FDM BEST PRACTICE

DESIGNING FOR FDM PRODUCTION PARTS

SOFTWARE / **PRODUCT** / FINISHING

Introduction

This best practice concentrates on factors that impact the quality, value, and performance of FDM® production parts. There are a number of design considerations, material choices and performance objectives that should be taken into account when designing for additive manufacturing. Tailoring these decisions to the FDM manufacturing process allows the benefits of FDM to be maximized.

Contents

CONSIDERATIONS IN BUILD ORIENTATION

Build orientation should be considered as a precursor to any detailed design. Failure to consider build orientation early on will result in the need to compromise part quality and requirements at later stages of design.

The current state of additive manufacturing technology results in parts that are highly anisotropic, making build orientation a critical design factor. Build orientation dictates the directional behavior of design requirements, including:

- Mechanical properties
- Aesthetics and surface finish
- Cost and time
- Accuracy

There are occasions when the best-case build orientation for a given design requirement often conflicts with the optimal build orientation of other part requirements. As such, a part should have build orientation selected as a precursor to any detailed design, as well as incorporating other multiple design requirements. Failure to select a build orientation early in the design phase results in a compromise that meets only the minimum design requirement, or worse, may eliminate or reduce certain requirements to accommodate the build process.

The ideal time for choosing build orientation is after the concept design is completed but before detailed design requirements are implemented. Stratasys offers several material options with a range of mechanical properties. The primary consideration is to optimize for a weighted set of requirements including cost, buildability, accuracy, aesthetics, mechanical properties and surface finish. From there, decisions can be made during the detailed design to mitigate the effects of anisotropy on secondary or lower-priority requirements.

If strength is the highest priority and the material will be highly stressed, build orientation can be selected based on mechanical properties. In this case, negative impacts to the other design requirements should be mitigated in the detailed design. Ultimately, an experienced designer will be able to easily visualize an optimal build orientation for a preliminary design that satisfies a weighted set of design requirements, and make design decisions to meet the less-critical design requirements considering the locked orientation. Iteration from the pre-selected build orientation is often necessary as a design evolves.

Mechanical Properties

Generally speaking, mechanical properties are best in the X-Y build plane and weaker across layers. Multiple loading conditions may require an orientation that provides the best compromise for each case.

Aesthetics and Surface Finish

Aesthetics and surface finish are best when features are aligned vertically. Minimizing the angle between the vertical axis and a wall ensures that layer stacking is as concentric as possible.

Cost and Time

Build time is reduced by minimizing Z build height and minimizing the numbers of layers requiring both model and support material.

Accuracy

A feature will have the best accuracy if its OML (outside mold line) is traced out on the X-Y build plane.

SELF-SUPPORTING FEATURES

FDM has the unique ability to produce certain geometric features without the use of support material.

Overview

Additive manufacturing typically uses a special support material to hold overhanging geometry in place as the model is created. While some additive manufacturing processes completely surround a part in support material, FDM has the unique ability to produce certain geometric features without the use of support material. This enables a fully enclosed lattice structure, fill patterns, or hollow parts.

Typically, an overhang does not require support material if its walls or features exist at a 45-degree angle or less from vertical. This is known as a self-supporting angle, which varies slightly depending on the material and toolpath parameters used. An overhang that spans a gap of a half inch or less does not require support material. If an overhang spanning a gap does not meet requirements for a self-supporting angle, support material will be generated and the user must manually delete.

Considerations for eliminating support material

There are a number of reasons a designer should eliminate support material within a design. Support material is one of the highest contributing factors in a part's cost, the predominant issue being part production time:

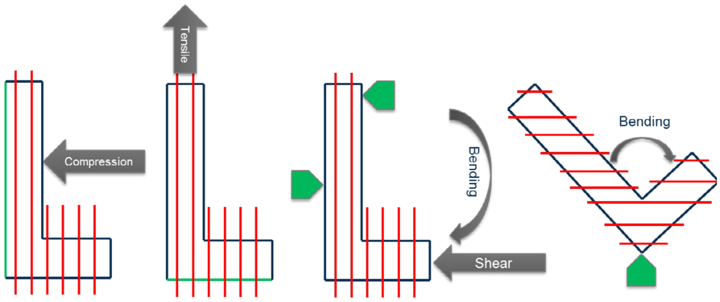

Figure 1: Optimal build orientation for a given loading condition with respect to mechanical properties.

Figure 2: Though identical in Z height, Orientation 2 will minimize build time, reducing support material usage. Also consider, due to the support material interface, the impact on the part's bottom face surface finish.

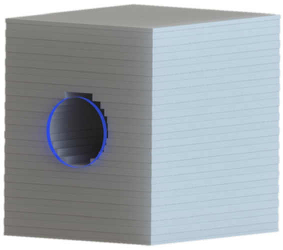

Figure 3: FDM approximation of a small hole in the horizontal direction. The optimal orientation for this hole is vertical.

