Optimization

As with any design, optimizing the part performance is crucial. Design optimization can be done in a number of ways, such as size optimization and topology optimization. Size optimization is often done on parametric models to fine-tune dimensions and is a relatively well-established field with many commercial software options. While size optimization is a very general method that can be applied to almost any optimization problem where constraints and objectives can be quantified, it quickly becomes unfeasible as more parameters are added. On the other hand, topology optimization can quickly indicate where material should be added or removed from a design to optimize its performance. However, topology optimization is limited in the types of constraints and objectives that can be handled and is thus not as general as size optimization.

Topology Optimization

Methods for topology optimization (TO) started being developed around the same time as additive manufacturing. Due to the organic and often quite complex shapes produced through topology optimization, the usefulness of the approach for products intended for traditional manufacturing methods has been limited, but with AM the number of feasible applications has grown. The number of commercial software intended for TO has grown quickly over the last years, as has the number of applications. At its core, topology optimization is used to minimize or redistribute material within a design space according to certain constraints and objectives, such as minimizing the compliance or deflection of the part. While TO has mainly been used for structural applications where light weight is of importance, other types of physics are possible (e.g. thermal, fluid flow, eigen frequencies). Topology optimization for AM is not limited to applications where the end product needs to be as light as possible, for instance within the aerospace industry, but is also beneficial for reducing the print cost and production time.

TO can largely be divided into 8 steps.

1. Define design space
2. Define non-design space
3. Define boundary conditions
4. Define constraints and objectives
5. Define optimization settings
6. Solve
7. Interpret the results
8. Validate.

Define Design Space

The design space is the volume within which the TO algorithm can redistribute material. In order to give the optimization algorithm, the best possible starting point, the design space should be as large as possible. An overly constrained design space will not allow the algorithm to find the optimal load paths, as illustrated in Fig. 1. A good starting point can be to look at the interfaces to other components that need to be respected. It is also beneficial to reduce the complexity of the design space as much as possible, that is, to remove unnecessary details like small rounds, threads, decorative elements or holes. Another factor in determining the design space is build size limitations of the intended machine. As with any optimization decision, iteration may be necessary to find the optimal design space if the results indicate that the optimal load carrying path exists outside of the design space.

Fig. 1 Areas where the design suggested by the topology optimization coincide with the design space boundary typically indicate that the design couldbe improved if the design space is expanded (Courtesy of Axel Nordin)

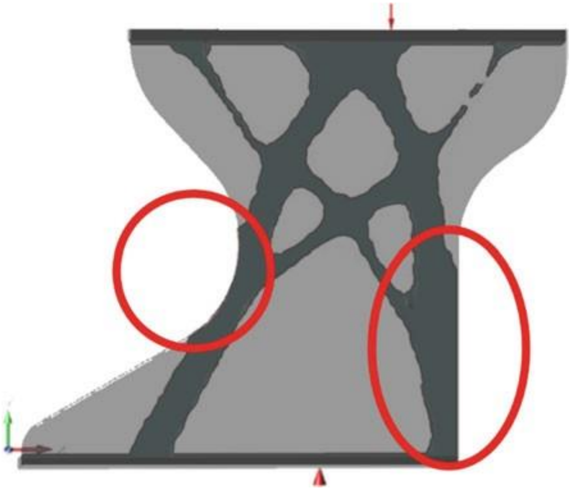


Fig. 2 Non-design space regions around areas with boundary conditions (Courtesy of Axel Nordin)

Define Non-design Space

As surfaces where the product interfaces with other parts, such as fasteners or bearings, should remain unaltered by the optimization, these should be marked as non-design space. While it sometimes is possible to simply exclude them entirely from the design space, it is often convenient to be able to apply boundary conditions on the non-design spaces, as shown in Fig. 2.

Define Boundary Conditions

As with any analysis, the boundary conditions need to be chosen with care. They should as closely as possible represent the real loading of the part. It is easy to, for instance, add a constraint that will take up forces in all directions, whereas in reality the support might only take up forces in compression. However, many topology optimization software are limited to conducting linear analyses, where, for instance, compression only supports may not be available. In those cases, special care needs to be taken to ensure that the supporting area is sufficient for the loads during validation.

As most structures will be subject to several different loads during operation, it is important to make sure that all major loads that may act on the part are represented. If one is missing, the optimization will typically recommend a design that is weak in that direction. A good strategy may be to add a new load case for each force acting on the structure.

Define Constraints and Objectives

There are a number of possible constraints and objectives that may be set up for TO. The most commonly available objectives include minimizing the compliance (*i.e.*, maximizing the stiffness) and minimizing the mass. These objectives will obviously not generate useful design by themselves and need to be coupled to constraints. For instance, minimizing the compliance is often linked to a constraint on the percentage of the volume that should be retained. Minimizing the mass is often linked to a constraint on the stresses or deformation of the part. As knowing beforehand what the optimal design should weigh is impossible, it is recommended to experiment with several percentages. It is also worth mentioning that stresses calculated during TO are only rough estimates, and do not necessarily correspond to the results calculated during validation. The topology optimization can remove or change the mechanical properties of individual elements which makes it difficult to accurately predict the stresses. Moreover, the mesh density that is suitable for TO is not necessarily suitable for stress calculations. While research is being done into how to get more accurate stress values, it is recommended to add additional constraints to the optimization, such as deflection, to get more robust results.

