suggested usually as ABS is a temperature sensitive material so it requires as closed wall 3D-printer; though Ender-3 pro is skeleton type printer which might affect the formation of any product using ABS for sure due to temperature deviations. Standard parameters were set, and experiments were conducted with variations in orientation, infill density, wall thickness, top/bottom thickness, infill patterns, and fill angles. The goal is to identify optimal settings that minimize build time, material usage, and non-value-added time while achieving a smooth surface finish.

Orientation Influence: The X0 Y90 Z0 orientation demonstrated the most advantageous results, exhibiting the shortest build time, minimal support material consumption, and the best surface finish. Deviating from this orientation increased build time and support material usage, emphasizing the intricate relationship between orientation changes and printing factors.

Infill Density Impact: Increasing infill density led to higher build times and material usage, showcasing the importance of selecting infill based on the intended strength needed. The recommended default is 20% infill without support for optimal results.

Wall Thickness and Line Count: Thicker walls and increased line count positively influenced structural integrity but led to longer build times and more material usage. A balance is needed to maintain strength while optimizing efficiency.

Infill Pattern Analysis & Fill Angle Exploration: The Grid infill pattern consistently outperformed Octet and Gyroid, providing efficient support structures and minimal time consumption. The choice of infill pattern significantly influences printing time and structural reliability. Variations in fill angles showed minimal impact on build time, material usage, and non-value-added time. Larger structures may exhibit more noticeable differences in infill angle effects.

Conclusion: The optimal configuration for the project involves X0 Y90 Z0 orientation, 20% infill without support, 1.2mm wall thickness with 3 wall lines, 1mm top/bottom thickness, and Grid infill pattern. This configuration yields a total build time of 50 minutes, minimal non-value-added time, and efficient material usage.

Recommendations for Future Work: Future work could focus on exploring advanced infill patterns, optimizing support structures, and incorporating real-time adjustments in software based on user preferences.

This project enhances understanding and provides practical insights for optimizing 3D printing parameters, contributing to the efficient utilization of resources and improved printing outcomes.

2. **Hemisphere with Engraved name**

The impact of different orientations on 3D printing parameters was thoroughly studied in this thorough study. The fourth orientation (X180 Y0 Z0) turned out to be the most beneficial one, with the shortest build time, the least amount of support material used, and the least amount of support material waste. Of all the orientations, it also had the cleanest surface quality. The long construction durations and higher need for support material resulting from deviations from this ideal alignment highlight the complex link between orientation modifications and important printing parameters.

Additionally, an increase in the proportion of infill corresponded to longer build times, non-value-added times, and higher build material utilization, all of which improved hull strength. It's interesting to note that the amount of support material consumed was unaffected by this change.

The examination of wall thickness and line count revealed a clear pattern: each 0.4mm increase in thickness resulted in an extra layer. Wider walls, which allowed for more direct material deposition, decreased non-value-added time while higher wall thickness increased build time.

A comparison of infill patterns showed that the lightning pattern was the most effective, requiring the least amount of material and producing faster printing. The pattern with zigzags showed the least amount of non-value-added time, whereas the one with cubic showed the most. These findings highlight the significance of orientation, infill %, wall thickness, and infill pattern in creating effective and resource-efficient results, and they offer useful considerations for optimizing 3D printing procedures. The following parameters were selected after optimizing all the parameters and the final result optimized result is shown below:

Parameter	**Value**	**Parameter**	**Value**
Material	PLA	Shell thickness	0.8mm
Nozzle diameter	0.4mm	Printing temp	200 °C
Resolution	Standard quality (0.2mm)	Build plate temp	60°C
Infill density	20%	Print speed	50 mm/s
Infill pattern	Lightning	Top/Bottom thickness	1 mm

3. **Oloid**

In this series of virtual experiments using Cura software, printer used in the experiment is Creality Ender-3 pro, various parameters were explored to understand their impact on 3D printing outcomes. Standard settings for ABS material included a shell thickness of 0.8mm, 0.4mm nozzle diameter, and a printing temperature of 230°C. The experiments considered factors like orientation, infill density, wall thickness, top/bottom thickness, and infill pattern.

Orientation significantly affected build time, support material usage, and surface finish. Optimal results were obtained with X60 Y0 Z0 orientation, providing a smooth surface without support. Infill density adjustments influenced build time and mass, with 20% infill recommended for efficiency. Wall thickness and line count modifications affected structural integrity, with 1.4mm thickness and 3 layers yielding the best results.

Top/bottom thickness alterations influenced build time and material usage, emphasizing the need for balance. Infill pattern analysis highlighted Tri-Hexagon as the most time-efficient, providing structural support with minimal material usage. Graphs illustrated layer thickness and build time correlations and the impact of orientation and infill on material usage.

Addressing software improvements, suggesting features like defining axes during orientation changes and incorporating surface smoothness indicators. G-code analysis revealed the bounding rectangle (MIN X: 73.271, MAX X: 134.775) and extrusion temperature commands (M104 S200, M109 S230). Bed temperature (M190 S80), maximum feed rate (F6000), and minimum feed rate (F300) were identified. The G91 command initiated relative positioning, retracting and raising Z for wiping (G1 E-2 Z0.2 F2400, G1 X5 Y5 F3000).

Fan speeds (M107, M106) were utilized for temperature control, starting and stopping during printing. The material deposition rate was estimated at 51.94 mm^3/s.

In conclusion, these experiments provide valuable insights into optimizing 3D printing parameters for efficiency, structural integrity, and surface quality. Recommendations include Tri-Hexagon infill, 20% infill density, and careful adjustments to orientation, wall thickness, and top/bottom thickness for desired outcomes. Software enhancements and detailed G-code analysis contribute to a comprehensive understanding of the 3D printing process.

4. Angle block set

In the context of additive manufacturing, a thorough process planning assignment was conducted using Cura software. The study explored several key parameters, including orientation, infill density, wall thickness, top/bottom thickness, and infill pattern, to optimize build time, material consumption, and surface quality.

Different orientations (X0 Y0 Z0, X90 Y0 Z90, X0 Y90 Z0) were systematically examined. The optimal orientation (X0 Y0 Z0) yielded minimal build time, reduced support material consumption, and superior surface finish. Deviations from this orientation led to increased build time, higher support material usage, and diminished surface quality.

Infill density variations (10%, 20%, 30%, 40%, 50%) were explored, indicating a direct correlation with increased build time and material consumption. The 20% infill density, without support, was identified as the best compromise between strength and weight.

Wall thickness (0.8mm to 1.8mm) and wall line count variations were studied, demonstrating their significant influence on build time, material consumption, and non-value-added time. The best-case scenario was identified as 20% infill, without support, with a wall thickness of 0.8mm and 2 layers.

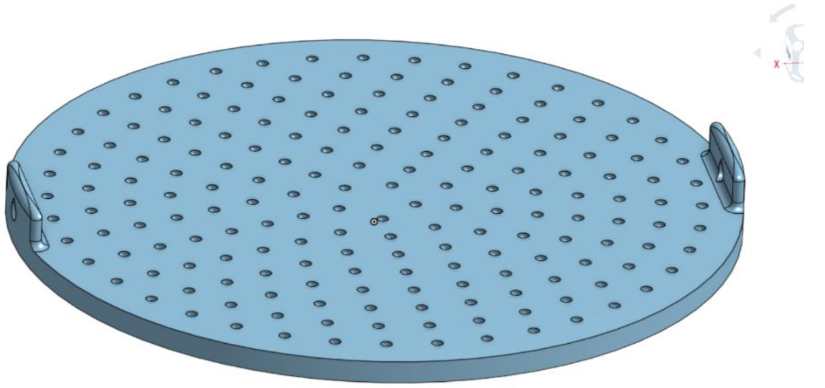
Top/bottom thickness variations (1mm to 5mm) were examined, revealing a direct relationship between increased thickness, additional layers, and extended build time. The optimal configuration was identified as 20% infill with support and a top/bottom thickness of 1mm.

Different infill patterns (Concentric, Cross, Quarter Cubic) were compared, emphasizing the balance between material usage and strength. Quarter Cubic was determined as the most suitable option, providing reasonable strength without compromising other factors.

The G-code was examined to determine the bounding rectangle, build time, extrusion and bed temperatures, feed rates, layers, and post-print commands. Noteworthy details included the use of G91 for relative coordinate movements and varying fan speeds during the print.

Zebrafish Embryo Mold Design – 1

The grooves are used for holding the zebrafish embryo at one place. By using this grooved dish, we can save time and resources by eliminating the step where we use Agarose gel and Mold to make contraption for holding Zebrafish embryo (refer Appendix A).



Advantages of the Spherically Grooved Dish:

Time Efficiency: The novel Petri dish eliminates the need for agarose gel preparation, significantly reducing the time required for setup. Researchers can quickly proceed to the microinjection stage, optimizing workflow efficiency.

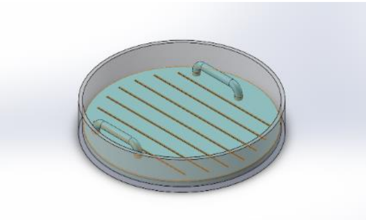
Improved Stability: The spherical grooves provide a secure and stable environment for zebrafish embryos, minimizing the risk of unintentional movements during microinjection. This enhances the precision and accuracy of the injection process.

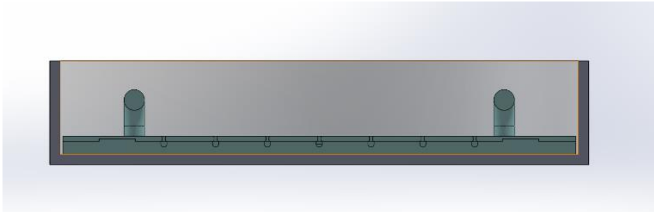
Enhanced Visibility: The transparency of the Petri dish facilitates real-time observation of the embryos during microinjection, allowing researchers to monitor the process and make adjustments as needed.

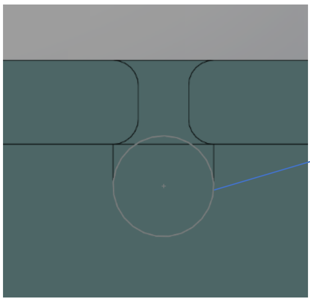
Reduced Embryo Handling: The dish simplifies the handling of zebrafish embryos, minimizing the risk of damage or stress during transfer. This is particularly beneficial for fragile embryos at early developmental stages.