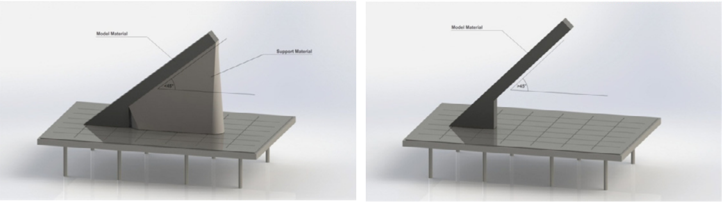

Figure 4: Non self-supporting geometry (left) vs. self-supporting angle (right).

- In a part's build, support material and model material are deposited successively through independent nozzles.
- During deposition, the inactive tip is kept at an idle state so that the material does not degrade or secrete from the tip onto the part. Reactivation of the idle tip requires reheating to extrusion temperature and purging/wiping the tip to restore it to a condition ready for accurate deposition.
- This process takes between 30 seconds to a minute, depending on the material. As a result, each layer that requires both model and support material adds a minimum of 30 seconds to the build time during which the machine is not producing a part.
- By eliminating non self-supporting features, part production time is reduced.

Besides process time and cost, a designer should consider aesthetics and finish in the elimination of support material by:

- Excluding trapped or hard-to-access support structures
- Avoiding the compounding effect of surface roughness, since the support material has the same surface finish as the FDM part
- Minimizing the surface finish at the support interface angle

Generally, self-supporting angles should be maintained whenever it does not conflict with other design requirements such as weight, aesthetics, or functionality.

TOLERANCE

The FDM process involves the deposition of a thermoplastic at elevated temperatures, so a part's tolerance will depend on its behavior while shrinking. When preprocessing part geometry, shrink factors are automatically applied.

Thermal mass and part geometry, among other factors, dictate how appropriate the default shrink factors are for matching actual part behavior. Since standard shrink factors are applied to an infinite possibility of geometries, the final tolerance of a part will vary from part to part. A high level of confidence exists for any given geometry that a part tolerance will meet or exceed. The specifications are listed below:

Fortus 360/400mc™:
± 0.005 in. or ± 0.0015 in. (± 0.127 mm or ± 0.0015 mm), whichever is greater.

Fortus 900mc™/Stratasys F900™:
± 0.0035 in. or ± 0.0015 in. (± 0.089 mm or ± 0.0015 mm), whichever is greater.

HOLES

Holes built using FDM, smaller than 1 in. (25 mm), are typically slightly undersized. When tighter tolerances are required, these holes can be drilled and reamed to ensure accuracy. The minimum hole size for an FDM part depends on the material used since different materials expand and shrink at different rates. All materials are capable of producing holes down to 0.0625 inch (1.6 mm).

To reduce material usage and decrease build time, support structures can be manually removed after support generation from horizontal holes that are less than 0.200 inches (5.0 mm) internal diameter. Re-designing holes as self-supporting diamonds or tear drops, will eliminate the need for support material regardless of the hole size, if the design requirements allow.

MINIMUM FEATURE SIZE

Minimum feature size is a function of slice thickness, toolpath width, and orientation. As a rule of thumb, the minimum feature size is 0.016 inch (0.4 mm). This is available on 0.005 inch (0.127 mm) slice thickness configurations. For features that approach or exceed the minimum suggested thickness, it is advisable to preprocess the part in the correct orientation to validate that the geometry can be filled sufficiently. Note that custom toolpath parameters are often necessary to accommodate small features. The custom toolpath parameters can either be applied globally to the part or locally to the small feature.

TEXT

Minimum suggested text size on the top or bottom build plane of an FDM model is a 16 point, boldface. Minimum suggested text size on vertical walls is 10 point, boldface. If vertical text is embossed inward, the supports for the text can usually be eliminated, because of the ability of FDM to bridge small gaps. In most cases, outward-protruding text can also be produced without support material, if the protrusion is less than .020 inch (0.51 mm).

WALL THICKNESS

FDM is a layered manufacturing process with anisotropic resolution characteristics. Minimum wall thickness must be calculated relative to the build directions, rather than using a thickness value normal to the part surface.

Minimum wall thickness in the Z direction is equal to the layer thickness. Note that when approaching the minimum thickness, the as-built thickness will be rounded down to the nearest multiple of slice thickness. For example, a 0.007 inch (0.178 mm) horizontal wall produced with a 0.005 inch (0.127 mm) slice thickness will likely be rounded down to 0.005 inch (0.127 mm), with the remaining 0.002 inch (0.051 mm) being filtered off.