Apart from structural constraints, many software vendors have added support for taking AM-specific manufacturing constraints into account. One can, for instance, reduce the need for support material by adding an overhang constraint, which limits the number of surfaces with an angle below the defined support angle and build direction. As the build direction may not initially be known, however, it may be beneficial to run optimizations in several build directions to evaluate the difference in build height, amount of support material, and part performance. It common for the overhang constraint to be contradictory to the structural objectives, and thus there is often a need to compromise between the amount of support material and the part performance, depending on whether cost or performance is more important.

Define Optimization Settings

Most software will allow the user to define parameters such as mesh density and minimum feature size. The mesh size affects the level of detail that the resulting geometry will have, and of course the computation time. In theory, the finer the mesh, the more optimal the results (as for instance Michell structures shown in Fig. 3, however, computational limits and production constraints will typically mean that the mesh size is fairly coarse, compared to traditional FEA meshes. This also means that traditional TO is not suitable for generating structures with the typical lattice structures, and other approaches are needed for generating them.

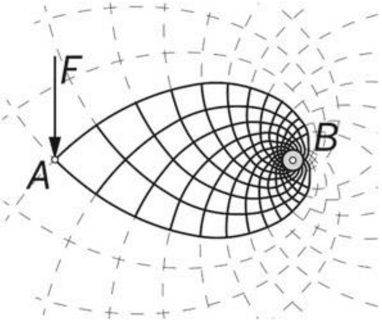


Fig. 3 A Michell structure (Courtesy of Arek Mazurek). https://commons.wikimedia.org/wiki/File:MichellCantilever.jpg

Solve

Solution time will go up with the number of load cases and the mesh density. While reducing the number of load cases to reduce the computational time may be tempting, doing so will yield results that are not useful. Reducing the number of elements, on the other hand, will not typically affect the overall distribution of material, but will only have local effects. For initial optimization of concepts, it is therefore recommended to adjust the number of elements.

Interpret the Results

The results from a topology optimization is most commonly in the form of a mesh-based model, although there are a number of experimental methods that aim to make the process from optimization to manufacturing more streamlined. While it could be possible to directly print this part, doing so is often not preferred, as the mesh from the TO is quite coarse, may have many stress concentrations, and may not fit well with interfacing components. Due to this, there is a need to remodel the mesh-based result to make it suitable for manufacturing and use. There are three main approaches for this, as illustrated in Fig. 4:

1. Mesh-based smoothing
2. NURBS interpretation
3. Manual re-design.

Mesh-Based Smoothing

There are several tools for smoothing the facet-based result and many smoothing algorithms. While this may produce satisfactory results for one-off prints, the facet-based model will not have any link to the initial CAD model, and it will be difficult and time consuming to do parametric changes to it, such as increasing dimensions.

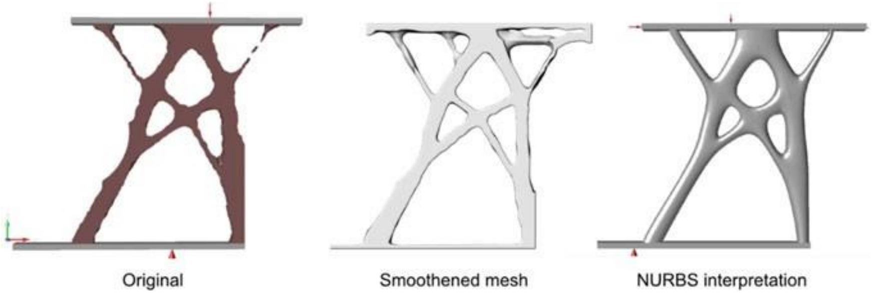


Fig. 4 Approaches for post-processing the topology optimization result (Courtesy of Axel Nordin)

NURBS Interpretation

There are several tools for reverse engineering a facet-based model into a parametric model, as discussed earlier. Some require manual input, and some are more automated, but all will require some amount of manual manipulation to get good results. Although these tools generate parametric models, the complexity of them is often very high, with few planar or cylindrical/spherical surfaces. Moreover, the model is "dead" and there is no information about the features or design intent. This makes it difficult and time-consuming to continue working with the model in standard CAD-packages, and the models are not easily parametrically controllable, if changes to the dimensions need to be done or if one wants to conduct a size optimization.

Manual Re-design

This method is obviously the most time-consuming, but also generates a model with high parametric controllability and design intent. If the model is to be further adapted for manufacturing or optimization, manual re-design is usually the most straight-forward option.

Validate

A design validation always needs to be done on the reinterpreted model as the results reported during the TO are not usually very accurate.

Generative Design for Additive Manufacturing:

1. Software Selection

Once the initial geometry evaluation for PBF-EB/M technology is completed, it is time to optimize the geometry using the computer tools. The software selected is Altair Inspire and the main reasons for this selection are explained below:

- It is an easy computer tool to navigate in its menus and it has a very intuitive interface.
- It is useful to obtain the first approximation to the optimized shape. The opti- mization process is complex and needs the model to be able to evolve in different phases and taking into account diverse factors.
- It allows obtaining fast results, avoiding the difficulties of selecting types of elements and meshing. As will be seen, this is both an advantage and a handicap when there are problems associated to the automatic mesh.