No Need for cover: Periphery of spherical grooves can be used to remove sticked embryos from needle which reduces the design complexity.

ZEBRA FISH EGG INJECTION APPARATUS – DESIGN 2







Diameter for this section is 1.2 mm and this diameter is enough to accommodate the Zebra fish egg.

As per the design requirements, the above design was created for the injection of zebra egg. The primary element chosen as a base of the design was the petri dish of dimensions 100mm inner diameter with 18mm height of the petri dish. The base plate is a 2.5 mm thick and 99 mm diameter plate (leaving 0.5mm clearance around the dish). There are 7 slots cut on the base plate with width to be 1.2mm. As the average diameter of the Zebra fish egg is 0.7mm [1], 1.2mm will be enough to accommodate the egg without any physical damage from the wall of the plate (as shown in figure). Apart from this, there are 2 locator pins which can be used to assemble and reassemble the apparatus together without any confusion (refer Appendix B).

General questions discussed by group: -

Q.1) What are the characteristics of the materials you are exploring and their uses?

Ans. We have used 2 types of material in our whole case study; as per industrial usage they are commonly named as ABS and PLA.

ABS, or acrylonitrile butadiene styrene, is the filament that is used second most frequently. Superior melt flow properties and moderate strength and flexibility characterise ABS 3D printer filament. ABS can tolerate high temperatures thanks to its excellent heat tolerance, but its endurance is actually what makes it stand out. For objects that are handled, dropped, or heated regularly, this filament is perfect.

ABS 3D printer filament produces exceptionally durable products. They are resilient to repeated shocks and extreme temperatures. It is important for users to keep in mind that as products cool down, high printing temperatures can cause warping. This issue should be resolved with a heating bed. We provide 1.75mm and 2.85mm diameters of ABS filament for sale. Toys with high wear, tool handles, phone cases, and auto trim parts are a few examples of items [2].

Filament Properties

Strength	Flexibility	Durability	Difficulty to Use
High	Medium	High	Medium
Print Temperature	**Print Bed Temperature**	**Shrinkage / Warping**	**Soluble**
210°C - 250°C	80°C - 110°C	Possible – care required; print with heated bed	In esters, ketones, and acetone

[2]

Developed about 1990, ABS was one of the first plastics to be utilised with industrial 3D printers. This thermoplastic polymer is referred to as a "terpolymer," which is a polymer that is created by combining three distinct monomers. In this instance, acrylonitrile and styrene are most frequently polymerized with polybutadiene; the resulting material is called ABS and is typically composed of 20% acrylonitrile, 25% butadiene, and 55% styrene. Furthermore, by adjusting these ratios, ABS's characteristics can be changed. For instance, ABS's stiffness and brilliance are attributed to styrene, while its impact resistance and low temperature features are attributed to butadiene [3].

PLA provides several advantages over other materials, including:

- Ecological (when disposed of properly)
- Safe for usage in applications including food containers and medical devices.
- Simple to 3D print
- Offers a large selection of composite and colour choices for varying appearances and qualities.
- Solvent welding is possible (such as using dichloromethane)

There are, however, some disadvantages with using PLA, including:

- Low heat resistance
- Comparatively low strength
- Machine processing can be difficult.

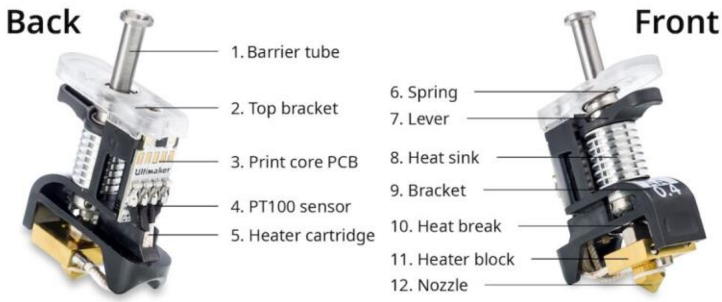
Property	**Value**
Heat Deflection Temperature (HDT)	126 °F (52 °C)
Density	1.24 g/cm³
Tensile Strength	50 MPa
Flexural Strength	80 MPa
Impact Strength (Unnotched) IZOD (J/m)	96.1
Shrink Rate	0.37-0.41% (0.0037-0.0041 in/in)

Creating PLA produces 68% fewer carbon emissions, requires 65% less energy than creating conventional polymers, and is toxin-free. It can also continue to be environmentally friendly if the proper end-of-life procedures are carried out. A 2017 investigation, however, revealed that after the material was immersed in saltwater at 25°C for more than a year, no degradation was seen, indicating that the rate of disintegration is extremely slow at room temperature. LA is superior than other plastics in many ways, particularly those related to the environment. PLA is utilised in the culinary and medical industries in addition to being widely employed in 3D printing and capable of being included into composites [4].

Q.2.) Are there print core codes for your machine? What do the print core codes mean?

Ans. No the terminology of print core comes only for the UltiMaker 3D printers.

A print core is a small, tool-free hot end that can be switched out. The nozzle, heater block, heater, sensor, and a tiny EEPROM chip are all located in the print cores. After being heated up inside the print core, the filament emerges from the nozzle [5].



[5]

UltiMaker 3D printers use a coding system to identify the type and function of each print core. The codes are made up of letters and numbers that indicate the nozzle size and the core type, respectively. As an illustration, "AA" denotes a general-purpose standard print core, "BB" is best suited for PVA water-soluble support material, and "CC" is meant for abrasive materials such as carbon fibre. The nozzle diameter is indicated by the code's numerals. The printers can print dual-material or material with PVA supports since they have two AA 0.4mm and one BB 0.4mm core. Cores can be combined, for example, AA 0.4mm and BB 0.8mm, if their layer heights are the same [6].

Q.3.) What is the printer type you chose for the printer brand?

Ans. Printer type- Ender-3 pro

Brand- Creality

Q.4.) What is the machine build envelope for your machine?

Ans. The Ender 3s' small 220 x 220 x 250 mm build volume makes it nearly ideal for use as a desktop 3D printer. For the majority of the products that a hobbyist or beginner might want to print, its build volume is enough.

Naturally, you may always go larger than the default measurements because the bed's actual size is 235 x 235 mm. Even though the printable area is 15 mm smaller in both directions than what is claimed, many customers have had no trouble using the entire surface.

Even though they're not as well known, all Ender 3s are excellent at making use of available space; the total gadget measures only 440 x 410 x 465 mm [7].

Q.5.) What is the G code flavour?

Ans. Marlin

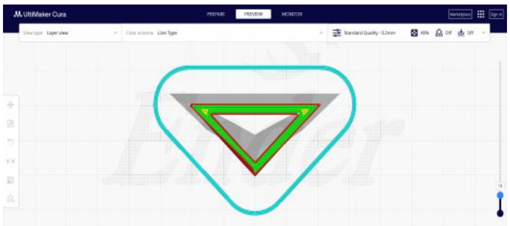
Q.5.) List the line type and colour scheme – show this for layer 1, 10, and 37 for one part.

Ans.

Line type	**Color**
Outer wall	Red
Inner wall	Green
Infill (Base surface)	Yellow
Skirt	Blue
Support	Blue

Layer-1

Layer-10



Layer-37

Comparative Analysis of 3D Printing Parameters Across the whole Project

The aim of this study was to optimize 3D printing parameters for different CAD models using Cura software on the Ender-3 Pro printer, with a focus on materials such as ABS and PLA. The projects involved the creation of a multi-face triangle, a hemisphere with an engraved name, an oloid, and an angle block set. The study systematically varied parameters such as orientation, infill density, wall thickness, top/bottom thickness, and infill pattern to identify optimal settings for minimizing build time, material usage, and non-value-added time while achieving high-quality prints.

Materials Used: Two primary materials, ABS and PLA, were explored throughout the study. ABS, known for its durability, impact resistance, and ability to withstand high temperatures, was used in the first and third projects. On the other hand, PLA, characterized by its ecological friendliness and ease of printing, was employed in the second project. The characteristics, uses, and environmental impact of both materials were discussed, providing a comprehensive understanding for the reader.

Printer Type and Build Envelope: The chosen 3D printer for the study was the Ender-3 Pro, manufactured by Creality. The Ender-3 Pro's build volume of 220 x 220 x 250 mm was deemed suitable for desktop 3D printing, with the actual bed size measuring 235 x 235 mm. The compact design and overall dimensions of 440 x 410 x 465 mm made it an ideal choice for the study.

Comparative Analysis:

Orientation Influence: In the multi-face triangle project, the X0 Y90 Z0 orientation proved to be optimal, demonstrating the shortest build time, minimal support material consumption, and the best surface finish. In contrast, the hemisphere project identified the X180 Y0 Z0 orientation as the most beneficial, emphasizing the intricate relationship between orientation changes and printing factors. The angle block set further supported the significance of optimal orientation, with X0 Y0 Z0 providing minimal build time, reduced support material consumption, and superior surface finish.

Infill Density Impact: Across all projects, increasing infill density led to higher build times and material usage. The recommended default of 20% infill without support consistently appeared as a good compromise between strength and efficiency.

Wall Thickness and Line Count: The influence of thicker walls and increased line count on structural integrity was evident in all projects. Balancing the need for strength while optimizing efficiency remained a common theme. The recommended configuration generally involved a 1.2mm wall thickness with 3 wall lines.

Infill Pattern Analysis & Fill Angle Exploration: Infill pattern analysis revealed the superiority of the Grid pattern over Octet and Gyroid in the multi-face triangle project. However, the hemisphere project favored the lightning pattern. The oloid project highlighted the efficiency of the Tri-Hexagon infill pattern. Fill angles showed minimal impact in most cases, emphasizing their less critical role in smaller structures.

Conclusion: The comparative analysis across multiple 3D printing projects using different materials and geometries yielded valuable insights. The study emphasized the importance of optimal orientation, infill density, wall thickness, and infill pattern in achieving efficient and high-quality prints. The recommended configurations provide practical guidelines for users working with similar materials and printers. Additionally, the inclusion of environmental considerations and future work recommendations contributes to a holistic understanding of 3D printing processes.

In summary, this study enhances the understanding of 3D printing parameters, contributing to the efficient utilization of resources and improved printing outcomes across diverse projects and materials.

INDIVIDUAL ASSIGNMENT

Here are some virtual sets of experiments that are assigned with different parameters on CAD model in Cura software.