Thickness in the X-Y planes are much less fixed, as they depend on user-adjustable toolpath widths. Thicknesses in the X-Y build plane are constrained

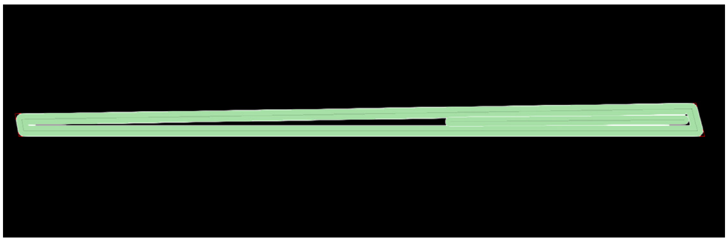

Figure 6: Gaps caused by X-Y thickness variance between 2x and 3x contour thicknesses.

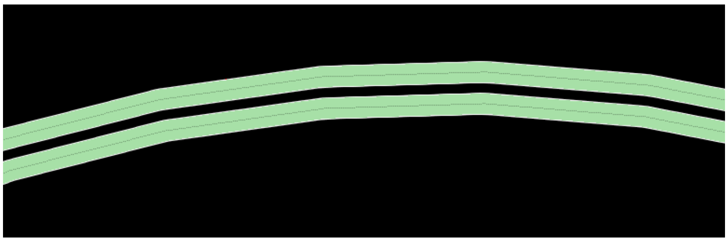

Figure 7: A boundary wall thickness of 0.05 inch (1.27 mm) with default toolpath width (0.02 inch [0.51 mm]). In this case, toolpath width should be manually adjusted to 0.025 inch (0.635 mm) to fill the void.

to a minimum of 2x the desired toolpath width, without using advanced “Single Bead” FDM design techniques (see Lightweight Structures Design Guide). FDM is constrained in producing toolpaths of varying thickness:

- In designing curved parts with constant normal thickness, trigonometry dictates that the resulting X-Y thickness begins to vary, a variance between 2x and 3x toolpath thickness causes voids to appear (see Figure 7).
- With X-Y thicknesses remaining constant when in-between 2x and 3x toolpath thicknesses to avoid the negative effects of partially filled thin members.
- Once a thickness rises above 3x toolpath, thickness and rasters can be included in the toolpath algorithm. Note that thin walls will likely require manual adjustment of toolpath thickness by the user to prevent gaps.

As a rule of thumb for users not able to produce constant X-Y thickness thin-walled geometry, the following “safe” suggested minimum X-Y wall thicknesses are provided. These are 3x the minimum toolpath thickness for each slice thickness.

Table 1: Minimum Toolpath Thicknesses

Slice Thickness inch (mm)	Minimum Vertical Wall inch (mm)
0.005 (0.13)	0.015 (0.56)
0.007 (0.18)	0.036 (0.91)
0.010 (0.25)	0.048 (1.22)
0.013 (0.33)	0.054 (1.37)

SUPPORT GENERATION CONSIDERATIONS

In general, support generation is fully automated and the user does not need to provide special attention to its creation. However, some cases require manipulation of the support structures in order to ensure build reliability. Additionally, a user may wish to modify support material structures as a means to reduce build cost, build time, or part quality. Many of these tactics are detailed throughout this document, while the remainder are provided here.

Model Material in Support

In order to reduce build time, a designer should minimize the number of layers that require both model and support material. If a part contains self-supporting angles throughout the majority of its height, however, a non-self-supporting feature at the top of the part will require support material from the base of the part up to the feature.

- In order to capitalize on the self-supporting aspect of the design, the support structure should be converted to model material where possible. Any layer containing support material that does not have a support interface for the part can benefit from changing the material type to model material. This can be done by selecting the boundary curves for a support structure and assigning them to a preset for the support material type being used.

- The material type should then be changed to model. A change from model to support material increases the likelihood of delamination during the build. To mitigate this, any given support column should be made of model material from its base to a few layers before it interfaces with the part.

Surround Support

Surround support offers greater part stability, however, at the expense of build time and reduced part surface quality. In general, it is not recommended to use surround support unless a part is otherwise unreliable in its build.

- The geometry of downward-facing wedges or a knife-edge is an example of a part needing surround support and without support the part is unreliable for two reasons:
    - The model material deposited in the first layer of the feature will not provide adequate surface area for the adherence of the subsequent layers.
    - The increasing amount of mass being deposited, along with the force implied by deposition, will likely dislodge the part from the small support interface area.
- It is not advised to mitigate failure in a small area surround support for an entire part. In order to localize the generation of surround support, a Z height can be specified. The remaining height of the part will contain standard support.

- Standard support should be generated initially for a part. Subsequently, in the ***Support Parameters*** dialog box, the ***Support Style*** can be changed to ***Surround*** and the starting height can be enabled and set to the desired value. Regenerating support material will re-create the support material up to the specified height while leaving the remainder of the previously generated standard support structures in place.