Figure 5 exhibits the workflow of Altair Inspire: geometry setting, case loads assignment, analysis and optimization, results evaluation and CAD model translation.

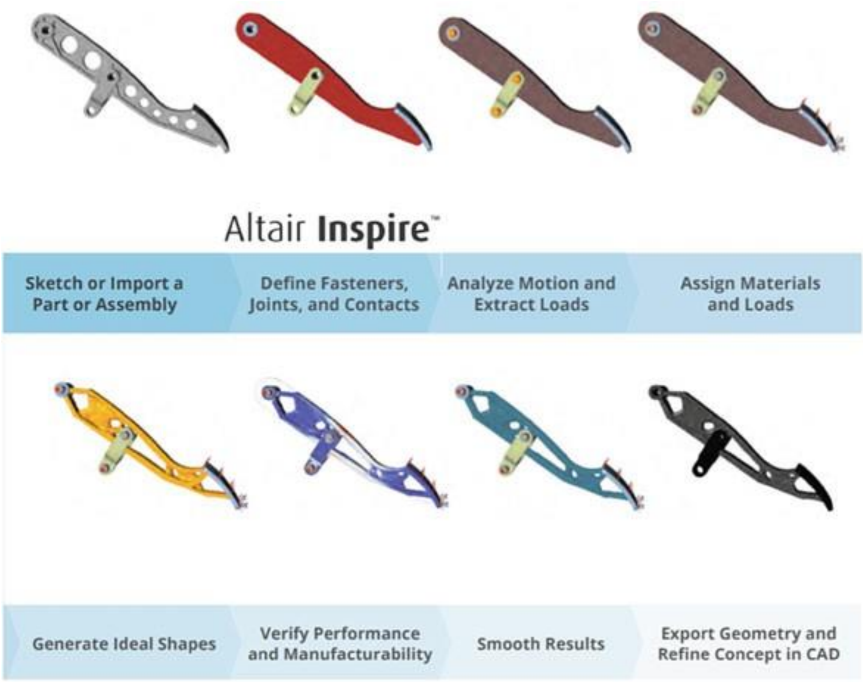


Fig. 5 Workflow of Altair inspire (Courtesy of Altair)

2. Geometry

First of all, it is important to learn the available abilities to create and modify the models. It is crucial to develop the initial geometry and widen the software´s opti- mization options. Generally, all the space available inside the maximum volume ofthe part should be included in the optimization process unless there are technical reasons not to, such as mounting space or interferences with other parts in its normal function (Fig. 6).

Simplifications are also required with these kinds of tools. In order to speed up the calculations, it is recommended that all the small fillets, rounds or chamfers that are not necessary in the structural behaviour should be deleted. This deletion should pay special attention to not include simplifications that can introduce significative errors in the results. Nonetheless, the final optimized shape should be validated afterwards through a new analysis.

Fig. 6 Altair inspire geometry tools (Courtesy of Altair)

Fig. 7 Altair inspire case setup tools (Courtesy of Altair)

Linked to the optimization process and the software layout, a distinction between design parts and non-design parts is required. Therefore, all parts that will have loads or constrains must be defined as non-design parts. To meet this requirement, the user should adapt the model using the geometry tools.

3. Load Cases Setup

After completing the geometry modifications, the load cases that will feed the soft- ware must be set in order to obtain the optimization. The typical tools to set loads such as forces, pressures or torques and constrains in any freedom degree can be easily found and applied in this software (Fig. 7), as in many other CAE software. Furthermore, in most cases, it is also necessary to set fasteners, joints or connectors to bond the parts.

Other less usual conditions that the selected software (Altair Inspire) allows us to add are imposed displacements, accelerations, temperature conditions and concentrated masses.

Contact definition between parts can also be found in this section and it offers us the possibility to define the faces in contact as bonded, contacting or without contact. In this section, the material properties to the components also need be set. The more accurate these properties are, the more trustworthy the results will be. However, it is also important to know that this software does not allow anisotropic materials setting and this is a significant deficiency because additive technologies normally require this material behaviour. The way to compensate this deficiency and assure safety is to consider the weaker properties. That is, the structural problem is solved as an isotropic material. Again, it should be mentioned that this optimization is a first stage and the final design will need to be validated.

4. Analysis/Optimization

On one hand, an option to perform structural analysis is included. Before the analysis can be launched, the user has to set the analysis parameters. The software chosen includes only a few parameters that can be modified: mesh size, contact's type andaccuracy. This is due to the simplicity and navigability of the software concept.

The analysis can be completed before and after the optimization. It is good to analyse it prior to the optimization to know the stress level or the model behaviour, in order to help the user, set the parameters in the optimization set up. The analysisis performed after the optimization to check the proposed shape in the different load cases. However, this second analysis can display high levels of stress concentrated in specific points because the raw model shape has imperfections that will be removed in the final model.

On the other hand, the optimization analysis offers different approaches depending on the objective. It can maximize the stiffness or minimize the mass. For the stiffness maximization, the program needs the mass target. For the mass minimization, the safety factor is required.

All of these analyses require the parameters of mesh size (defined as thick- ness constraints), level of accuracy (faster/more accurate) and contact's assumption (sliding only/with separation). In case of error, the user can modify the size mesh. Reducing it can help avoid meshing problems due to geometry imperfections. An alternative option is suggested: to check and correct the geometry imperfections, since this reduces the work time increase that would result if a smaller mesh size were used.