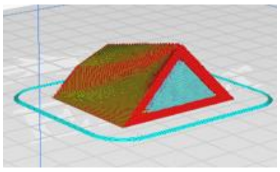
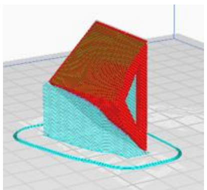
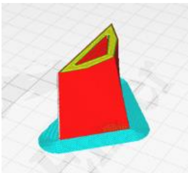
Orientation

Data & Analytics: Standard parameters used for this study are:

Parameter	Value	Parameter	Value
Material	ABS	Shell thickness	0.8mm
Nozzle diameter	0.4mm	Printing temp	200 °C
Resolution	Standard quality (0.2mm)	Build plate temp	60°C
Infill density	20%	Print speed	50 mm/s
Infill pattern	Grid		

Influence on parameter: This study carefully investigated different orientations to evaluate their effects on several parameters. Remarkably, the 3rd orientation (X0 Y90 Z0) was found to be the most advantageous, exhibiting the shortest build time together with the lowest consumption of support material and the least amount of mass waste of support material. Moreover, out of all the examined locations, this orientation had the nicest surface finish. On the other hand, deviating from this orientation lengthened the build time and required more support material. As a result, these differences increased the need for construction material and brought attention to the complex interplay between orientation changes and important printing factors concerning productivity, material usage, and surface quality.

With support

Orientation	X0 Y0 Z0	X90 Y0 Z90	X0 Y90 Z0
Figure			
Infill (min.)	0	3	3
Inner walls (min.)	6	13	14
Outer walls (min.)	7	14	14
Retractions (min.)	19	20	8
Skin (min.)	25	5	2
Skirt (min.)	0	1	0
Support (min.)	7	18	7
Support Interface (min.)	5	3	0
Travel (min.)	6	9	9

Total Build time (min.)	80	91	62
Non-value-added time* (min.)	32	22	19
Build material mass (g)	9	10	7
Support material mass (g)	2	3.5	1
Surface Comments	Near to Smooth	rough	Smooth

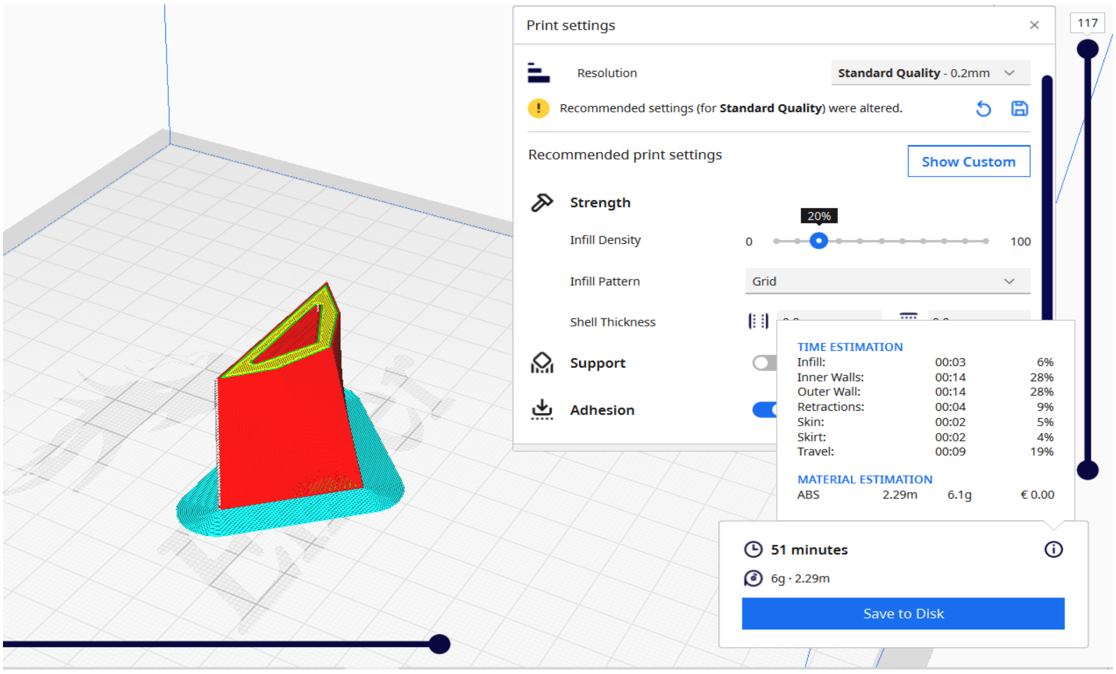
* Non-value-added time = Retractions + Skirt + Support + Travel + Support Interface

Without support

Orientation	X0 Y0 Z0	X90 Y0 Z90	X0 Y90 Z0
Infill (min.)	0	3	3
Inner walls (min.)	6	13	14
Outer walls (min.)	7	14	14
Retractions (min.)	5	13	4
Skin (min.)	25	5	2
Skirt (min.)	0	1	2
Travel (min.)	6	9	9
Total Build time	52	73	51
Non-value-added time* (min.)	13	32; without adhesion	15
Build material mass (gram)	7	8	6
Support material mass (gram)	0	In this case support is must	0
Surface Comments	Near to Smooth	rough	Smooth

Best case simulated as per Orientation: -

[X0Y90Z0]; with No support & Smoothest surface



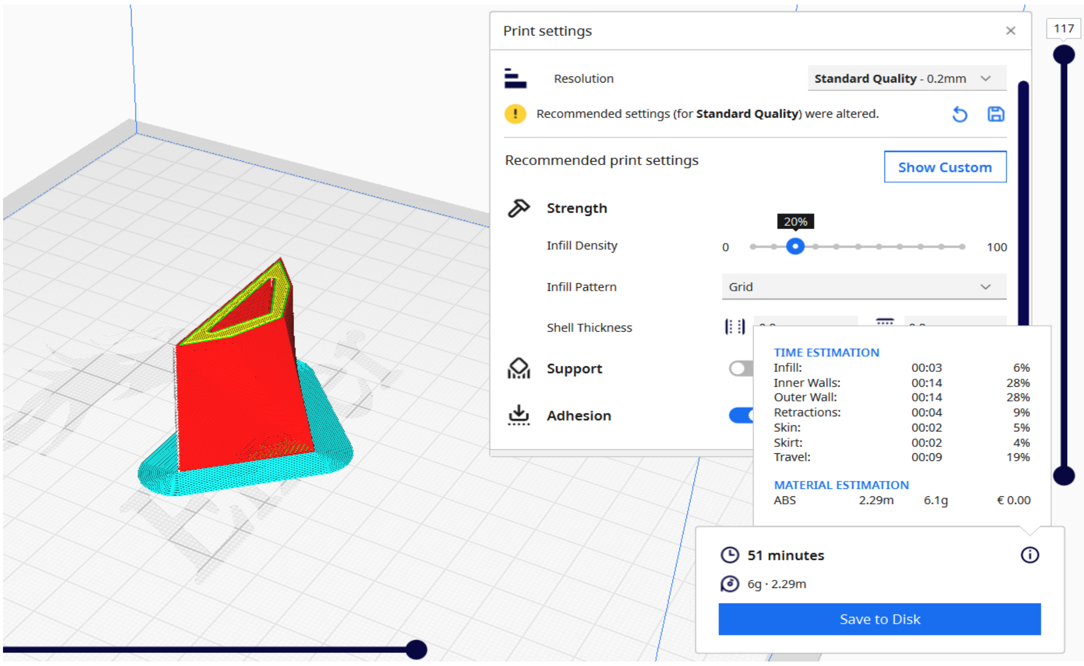
Infill density

Infill density	10%	20%	30%	40%	50%
Total Build time(min.)	46	51	54	56	58
Non-value-added time*(min.)	13	15	17	17	17
Build material mass(g)	6	6	6	7	7
Support material mass(g)	0	0	0	0	0

__Influence on parameter__: Infill means the material is being poured inside the print. So, if infill density is increased, then build time and build mass is also going to increase as shown in table. Increased build time, non-value-added time, and build material are caused by an increase in the infill percentage. This strengthens the hull and is selected according to the strength needed for the intended use. On the other hand, it has no bearing on the amount of support material used.

Best case simulated as per Infill: -

20% Infill, without any support; least material wastage and smoothest formation



Wall thickness and wall line count

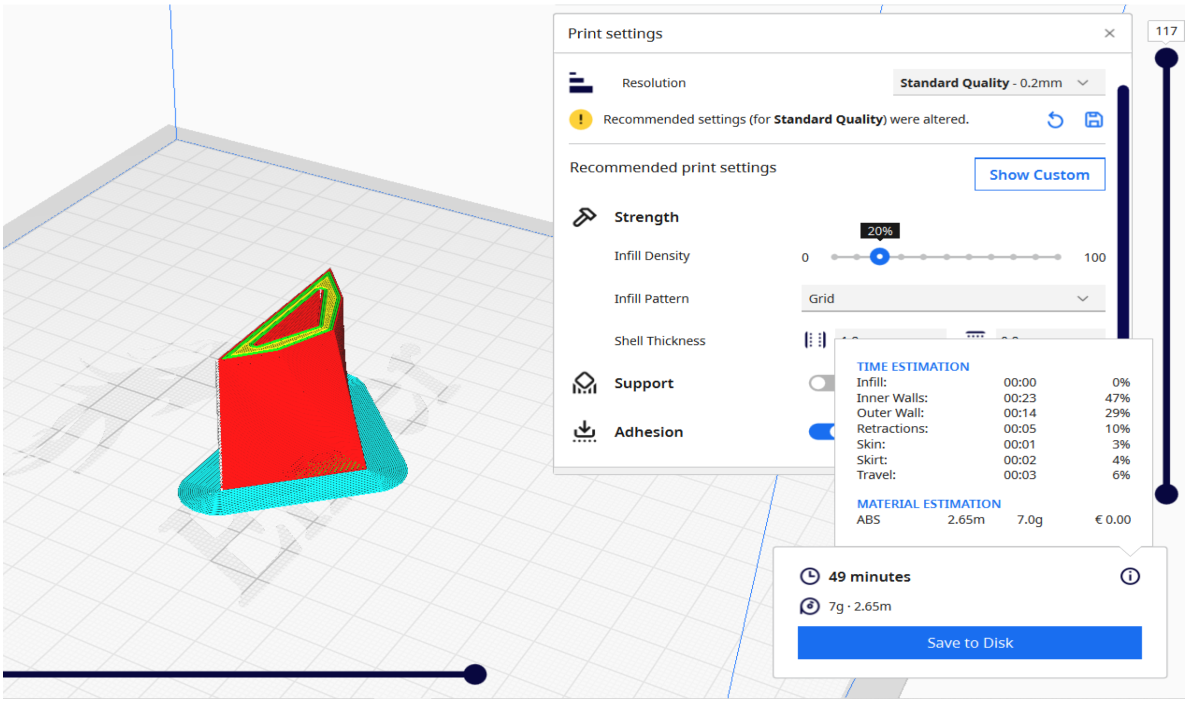
Wall Thickness(mm)	0.8	1.2	1.4	1.6	1.8
Total Build time(min.)	51	4	49	49	49
Non-value-added time*(min.)	15	10	10	9	9
No. of layers	2	3	3	4	4
Build material mass(g)	6	7	7	7	7
Support material mass9g)	0	0	0	0	0

<u>Influence on parameter</u>: Among the important characteristics that influence other parameters are wall thickness and wall line count. The build material and overall build time are definitely increased, just like the wall thickness is raised. Conversely, non-value-added time has demonstrated the value decline.

