Fused Deposition Modeling (FDM)

Best Practices & Things to Keep in Mind

Contents

Fused Deposition Modeling (FDM) is one of the most common 3D printing methods today and has long been a "go-to" method for many creators. In FDM, a print nozzle with a heating element acts as a micro-extruder to lay down melted plastic filament in layers to create a complete part. But, despite the familiarity of this method of printing, there are still considerations that must be addressed before printing. Issues such as use of support material, layer lines and general aesthetics, load planning, printing time and others must be considered before diving into a new project. And all these issues point back to a solid design at the outset.

Why design matters

All 3D printed objects start with a print file containing the specifics of the part. This file is produced using 3D CAD software that slices the scan or representation of the object into layers. Each layer – or slice – is digitally converted to code that will tell the printer how to direct the nozzle in its path.

Determining the right design is critical because it impacts the entire project both physically and practically. Areas impacted by design considerations include:

Cost

As the first step in the process, design will ripple throughout the print and post-processing stage and can increase or decrease the overall project cost. A design that is not optimized may use more material than necessary. Or it may require the use of consumable support materials that must be printed with the part. It may also drive increased labor at the post processing stage should the part require additional post-processing handling such as removal of excessive support structures, sanding, drilling, or inserts or bushings to shore up holes.

Time

Part design can also impact overall time for printing. A simple design feature such as tabs or overhangs can add hours to the completion of a print. And an orientation that is not thought through relative to part requirements can increase print time and can double or triple the time compared to a different orientation for the same part.

Mechanical Characteristics

One of the biggest considerations for design is to assess the required mechanical characteristics and design the part based on those requirements. This often means taking a step back and considering what is needed mechanically. It is natural to view a CAD file with the view of the part as it will be used in its final iteration, or to view a part with an eye to what seems to be an intuitive orientation. However, as FDM parts are anisotropic, issues such as load, sheer, stress and other factors need to be examined and built into the design.

Anisotropic part strength occurs when parts have physical properties that vary in different orientations. It is an issue with additively manufactured parts due to the layer-by-layer material deposition to create the part. The bond strength between layers is the determining factor for z-strength which is always the weakest. The highest values for tensile strength will be in the path of the tool head (x-y plane). Build orientation, part geometry and pre-processing techniques can all be used to account for and mitigate these process-specific strength characteristics.

Surface Features

Design also matters when looking at the final surface features the part will have. Resolution may impact visual aesthetics, especially when trying to design a part with words, lettering or logos. Other features such as holes, overhangs and undercuts can also be problematic and appear different in the final print once resolution, build layer thickness, part orientation, material being printed, and the impact of support removal are all taken into consideration.

Trade-Offs

Because part design is so critical to final production, it is important to understand that there could be trade-offs in design requirements in order to result in a printed part that meets as many requirements as possible. If the part requires higher expectations for aesthetics, then part design may need to trade strength for features.

Conversely, if the load, stress or other mechanical requirements are critical, the design may need to trade features for strength. It may even require the part, or elements of the part, be printed separately and joined through alternative means to achieve the desired strength.

Best FDM Practices for Design

The design of a 3D printed part has two stages, the concept stage and the detail stage. It is during the concept stage that part orientation should be determined. This will affect almost every aspect of the finished part. Once the orientation is determined, the detail design phase is where features and other factors may be adjusted to allow the print to work in line with the orientation.

Here are some of the considerations and best practices for design for FDM parts in the design detail stage:

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.

Tabs

Tabs are a common feature in traditionally manufactured parts that can be problematic for FDM parts. They may also be impacted by the part size with larger parts having increased susceptibility to breaking. The simplest way to address this issue is to increase the size of the tab or build side structure or infill to strengthen the area of concern. Depending on orientation, it may also be possible to orient tabs along the XY axis to take advantage of this plane's higher inherent strength during printing. And in cases where orientation or part structure doesn't allow, it may be possible to print tabs separately and to design both tabs and master part with an interlocking "jigsaw" insert at the post-processing stage.

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.