Finally, the user can select what load cases are used in the optimization. It is not possible to assign different weights to each load case directly. Different weight assignments need to be made through new load cases modifying the load.

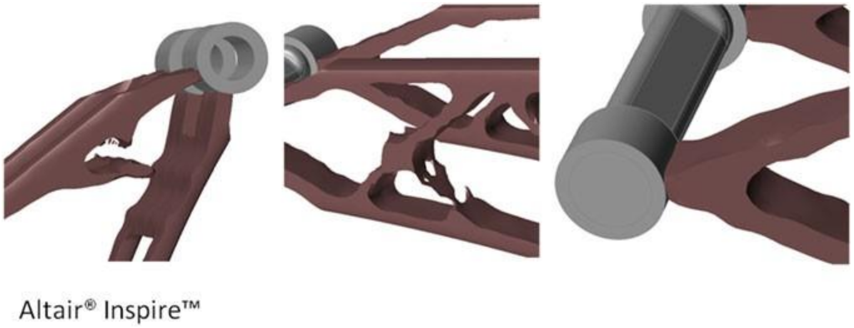
Results/Comparison

The results should be evaluated visually first. Some cases can give us incomplete shapes with reinforcing linking elements that are not fully developed. If it happens, there are two options: (1) the user can modify the result directly, increasing the mass with a slider tool included in the software, or (2) the optimization can be recalculated, increasing the level of mass or the safety factor.

The junction between design and non-design spaces is usually a week point and also needs to be evaluated to decide if it would meet the structural requirements. These issues can be pointed out as difficulties, even if they are accepted, they should be checked in the next phases (Fig. 8).

The next step is to establish the goals and to select the results that allow the evaluation and comparison in the different optimized models. The most used results are the stress level, the safety factor and the displacement. The software has utilities

Fig. 8 Optimization issues (Courtesy of Altair)



3D Model Preparation

Since the generation of a part using any GMP is primarily a conversion of CAD data to the real object the computer representation and its accuracy are of significance. Most RP systems receive their data from CAD systems in either 3D surface models or 3D solid models. RP systems require data in a particular format. Since the 3D Systems Inc., who first marketed a GMP based on Stereolithography (STL), developed an STL file forrrat, and such machines far outnumber all other types of machines, the STL format has become the de facto standard for all RP technologies. This system is based upon creating a mesh of connected 3D triangles (actually triangular laminae oriented three-dimensionally) whose vertices are ordered to indicate which side of the triangle

contains material and needs to be created in the process. Figure 1.5 indicates how triangles can beused to create surfaces and objects. Of course, the number of

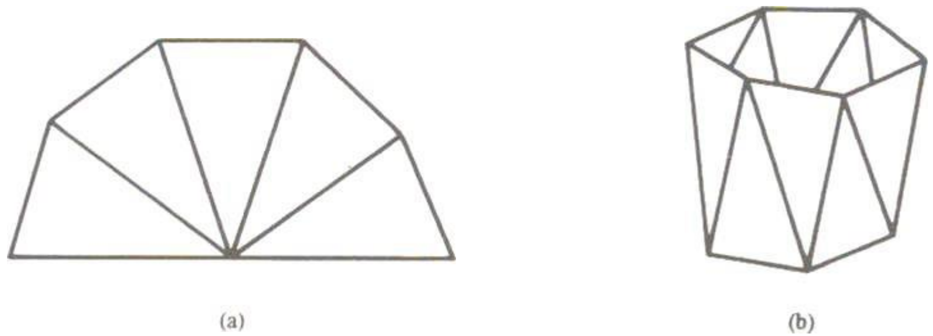


Figure 1.5: Representation of surfaces by connected triangles

triangles has to be very much larger in an actual STL file for accurate representation of the object to be created. It is also important to keep the orientations of the vertices correct to represent which sideof the triangle the material of the object exists. Figures 1.6 (a) and (b) show the correct and incorrecttriangle orientations, respectively.

The slicing of the CAD model is done by using a ray-tracing algorithm which scans through a particular z-level of the model. The resulting cross-section would be one or more closed paths and a complete representation of the area filled with material. Cross-hatching algorithms create paths for quick development of the material (either by solidification or by deposition). The orientation of the object has to be judiciously chosen for optimization of the process time and accuracy. The software needed for slicing and generation of data to control the GMP system movements is not a general oneand depends on the specific GMP system. The new file (SLI) pilots the movements of the processing unit. Figure 1.7 indicates the pre-processing of data diagrammatically.

The resolution of the CAD model depends on the accuracy desired i.e. the maximum deviation of the desired surface from the chords generated in the CAD model. Allowing smaller deviation makes the model more accurate, but the STL file size increases which leads to increased slice time and slice file size. Scaling the part geometry to take care of shrinkage (which is an unavoidable phenomenon in Stereolithography and a number of other GMP's) can be accomplished in the RP software.

The smallest feature size depends on the specific process used for RP. In processes where a laser beam is used (either for curing or for sintering or cutting) the beam diameter plays a crucial role in deciding this. It is usually in the range 0.175 mm to 0.3 mm.