Best case simulated as per Wall thickness and wall line count: -

20% Infill, without any support; least build and non-value-added time, least material wastage and smoothest formation

[Consideration of 1.2m thickness]-3 layers



Top / Bottom thickness

Considered best case for instance - without support; Wall thickness- 1.2mm

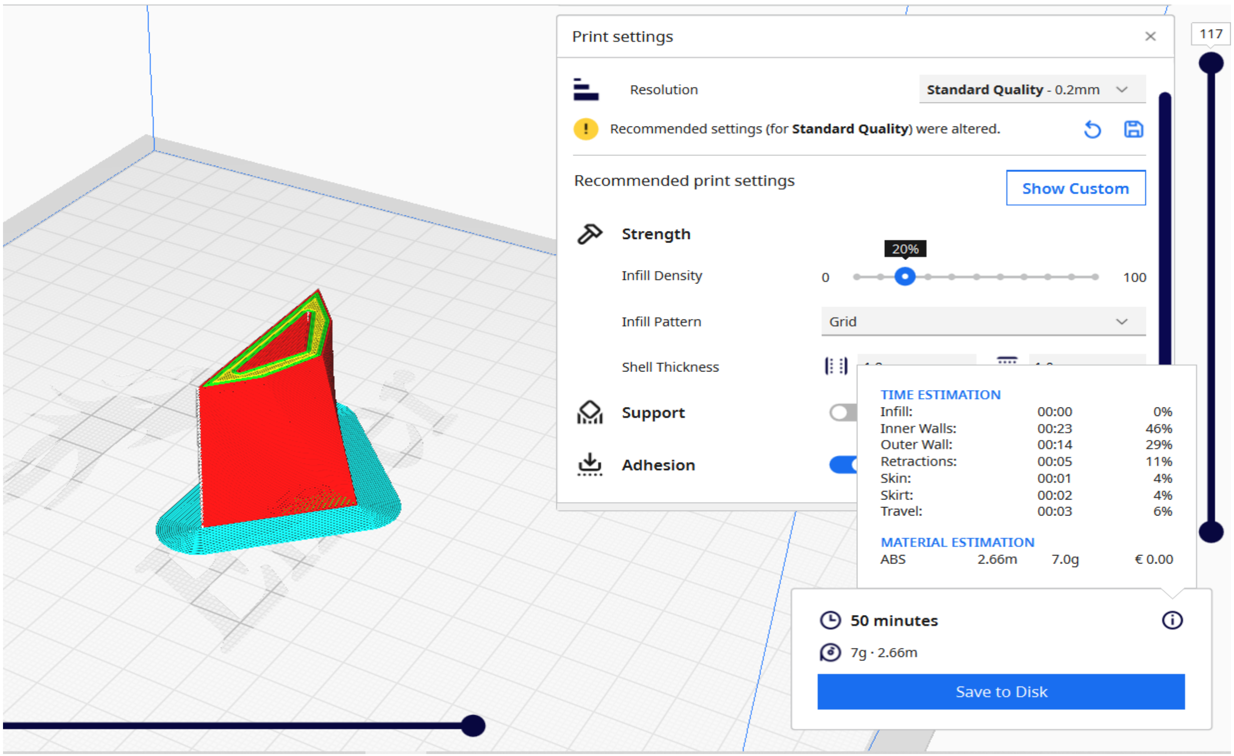
Top / Bottom Thickness(mm)	1	5	10	15	20
Total Build time(min.)	50	51	52	54	5
Non-value-added time*(min.)	10	10	11	11	13
Build material mass(gram)	7	7	7	7	16
Support material mass(gram)	0	0	0	0	0

Influence on parameter: The total number of layers increases in tandem with an increase in top/bottom thickness, which also results in an increase in build time and material. As the model rests in its default orientation, the mass of the support material stays zero. The model's quality and surface finish can be altered by varying the number of layers and their thickness at the top and bottom.

Best case simulated as per Top / Bottom thickness: -

20% Infill, without any support; least build and non-value-added time, least material wastage and smoothest formation

[Consideration of 1 mm Top/Bottom thickness]

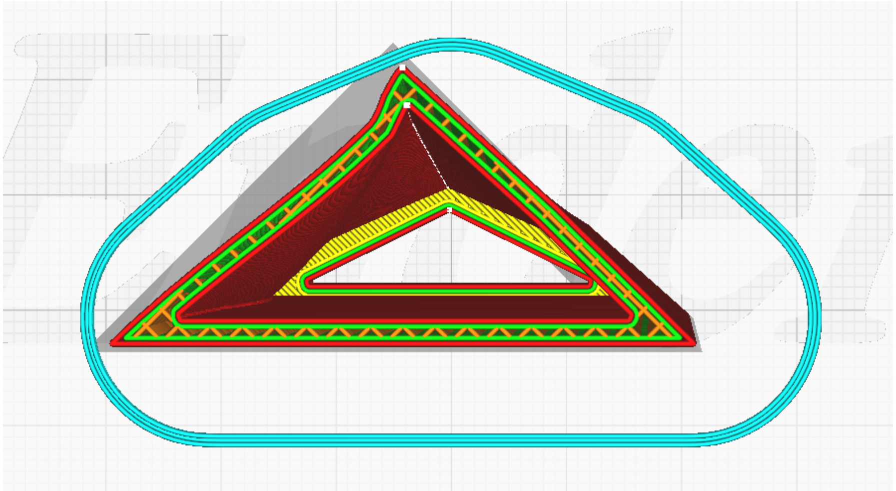


Infill Pattern

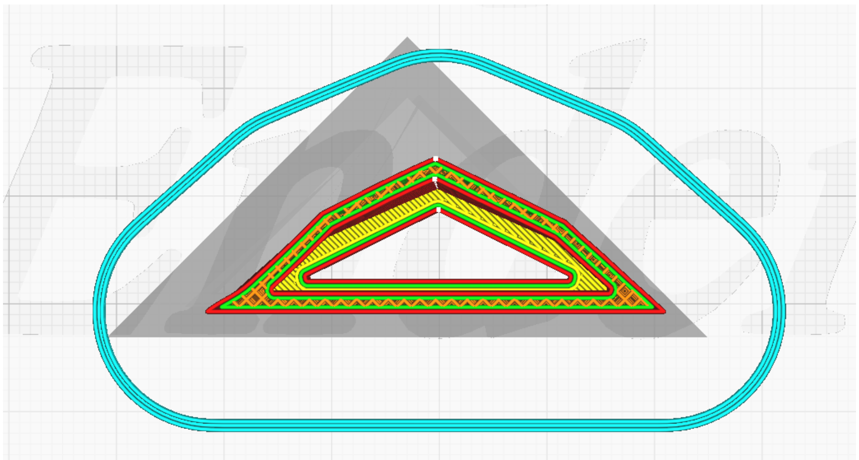
Considered-Wall thickness- 1.2mm Top/Bottom thickness-1mm

Configuration	Grid	Octet	Gyroid
Total Build time (min.)	49min.	49min.	49min.
Non-value-added time*(min.)	9m	9m	9m
Build material mass (g)	7g	7g	7g
Support material mass	0g	0g	0g

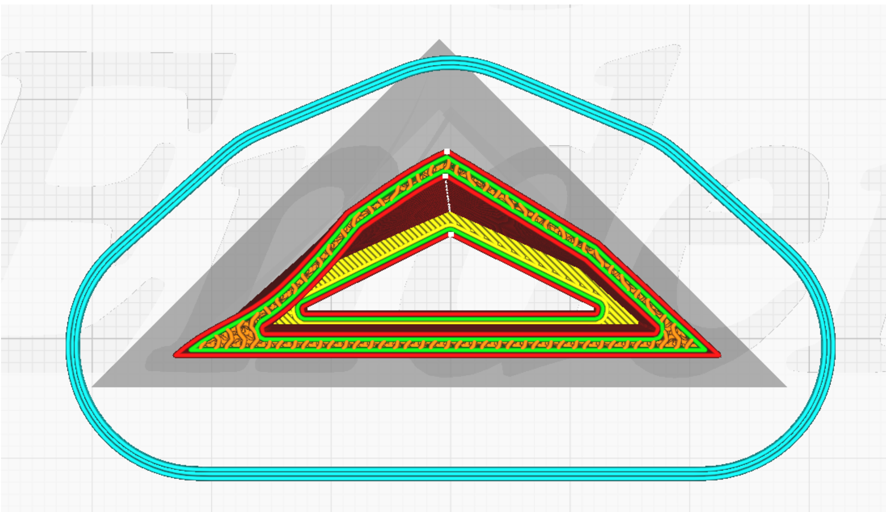
__Influence on parameter__: Here as it can be seen that all of the infill patterns have same stimulated time, mass and Non-value added time; this suggests if the product is small in shape with less wall thickness and lesser infill to be induced as such to Multi-face model then the effect of change of infill patterns is minimal on the outcome.