LEVERAGING DESIGN FREEDOM

FDM offers a designer a path to design freedom, ultimately producing parts with better performance levels and functionality compared to traditional manufacturing processes

Beyond the obvious benefits of using FDM to reduce costs and accelerate the design process and part delivery, the real opportunity with FDM rests in the mind of the designer. It removes the restraints of conventional manufacturing and has produced parts that have:

- Bent light

- Created extremely RF-transparent (radio frequency) radomes.

- Produced large sections for a highly sophisticated unmanned aircraft from single components

Thin Walls vs. Sparse Fill

With additive manufacturing's design freedom, designers can produce semi-dense parts, enclosed lattice structures, or fill patterns.

- In many cases, use of FDM with sparse fill will create a structure resulting in higher stiffness for a given material mass. For example, consider a wall designed with stiffening ribs for injection molding. It may be more advantageous when designing a similar component with FDM to substantially increase all thicknesses using a sparse fill pattern, instead of using stiffeners, maintaining a similar component weight.
- A hollow cube is an example where utilization of a fill pattern creates an advantage. If built with faces parallel to the X-Y build plane, reducing build time and potentially eliminating trapped support material inside of the cube, the internal volume could utilize a fill structure without a large weight penalty.

Light Weight

Producing lightweight structures with FDM parts requires multiple design decisions not covered in this document. See the FDM Lightweight Structures Design Guide for direction on designing parts with a high stiffness-to-weight or strength-to-weight ratio.

Topology Optimization

Topology optimization is a mathematical tool that uses a physics-based algorithm to design distribution of material within a user-defined boundary, based on a set of requirements. The algorithm can optimize for aerodynamic efficiency, electrical performance or heat transfer, among others.

Structural topology optimization has the best defined workflow, with several simulation companies offering a streamlined software that automates the modeling process. Typically, the results of a topology optimization will yield a geometry that is highly complex and organic in shape.

Traditional manufacturing processes are constrained in these cases, requiring interpretation and adoption of results to suit the limits of the manufacturing process. The design freedom allowed by additive manufacturing provides a path to directly utilize the results of the optimization. In some cases, the results can be directly sent to a printer after some cleanup of the output mesh.

Consolidation of Parts

FDM provides a method to consolidate an assembly of multiple components, to create a single component. For example, because it does not impact the cost greatly, an aircraft environmental control system duct can be printed with integrated attachment brackets. However, it is advised to combine components of an assembly only when doing so does not violate any engineering constraints, such as material performance, part cost or build time. With any part consolidation, the build orientation must meet the requirements of the individual parts within the combined part.

Peripheral Component Packaging and Mounting

Similar to the consolidation of parts, an FDM part should be designed to integrate mounting features and packaging of peripheral components. This integration allows for high levels of modularity, as well as the ability to reduce the footprint of assemblies.

DESIGNING FOR FDM PRODUCTION PARTS

Designing for FDM Production Parts – Key Takeaways:

- Build orientation dictates the directional behavior of design requirements, including:
    - Surface finish and aesthetics, build cost and buildability, tolerances and accuracy and mechanical, electrical, and thermal properties
    - A part should have build orientation selected as a precursor to any detailed design.
    - The ideal time for choosing build orientation is after the concept design is completed but before detailed design requirements are implemented.
- While some additive manufacturing processes completely surround a part in support material, FDM has the unique ability to produce certain geometric features without the use of support material. Support material is one of the highest contributing factors to a part's cost.
- FDM deposits thermoplastic at elevated temperatures, so a part's tolerance will depend on its behavior while shrinking.
- FDM is a layered manufacturing process with anisotropic resolution characteristics. Minimum wall thickness must be calculated relative to the build orientation, rather than using a thickness value normal to the part surface.
- Surround support offers greater part stability, but, at the expense of build time and reduced part surface quality.
- With additive manufacturing's design freedom, designers can produce semi-dense parts, enclosed lattice structures or fill patterns. In many cases, use of FDM with sparse fill will create a structure resulting in higher stiffness for a given material mass.
- FDM provides a method to consolidate an assembly of multiple components, creating a single component. An FDM part can be designed to integrate mounting features and packaging of peripheral components, allowing for high levels of modularity.

CONTACT

For questions about the information contained in this document, contact Stratasys at www.stratasys.com/contact-us/contact-stratasys.

stratasys

STRATASYS.COM

HEADQUARTERS

7665 Commerce Way, Eden Prairie, MN 55344
+1 800 801 6491 (US Toll Free)
+1 952 937 3000 (Intl)
+1 952 937 0070 (Fax)

1 Holtzman St., Science Park, PO Box 2496
Rehovot 76124, Israel
+972 74 745 4000
+972 74 745 5000 (Fax)

ISO 9001:2008 Certified