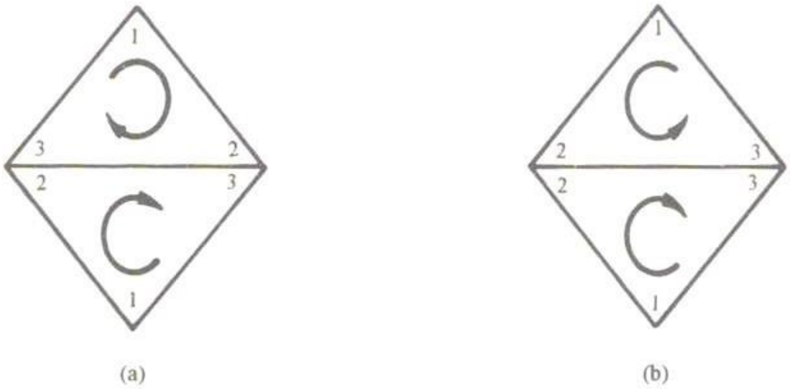


Figure 1.6: Correct and incorrect orientations of adjacent triangles

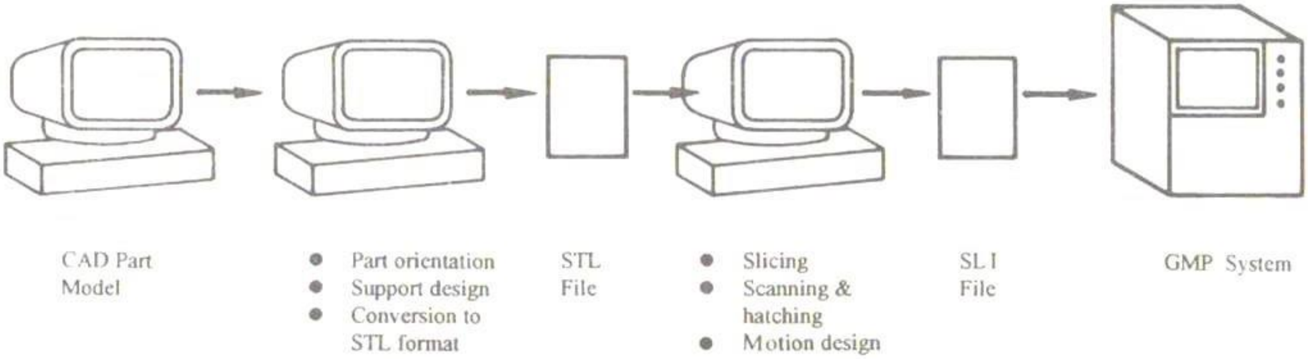


Figure 1.7: Pre-processing of CAD data

PART ORIENTATION

Choice of part orientation is important as the orientation within the processing chamber influences the build time, part resolution and surface finish. Higher resolutions can be obtained in curved surfaces by orienting them in the horizontal plane (normal to the direction of layer deposition). Figures 1.8(a) and (b) show clearly how the 'stair-step' appearance can be avoided by a proper choice of orientation. In cases where formation of curved surfaces in the direction of layer deposition is unavoidable multiple layer thickness within a building cycle is feasible. Surfaces with large slants should be developed by using thinner layers to reduce the 'stair-step' effect. The orientation can also affect the trapped volume in case of liquid-based GMP's. The trapped volume is represented by that space which holds liquid that is completely separate from the liquid in the main vat. Figure 1.9 shows how a correct choice of orientation can eliminate (or reduce) the problem.

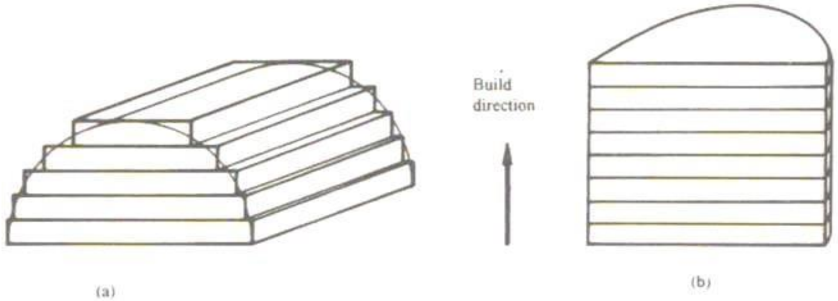


Figure 1.8: Effect of orientation on accuracy and finish

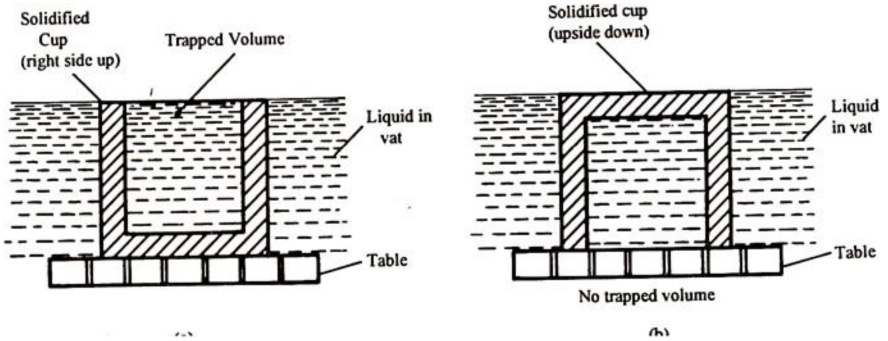


Fig. 1.9: effect of part orientation on trapped volume.