Grid



Octet



Gyroid

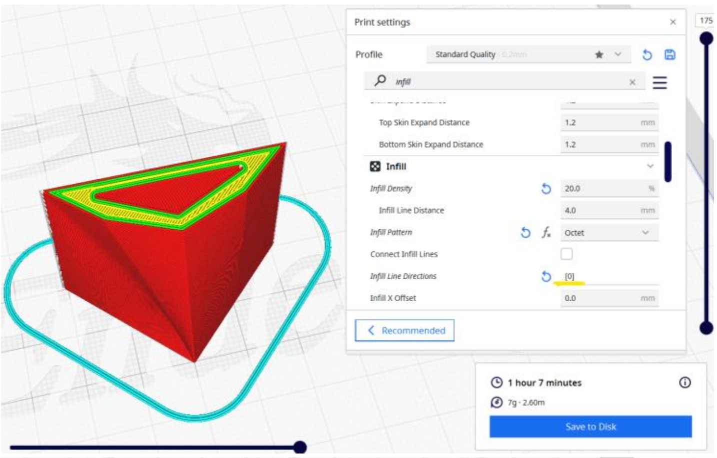
Exploring the Fill angle

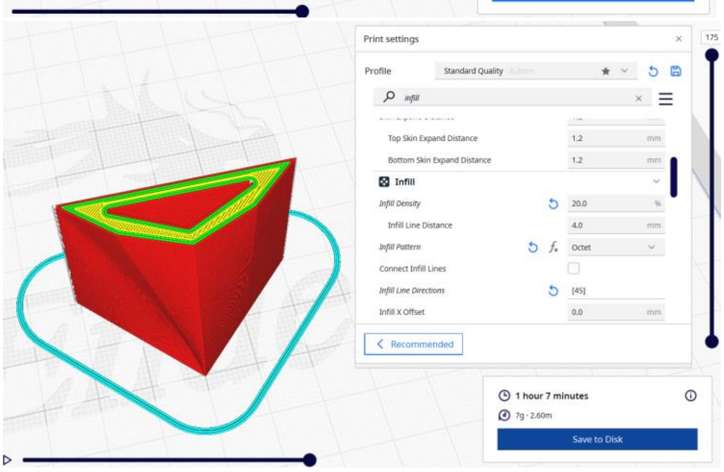
Considered-Wall thickness- 1.2mm Top/Bottom thickness-1mm

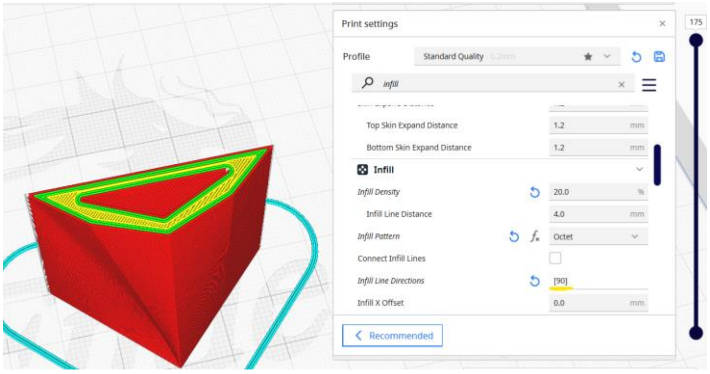
Configuration	0°	45°	60°	90°
Total Build time (min.)	67	67	68	67
Non-value-added time*(min.)	12	12	13	12
Build material mass (g)	7	7	7	7
Support material mass(g)	0	0	0	0

Influence on parameter: Here as it can be seen from various consideration that all of the infill line directions which can be understood as Fill angle have same stimulated time, mass and non-value-added time with just some minute changes in 60° Infill line directions; this suggests if the product is small in shape with less wall thickness and lesser infill to be induced as such to multi-face triangle model then the effect of change of infill patterns is minimal on the outcome and can be rarely noticed with any kind of difference.

Wherever; when I tried and simulated the same kind of simulation for bigger structures such as Oloid and Angle block set the difference in the infill angle and line directions can be clearly seen as it carries large amount of infill in it for elaboration and proper structure formation.