Gaps

If the part is to have gaps, it is recommended that the width be greater than 5mm. This will allow for the removal of support material when required. If possible, orientation of the gaps along the XY axis will help ensure proper resolution and will eliminate the need for support material.

Wall Thickness

FDM is a layered manufacturing process with anisotropic resolu tion 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 - when approaching the minimum thickness, the as-built thickness will be rounded down to the nearest multiple of slice thickness.

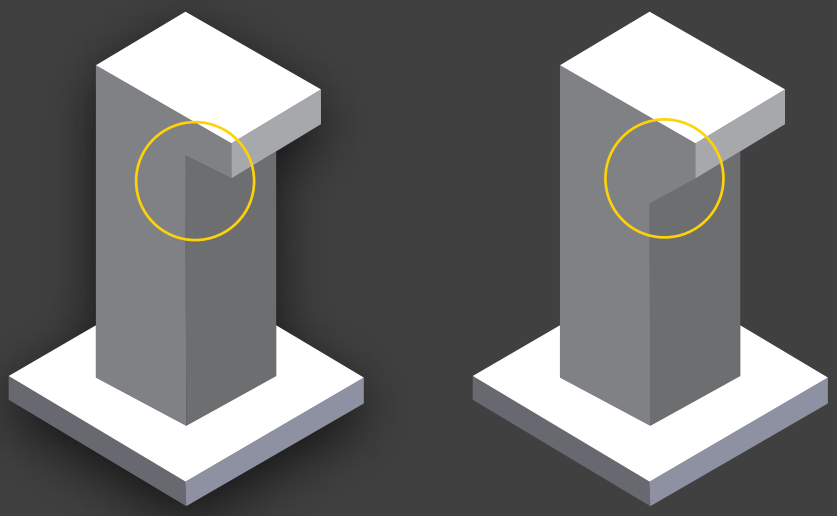
It is also important to remember that closed profile walls will have better resolution than open profiles as filament can be used to infill the wall. Walls can be produced using dense fill, lattice structures or use a specified fill pattern, depending on strength and other mechanical requirements.

Support Structures

Support structures are necessary to hold the part down on the build platform and to provide support for the part to be printed and the material to harden as it cools. If a part's orientation requires it be built in a certain direction to take advantage of increased mechanical strength, additional support material may be required as determined by the slicing software. Support structures offer greater part stability but add the expense of build time and reduced part surface quality. And removal of supports often requires sanding or other surface treatment in post-processing.