Slicing:

The slice programme converts the three-dimensional object in the STL file into two-dimensional cross-sections. The slice axis is defined as the normal to the plane created by slicing and this is also the build direction while creating the part by GMP. The thickness of slice dictates the texture, accuracy and build time. The layer thickness is normally in the range 0.0625 mm to 0.75 mm. It is, however, not correct to assume that using thicker layers (and reducing the number of layers) leads to reduced build time in all cases. In many processes the speed of scanning of the activating element (laser beam in many processes) depends greatly on the layer thickness. So, the time required for creating individual layers increases greatly when large thickness is used. Figure 1.10 shows the typical characteristics of how the build time changes when the layer thickness is gradually increased for three different power levels of the beam used. It is seen that the range 0.125 mm to 0.25 mm is the optimum irrespective of the beam power.

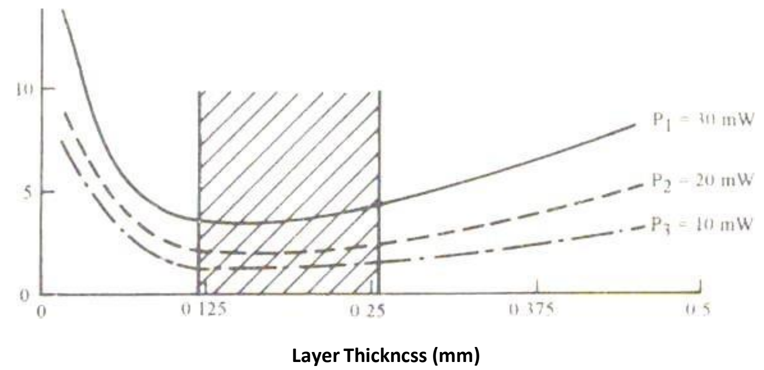
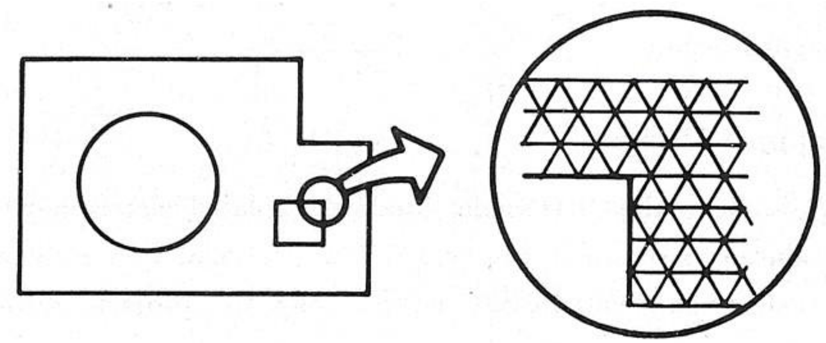


Figure 1.10: Effect of layer thickness on build time

Internal Hatching and Surface Skin Fills

To solidify (or to create) the area inside the part surrounded by the outer boundaries, internal hatching is used to reduce build time. Initially the boundary lines are created and then the interior is criss-crossed with lines, giving the part adequate internal stiffness. The style of hatching can vary. The pattern may consist of parallel lines making 0°, 60° and 120° with the x-axis resulting in an internal structure which consists of equilateral triangles as indicated in Figure 1.11. The spacing between the consecutive lines is about 0.625 mm, and this common hatching pattern is called Tri-Hatch. When liquid photopolymers are used in the process, the material trapped inside the triangles remains liquid till the part is post cured following the completion of the shaping process. Recently, a new pattern has been introduced which is called WEAVE™. In this, the scanning lines are parallel to the x- and y-axis, the spacing being about 0.28 mm when the layer thickness is about 0.25 mm. When the layer thickness is 0.127 mm,the spacing is made to be 0.229 mm. In the Tri-Hatch system too much (« 50%) liquid material remains trapped and this leads to considerable post curing distortion. Attempts to reduce the fraction of



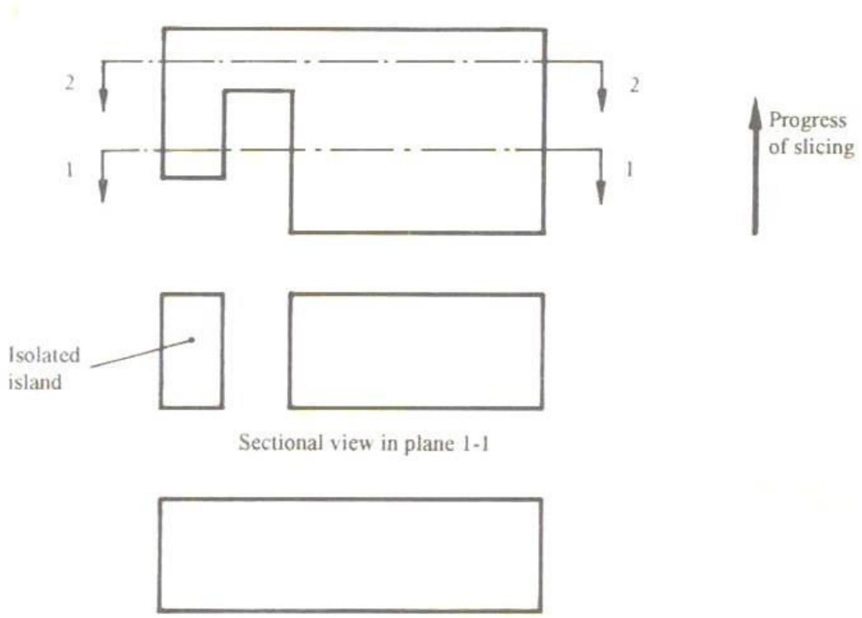
trapped volume in the Tri-Hatch system by reducing the hatch spacing lead to increased curl distortion. With the WEAVE™ system, a reduction of the fraction of trapped residual volume without resulting in large curl distortion is possible.