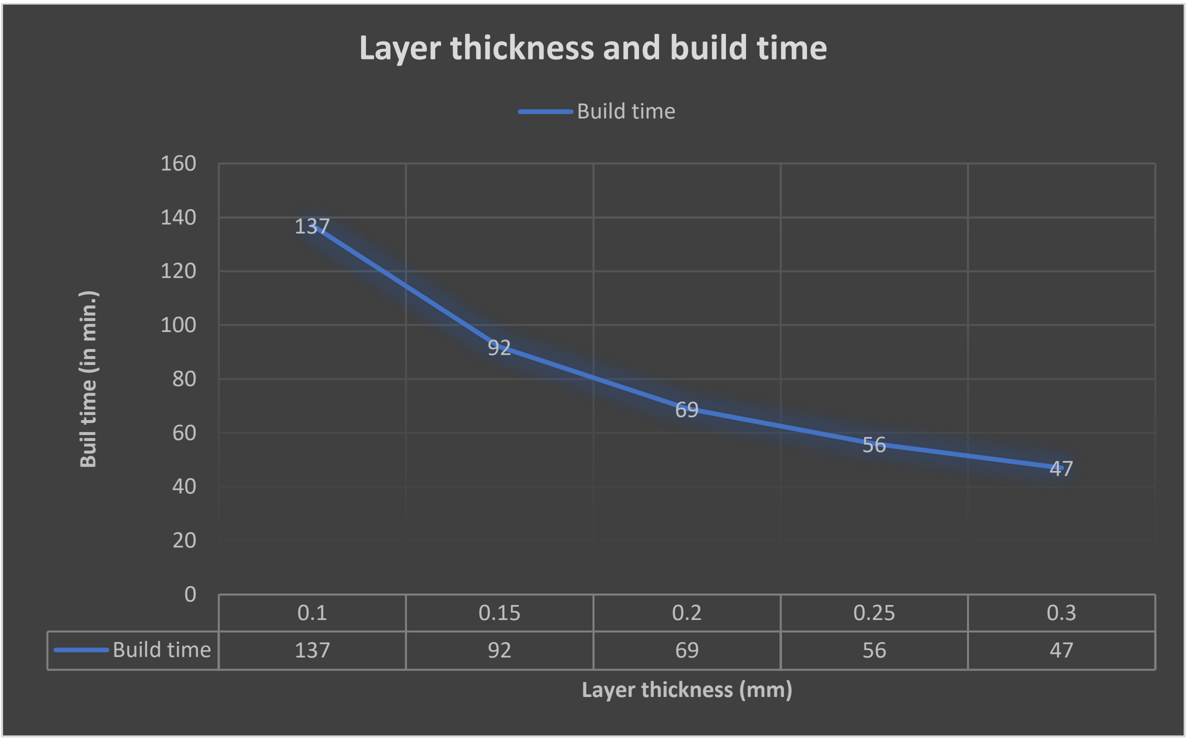




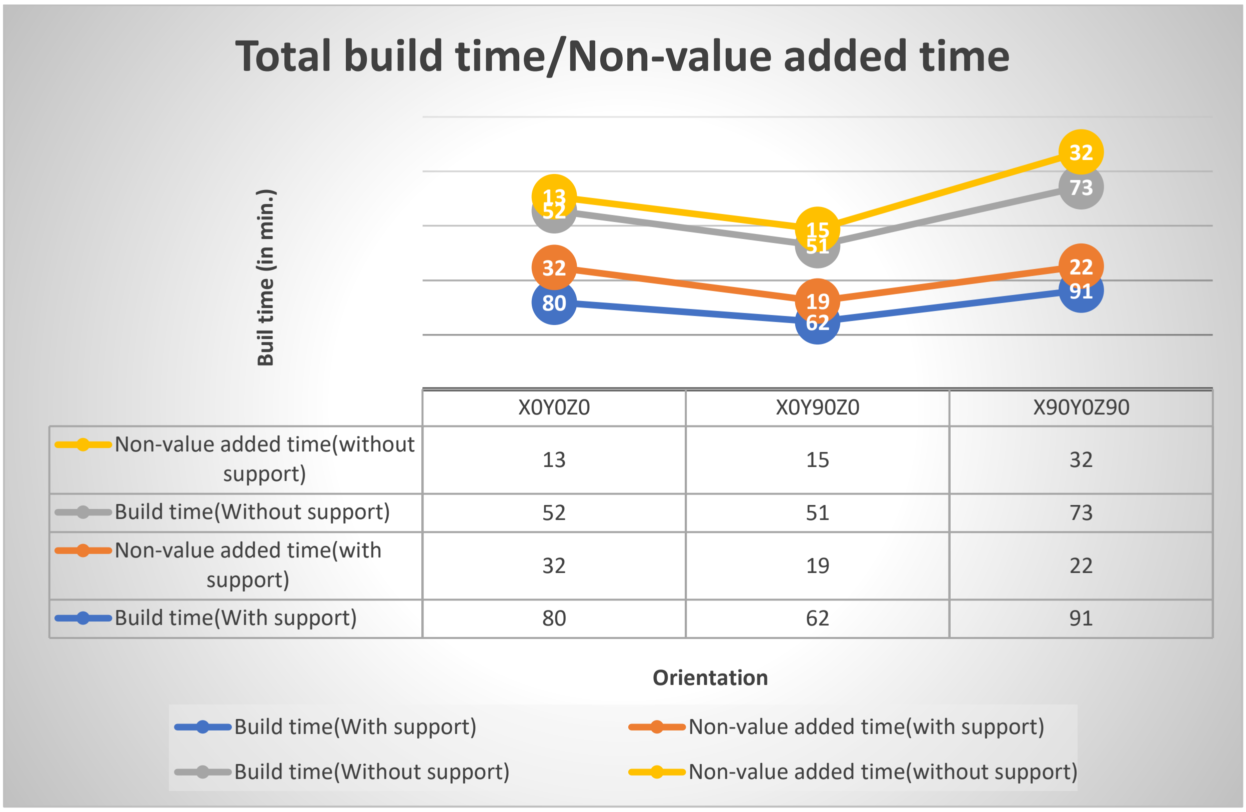


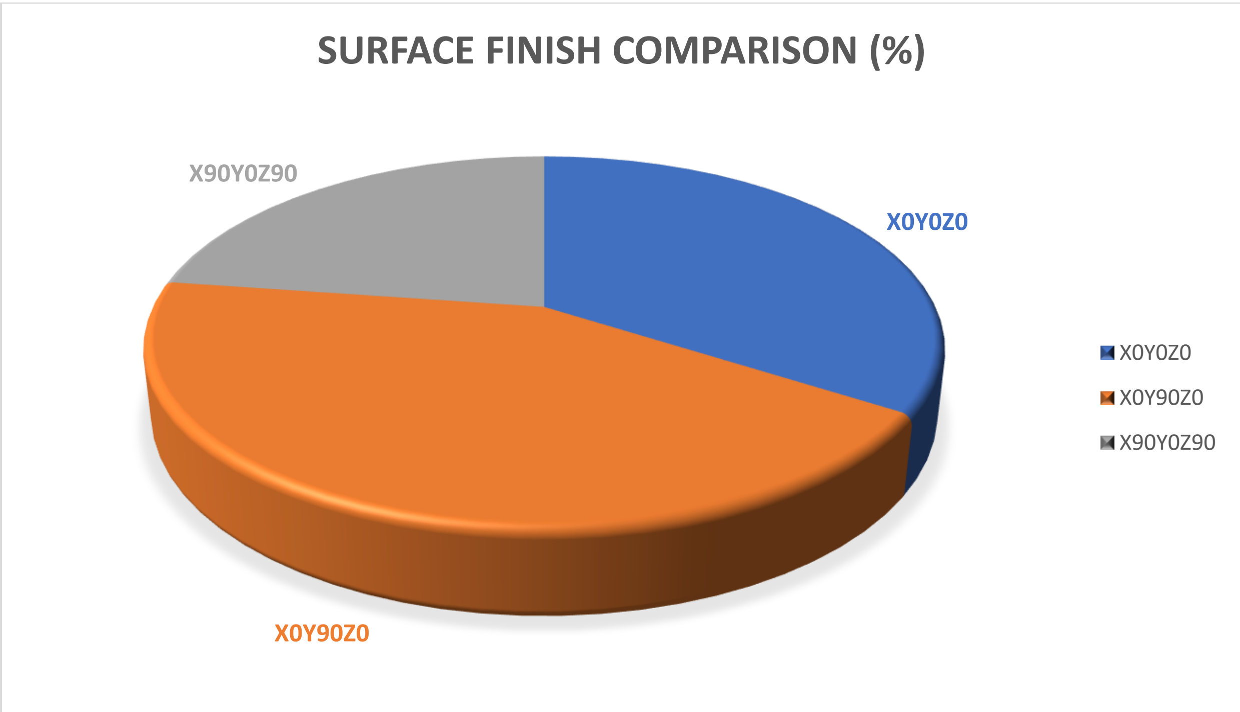
Graph observations

• Layer thickness and build time

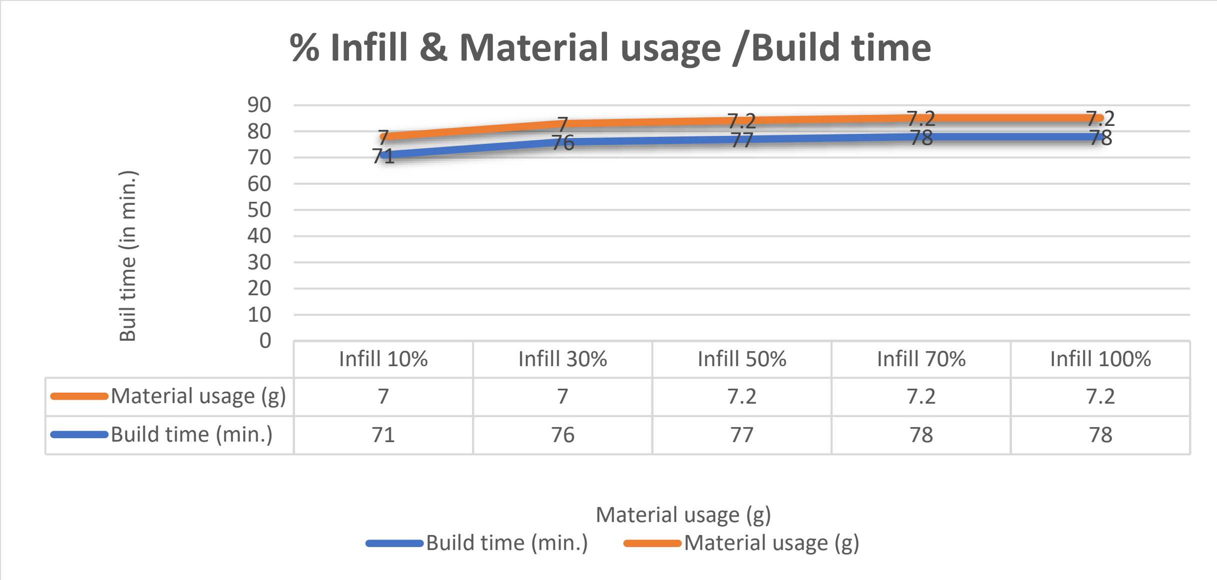


- **Illustrate the influence of orientation**



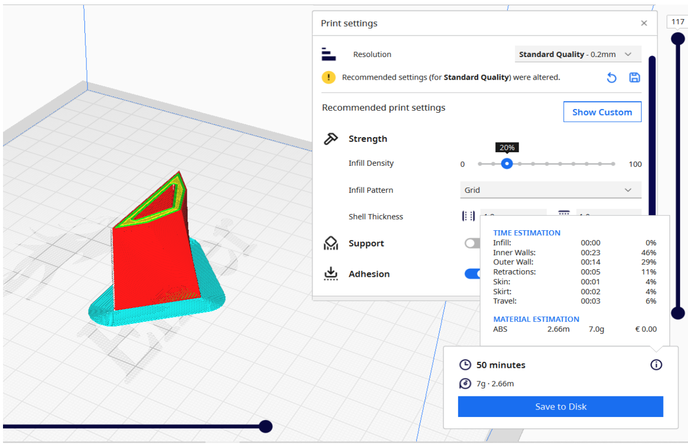


- **Illustrate the influence of the percent infill on the material usage, build time**



Using all the final parameters done above, we can do a final study using the best parameters from the top study:

Parameter	Value	Parameter	Value
Material	ABS	Shell thickness	1.2mm
Nozzle diameter	0.4mm	Printing temp	200 °C
Resolution	Standard quality (0.2mm)	Build plate temp	60°C
Infill density	20%	Print speed	50 mm/s
Infill pattern	Grid	Top/Bottom thickness	1 mm



Final Parameters:

Total Build time (minutes)	50
Non-value-added time*(minutes)	10
Build material mass (g)	7
Support material mass (g)	0

Question-1: Discuss issues with modifying the wall thickness compared to the wall line count

Thinner walls could have an effect on the part's structural soundness. Walls that are too thin could be fragile and more prone to cracking, while walls that are too thick could require more materials and weigh more. The wall line count affects the internal structure of the part. Reducing the number of wall lines in a part can weaken it and make it more prone to deformation or failure when under stress.

Question-2: Discuss the influence of the top / bottom thickness. How does the geometry influence the results for the wall thickness and top / bottom thickness?

The data above indicate that as it increased, so did the overall build time and the amount of build material need. Several model studies have shown that in some positions, the model needed support if the orientation was altered. Changing any of the parameters will result in an increase in the overall build time and material if we alter the shape and then increase the top/bottom or wall thickness. Increasing the thickness at the top and bottom usually results in longer print times and more material used, which reduces efficiency.

Question-3: What infill strategy would you recommend as a default strategy and why?

Experimental results unequivocally demonstrate that the Grid Infill pattern stands out for its efficient support provision and minimal time consumption. In contrast, the Octet, Grid, and Gyroid patterns consistently exhibit superior structural integrity. The findings affirm the Grid Infill's efficacy in optimizing printing time while ensuring robust support structures. This insight highlights the potential for enhanced printing efficiency and structural reliability by strategically selecting infill patterns based on specific performance criteria.

Question-4: What features should the software have to be more effective?

To enhance effectiveness, Cura Slicer should offer intuitive user interface, precise layer controls, and customizable support structures. Advanced features like adaptive layering, smart infill algorithms, and real-time print simulation would optimize print quality. Seamless integration with various 3D printers, continuous updates for new materials, and automated settings adjustment based on user preferences would further elevate Cura Slicer's usability and overall performance.

Project Report Summary: Optimizing 3D Printing Parameters Using Cura Software

Introduction:

The study aims to optimize 3D printing parameters for an ABS material CAD model using Cura software. The goal is to identify optimal settings that minimize build time, material usage, and non-value-added time while achieving a smooth surface finish.

Orientation Influence:

The X0 Y90 Z0 orientation demonstrated the most advantageous results, exhibiting the shortest build time, minimal support material consumption, and the best surface finish. Deviating from this orientation increased build time and support material usage, emphasizing the intricate relationship between orientation changes and printing factors.

Infill Density Impact:

Increasing infill density led to higher build times and material usage, showcasing the importance of selecting infill based on the intended strength needed. The recommended default is 20% infill without support for optimal results.

Wall Thickness and Line Count:

Thicker walls and increased line count positively influenced structural integrity but led to longer build times and more material usage. A balance is needed to maintain strength while optimizing efficiency.

Top/Bottom Thickness Significance:

Higher top/bottom thickness increased build time and material usage, emphasizing the need for careful consideration to avoid compromising efficiency.

Infill Pattern Analysis:

The Grid infill pattern consistently outperformed Octet and Gyroid, providing efficient support structures and minimal time consumption. The choice of infill pattern significantly influences printing time and structural reliability.

Fill Angle Exploration:

Variations in fill angles showed minimal impact on build time, material usage, and non-value-added time. Larger structures may exhibit more noticeable differences in infill angle effects.

Software Enhancement Recommendations:

Cura Slicer could benefit from an intuitive interface, precise layer controls, and customizable support structures. Advanced features like adaptive layering, smart infill algorithms, and real-time print simulation would enhance overall performance and user experience.

Conclusion:

The optimal configuration for the project involves X0 Y90 Z0 orientation, 20% infill without support, 1.2mm wall thickness with 3 wall lines, 1mm top/bottom thickness, and Grid infill pattern. This configuration yields a total build time of 50 minutes, minimal non-value-added time, and efficient material usage.

Recommendations for Future Work:

Future work could focus on exploring advanced infill patterns, optimizing support structures, and incorporating real-time adjustments in software based on user preferences. This project enhances understanding and provides practical insights for optimizing 3D printing parameters, contributing to the efficient utilization of resources and improved printing outcomes.