Figure - Tri-Hatch pattern

It is obvious that the outer surfaces of the generated solid cannot end up being porous. Thus, skins are created by skin fills which consist of closely spaced scan lines. The spacing between the scan lines is in the range 0.0762 mm to 0.127 mm. The skin fills are scanned after the borders and internal hatch. However, with the introduction of WEAVE™ the importance of skin fill has been greatly reduced since very little residual liquid remains trapped inside.

Support Structure:

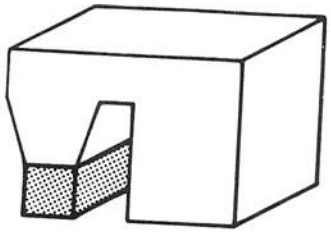
While slicing the CAD model into layers isolated islands may be produced as shown in Figure 1.12. The sectional view in plane 1-1 shows an isolated island which belongs to a projection from the main object. The connection of the projection to the parent body is from the top and while generating the shape by a GMP it will be built later. Thus it becomes essential to de sign a support for the isolated islands to prevent their fall under gravity, as they are created if the process is liquid-based like Stereolithography. When the whole object is formed the extra supports are removed. Due to similar reasons, supports are essential for long cantilevered projections also. Though isolated islands are not formed, the thickness of the projection may be too thin to support the weight of the



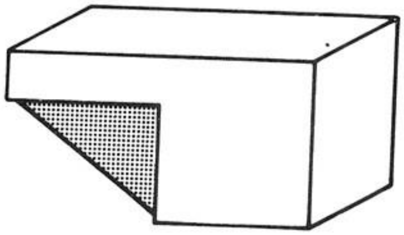
Sectional view in plane 2-2

Figure 1.12: Formation of isolated islands

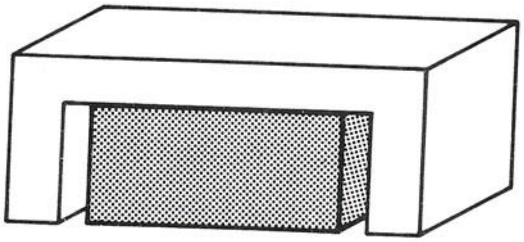
cantilever. Thus, supports in a GMP system are analogous to job holding devices for conventional machining. In addition to preventing the fall of isolated islands, supports are generally provided to hold the main part body also. In future suitable materials for GMP may be developed to eliminate the need of supporting the main object. At present supports are essential to hold the material during operation even if the component is devoid of cantilevers and projections. Figure 1.13 shows different types of supports.



Island



Gussets

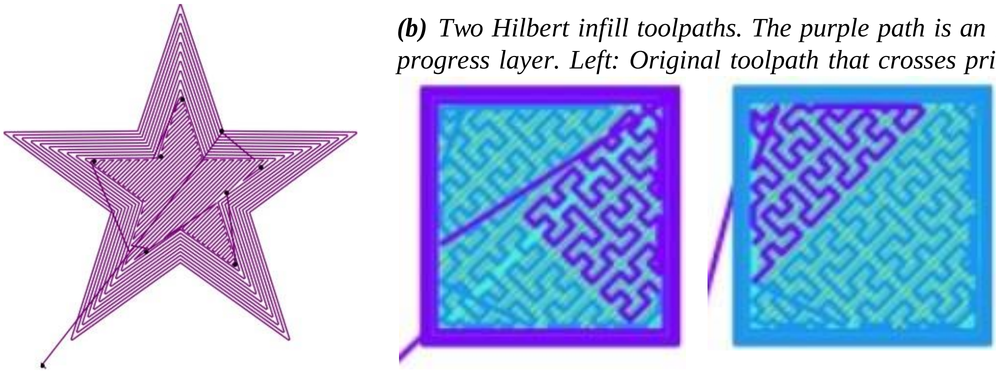


Ceiling

1.13: Various types of Supports

TOOL PATH GENERATION

Advances in additive manufacturing technology make 3D printing a viable fabrication method for a varietyof materials beyond thermoplastics, including elastomeric materials such as silicone, glass, epoxies, and concrete. When 3D printing viscous materials, as opposed to traditional thermoplastic filaments, it is difficult to stop and start the flow of material. Most print path generation software makes use of the printer’soption to enable or disable material flow when constructing the toolpath. These toolpaths tend to produce low-quality prints when used with materials that must be extruded continuously. In this paper, we propose analgorithm for tuning existing toolpaths to reduce extruded material so that the resulting path is better suitedto 3D printing fluids.



(a) *First layer of a rectilinear infill toolpath. Extrusion should stop before travel moves, indicated with black dots.*

(b) *Two Hilbert infill toolpaths. The purple path is an in-progress layer. Left: Original toolpath that crosses print. Right: Reordered toolpath that avoids crossing print.*

Figure 1: *Visualizations of toolpath curves and discontinuities.*

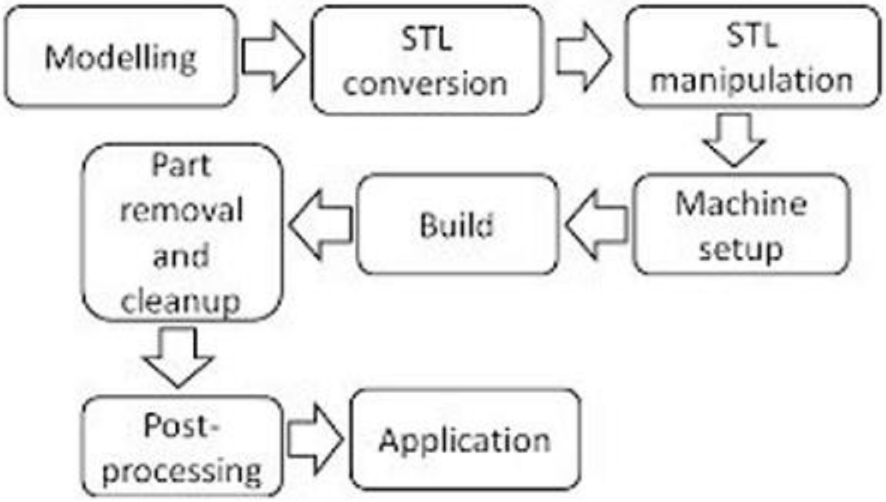
The mechanisms for additive manufacturing of viscous fluids differ significantly from the operation of a thermoplastic printer. Pumps push the fluid through tubing and a nozzle to print, and pressure must accumulate in order to get viscous formulations to flow. The nature of the fluid as well as the pressure builtup in the pump makes it difficult to retract the fluid once it starts to flow. In contrast, thermoplastic materialextrusion relies on a motor to pull the filament down into a heated extruder head so it flows and prints, andfilament extrusion pauses when the motor stops.

Generating a toolpath for a print head to fill the boundary of an arbitrary model shape in one continuousmotion is a nontrivial problem. Conventional slicing algorithms take advantage of the fact that thermoplastic- type systems can precisely control filament flow, so the resulting extruder paths need not be continuous asthe material flow can pause while the print head moves to another region of the print. Because fluid extrusionis continuous, these paths that pass over the middle of the print (as shown in Figure 1a and 1b) or cut acrossconcave outlines drop excess material onto the printed object which detracts considerably from print quality.

In this paper, we formulate an algorithm for rearranging the set of space-filling curves in a toolpath toaccommodate continuous extrusion and demonstrate the performance of this method on an open-source 3D printer. We provide a formal definition of a toolpath for additive manufacturing in terms of its component space-filling curves and layers, which we rearrange by combining a Traveling Salesperson Problem (TSP) solver with a greedy approach to planning. This algorithm is applied to several models of varying complexity and its performance is assessed in terms of the quantity of material extruded.

DESIGN RULES FOR EXTRUSION BASED AM

From CAD description to physical result, AM involves several steps. The process will vary depending on the product. It is likely that smaller, simpler products make use of AM only for visualization purposes, whereas larger, more complex products may incorporate AM at multiple stages and iterations throughout the development process. Also, in the early stages of product development, rough parts may be needed, and AM is generally used due to its rapid fabrication capability. As parts advance through the process, they may need cleaning and post-processing (such as coating and sanding) before they can be used; in this regard, rapid prototyping is useful due to the complexity of forms that can be created without involving tooling.



Step 1: CAD

CAD models that fully describe the external geometry are required for all AM parts. Any professional CAD solid modelling software can be used to create this, but the final product must be a 3D solid or surface model. To create such an image, reverse engineering equipment (for example, laser and optical scanning) can also be used.

Step 2: Conversion to STL

Upon completion of the digital model, the STL (Standard Tessellation Language) file format must be used to create the stereolithography. Nearly every CAD system supports this format, which is how AM machines communicate. The STL file serves as the basis for calculating the slices of the model.

Step 3: Transfer to Machine

In the third step, the STL file is transmitted to the AM machine. As a result of this step, it is possible to adjust the build so that it is positioned and sized correctly. A computer controls the AM machine. The AM machine is controlled by the computer, that computer only generates the required instruction in the form of G-codes and M-codes based on the given process parameters. It generates instructions automatically, if any correction is needed for the betterment of the part to be built it can be corrected.

Step 4: Setup

Before the building starts, the equipment has to be set up. The settings can constitute power, speed, layer thickness, and other several parameters related to material and process constraints, etc.

Step 5: Build

The fifth step is the actual building of the CAD model, melting layer by layer. This process can be semi or fully automated but some online monitoring is often conducted, so that the machine does not run out of material or that some software error occurs.

Step 6: Part Removal

Once the part is manufactured it has to be removed from the process, which is normally done manually. This may require interaction with the machine, which may have safety interlocks to ensure, for example, that the operating temperatures are sufficiently low or that there are no actively moving parts.

Step 7: Post-processing

After the build, the part might need some post-processing before it is completely finished. Of course, depending on the material and AM process used, some parts might need machining, cleaning, polishing, removal of support structures, hot isostatic pressing (HIP), and heat treatments.