References

[1] J. Y. Hoo, Y. Kumari, M. F. Shaikh, S. M. Hue and B. H. Goh, "Zebrafish: A Versatile Animal Model for Fertility Research," *National Library of Medicine,* 31 July 2016.

[2] "ABS 3D Printer Filament," [Online]. Available: https://www.terrafilum.com/products/abs/.

[3] C. V, "All You Need to Know About ABS for 3D Printing," 27 March 2023. [Online]. Available: https://www.3dnatives.com/en/abs-3d-printing-060620194/#!. [Accessed 02 February 2024].

[4] "WHAT IS PLA? (EVERYTHING YOU NEED TO KNOW)," [Online]. Available: https://www.twi-global.com/technical-knowledge/faqs/what-is-pla#FutureofPLA.

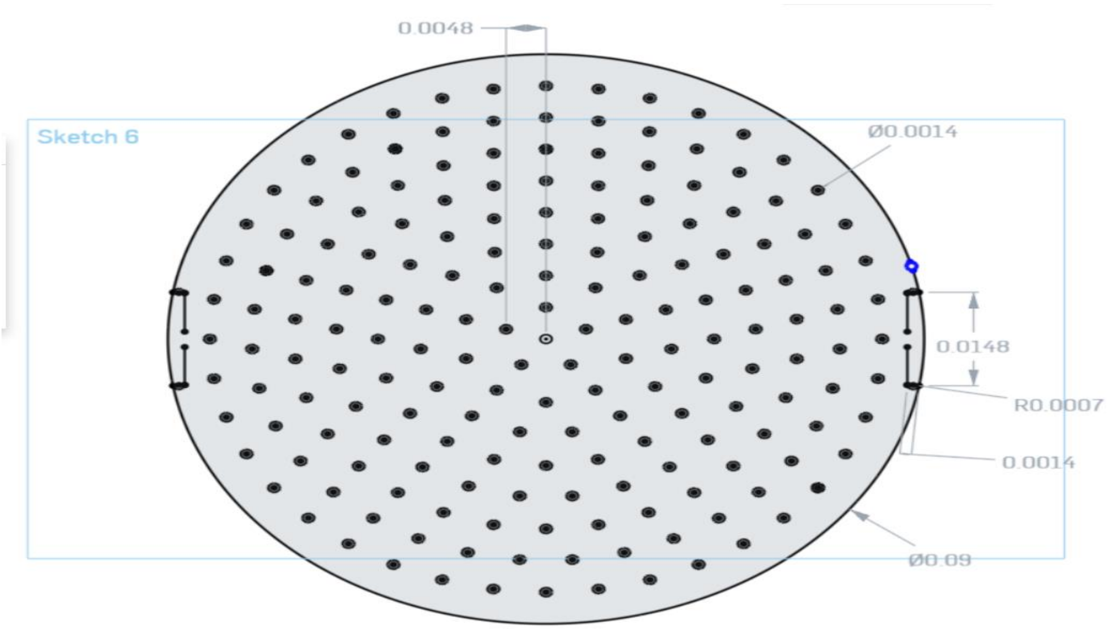
[5] UltiMaker, "UltiMaker print cores," [Online]. Available: https://support.makerbot.com/s/article/000002903. [Accessed 04 February 2024].

[6] "Which Ultimaker Print Cores Should I Use?," [Online]. Available: https://support.shop3duniverse.com/portal/en/kb/articles/which-ultimaker-print-cores-should-i-use. [Accessed 04 February 2024].

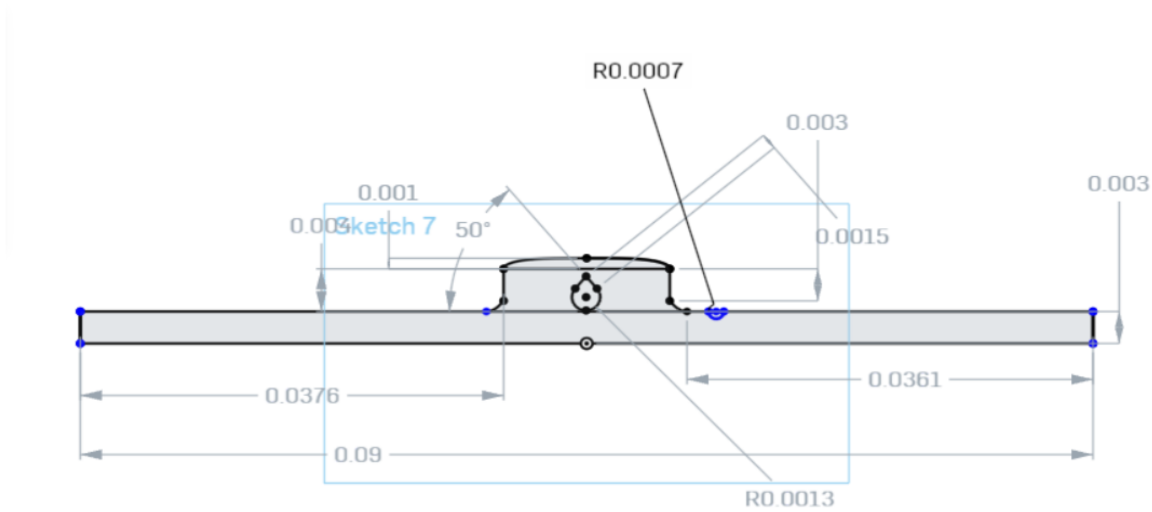
[7] P. Gharge, "ENDER 3 (V2/PRO) DIMENSIONS: WHAT ARE THE SIZES?," 21 August 2021. [Online]. Available: https://all3dp.com/2/ender-3-dimensions-how-big-is-the-3d-printer/.

Appendix A:

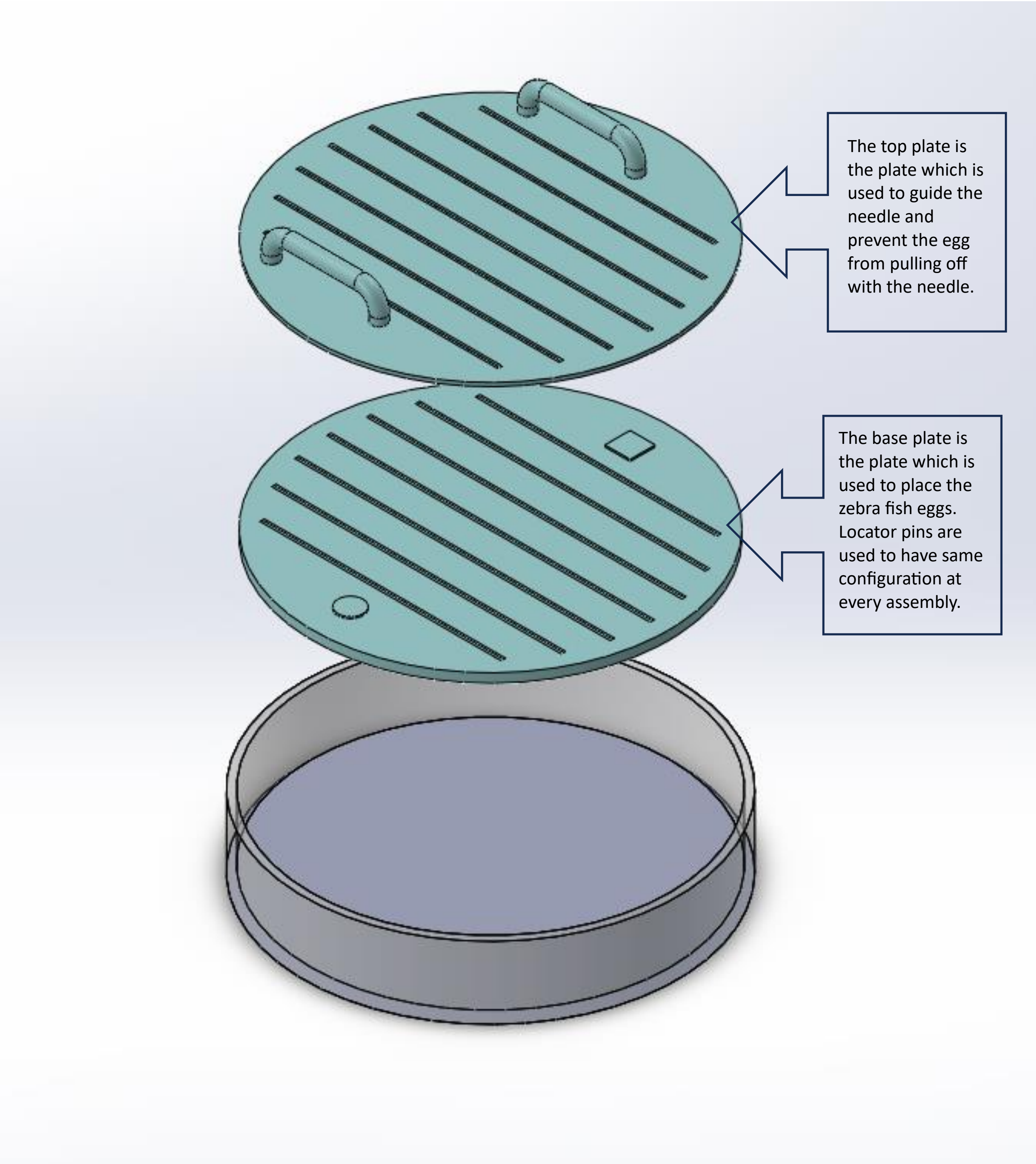
Top view:

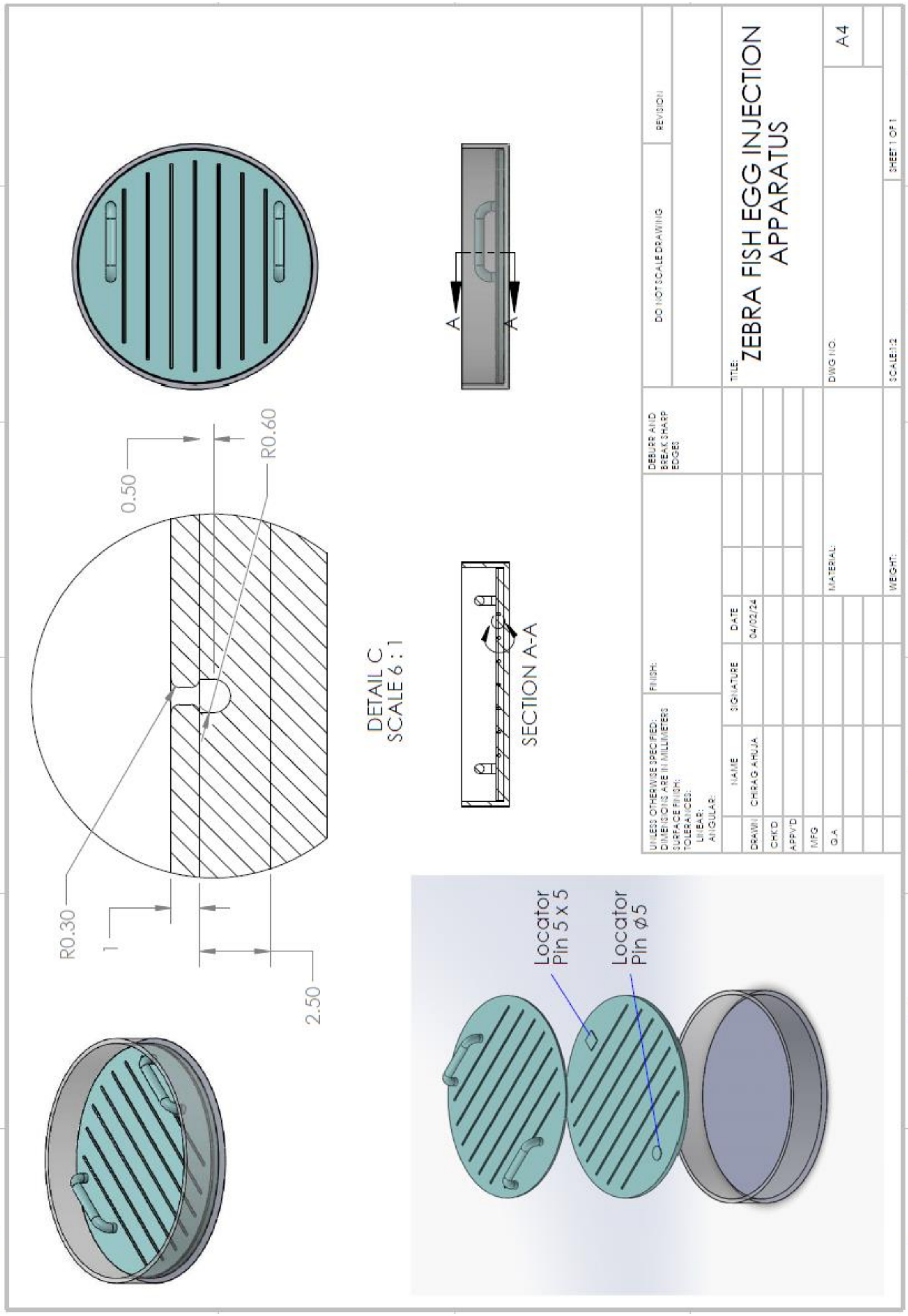


Side View:



Appendix B:


The top plate is the plate which is used to guide the needle and prevent the egg from pulling off with the needle.
The base plate is the plate which is used to place the zebra fish eggs. Locator pins are used to have same configuration at every assembly.



ZEBRA FISH EGG INJECTION APPARATUS
DETAIL C
SCALE 6 : 1
SECTION A-A
A
A
C
R0.30
R0.60
0.50
1
2.50
Locator Pin 5 x 5
Locator Pin ⌀5
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBURR AND BREAK SHARP EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHIRAG AHUJA
04/02/24
CHK'D
APPV'D
MFG
Q.A
MATERIAL:
WEIGHT:
TITLE:
DWG NO.
A4
SCALE:1:2
SHEET 1 OF 1
1
2
3
4
5
6

Appendix C:

G code file for multi-face triangle

1. **What is the bounding rectangle for the part - identify the min (X, Y, Z) and max (X, Y, Z)**

   Ans. The bounding rectangle for the hemisphere with the name is:

MIN X	81.14	MAX X	138.82
MIN Y	91.706	MAX Y	128.3
MIN Z	0.2	MAX Z	35

2. **What is the build time?**

   Ans. The total build time coming out in the G-code file is 3000 seconds i.e. 50 minutes (00:50:00)

3. **What is the extrusion temperature? Discuss the code for this.**

   Ans. The temperature to which the extruder is heated to melt the filament material into a flow stage is called extrusion temperature. There are typically two G-codes mainly used to depict extrusion temperature.

   - M104 SXXX: This code commands the machine to heat the extruder to depicted temperature has been set. However, it wouldn't wait until the temperature is reached and moves down the line to follow next command.
   - M109 SXXX – This code instructs to dwell and wait till the depicted temperature is reached.

```
G1 Z5.0 F3000 ; Move Z Axis up a bit during heating to not damage bed
M104 S200 ; Start heating up the nozzle most of the way
M190 S80 ; Start heating the bed, wait until target temperature reached
M109 S230 ; Finish heating the nozzle
G1 Z2.0 F3000 ; Move Z Axis up little to prevent scratching of Heat Bed
```

   For this code, the extrusion temperatures for M104 are 200 degrees Celsius and for M109 is 230 degrees Celsius.

4. **What is the bed temperature?**

   Ans. The print bed is also heated to the specified temperature and dwell until the temperature is reached. This is represented by code M190. As shown in image above, the bed temperature for this code is 80 degrees Celsius.

5. **What is the maximum feed rate?**

   Ans. The maximum feed rate for the code is 9000 mm/min as depicted by the code 'F9000'.

```
G0 F300 X89.494 Y120.3 Z6.8
G0 F9000 X90.078 Y120.063
```

6. **What is the minimum feed rate?**

   Ans. Ans. The minimum feed rate for the code is 300 mm/min as depicted by the code 'F300'.

7. **How many layers are there in your file?**

   Ans. There are 175 total layers in the generated file.

```
;LAYER_COUNT:175
;LAYER:0
```

8. Where are there negative E values?

```
G92 E0
G92 E0
G1 F1500 E-6.5
```

Ans. G1 F1500 E-6.5

9. Is there a G91? If so, sketch and explain what is happening?

Ans. G91 is the command which instructs the code to switch to relative coordinate mode rather than the absolute coordinate system.

Yes, G91 was used in the end of the code to retract the tool and resets back to the home position.

```
G1 F1500 E2647.44001
M140 S0
M107
G91 ;Relative positioning
G1 E-2 F2700 ;Retract a bit
G1 E-2 Z0.2 F2400 ;Retract and raise Z
G1 X5 Y5 F3000 ;Wipe out
G1 Z10 ;Raise Z more
G90 ;Absolute positioning

G1 X0 Y220 ;Present print
M106 S0 ;Turn-off fan
M104 S0 ;Turn-off hotend
M140 S0 ;Turn-off bed

M84 X Y E ;Disable all steppers but Z

M82 ;absolute extrusion mode
M104 S0
;End of Gcode
;SETTING_3 {"global_quality": "[general]\\nversion = 4\\nname = Standard Quality
;SETTING_3  #2\\ndefinition = creality_ender3pro\\n\\n[metadata]\\ntype = qualit
;SETTING_3 y_changes\\nquality_type = standard\\nsetting_version = 22\\n\\n[valu
;SETTING_3 es]\\nadhesion_type = brim\\n\\n", "extruder_quality": ["[general]\\n
;SETTING_3 version = 4\\nname = Standard Quality #2\\ndefinition = creality_ende
;SETTING_3 r3pro\\n\\n[metadata]\\ntype = quality_changes\\nquality_type = stand
;SETTING_3 ard\\nsetting_version = 22\\nposition = 0\\n\\n[values]\\ninfill_patt
;SETTING_3 ern = grid\\ninfill_sparse_density = 20\\nspeed_print = 50\\ntop_bott
;SETTING_3 om_thickness = 1\\nwall_thickness = 1.2\\n\\n"]}
```

This G-code of a 3D printing process can be understood as follows;

1. Relative Positioning (G91): Switches to relative coordinate mode for subsequent movements.
2. Wipe Movement (G1 X5 Y5 F3000) : Moves the print head to coordinates X=5, Y=5 for wiping.

3. Z Lift (G1 Z10): Raises Z axis by 10 units.

4. Absolute Positioning (G90): Switches back to absolute coordinate mode.

5. Present Print Position (G1 X0 Y220): Moves to X=0, Y=220 for the end of the print.

6. Fan and Temperature Control (M106 S0; M104 S0; M140 S0): Turns off the fan, hotend, and bed.

7. Motor Disable (M84 X Y E): Disables X, Y, and E (extruder) stepper motors.

8. Extrusion Mode (M82): Switches to absolute extrusion mode.

9. Temperature Off (M104 S0): Ensures the hotend temperature is off.

This G-code snippet reflects the end-of-print routine, including movements, temperature control, and stepper motor disable commands for a Creality Ender 3 Pro 3D printer.

10. What fan speeds are used and when do these settings occur?

```
;LAYER:3
M106 S255
```

```
;LAYER:1
M106 S85
```

```
;LAYER:2
M106 S170
```

```
G1 X0 Y220 ;Present print
M106 S0 ;Turn-off fan
```

Ans. The fan speeds used during the entire code were**: 0, 85, 170, 255**.
Whenever the machine switches to print the support interface, fan starts. I believe this is to ensure that the support material interface cools quickly and the deposited material doesn't get bonded with support material for its easy removal.

11. Estimate the material deposition rate. Hint (calculate the travel distance & use the feed rate the extruded material length). Is it constant?
Ans. To find the material deposition rate, first we need to calculate the volume of material consumed.

$$V=\pi \times r^2 \times \text{Layer Height} \times \text{Extrusion Length}$$

Where; r is the filament radius,
Average diameter of filament used on most printers is 1.75mm so let's consider the filament radius to be 1.75/2=0.875 mm.
π is a mathematical constant
Layer Height is the layer height=0.2mm
Extrusion Length is the amount of filament extruded (given by the "E" parameter) =2.60 m=2600mm
So,

$$V = \pi \text{ x } (0.875)^2 \text{ x } 0.2 \text{ x } 2600$$
$$=1250.11 \text{ mm}^3$$

Deposition rate, $Q = V / t = 1250.11 / 3000 =$ $\mathbf{0.416\ mm^3 / s}$