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



Non-Self-Supporting   Self-Supporting

Cost Difference

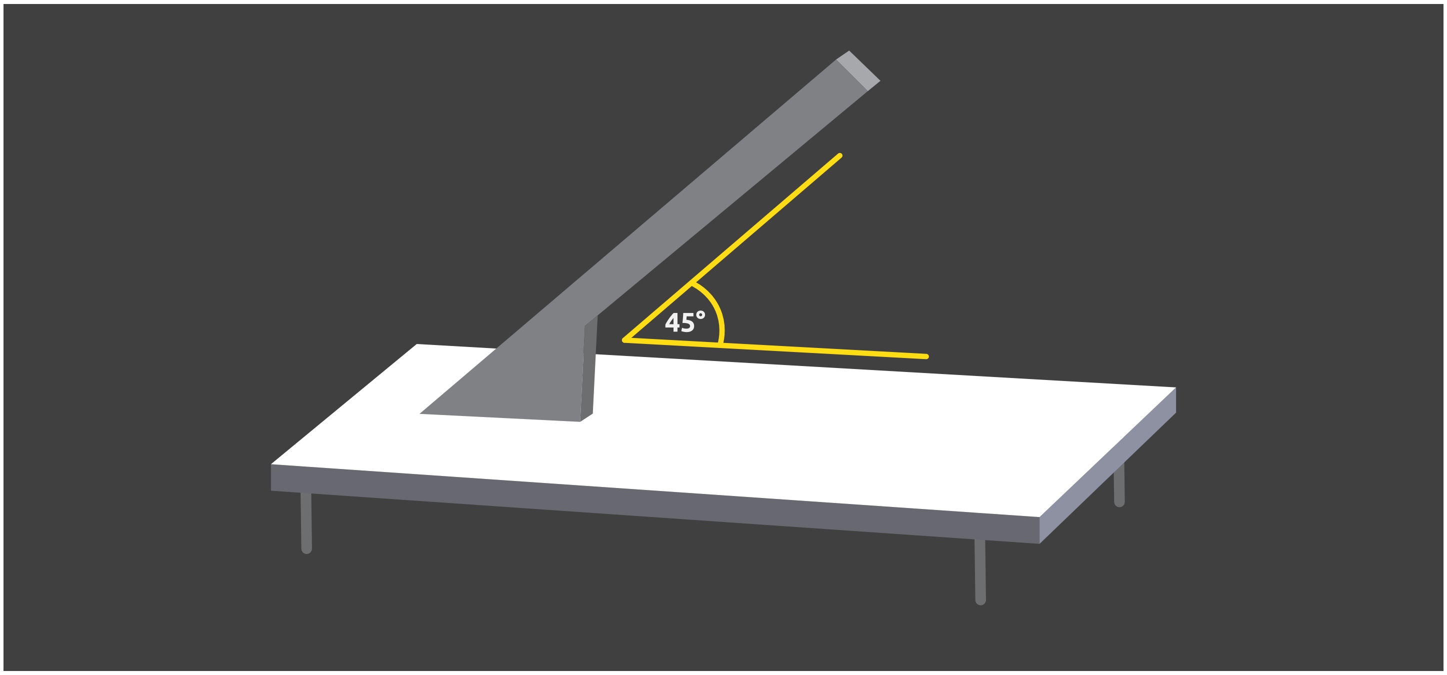
The cost difference between a non-self supporting part (left) and self-supporting part (right) can be very significant. In this case, the non-self supporting geometry equates to a cost difference of 2.5x less when factoring in the standard cost for machine utilizations.

Overhangs

While some additive manufacturing processes 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

Fillets

Design for FDM, and for 3D parts in general, often requires that the part have features that would not be present in a traditionally manufactured part. This is because features specific to AM help strengthen inherently weak areas or mitigate structural inconsistencies inherent to the FDM process. One example is the use of fillets. While FDM can generally achieve parts with overhangs of less than 45°, parts greater than 45° require support. Fillets allow for increased structural rigidity while allowing the part to build up to a required height without support. The look of the part will be more "rounded" compared to traditional manufacturing, but the technique can help address issues where support structure is not practical, but the angle of build is greater than 45°



Infill

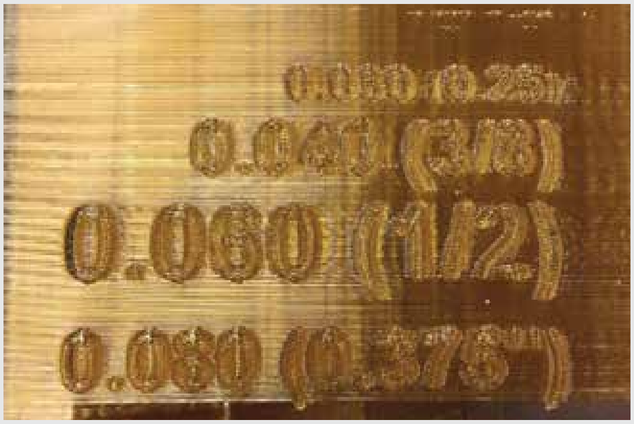
Infill is a key consideration in FDM design. Infill can be consistent throughout or it can vary in different areas of the part, depending on need. In addition to strength consideration, infill can impact the cost and print time of the final part as well. For parts requiring maximized strength, a part can be filled with no voids or interstitial space. For parts with some strength requirements, a lattice or "cross-hatch" pattern can be used. This adds strength but reduces cost and print time. Finally, a part can be designed with sparse infill, resulting in a lower weight part that has reduced cost and print time albeit at a trade-off for strength.

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).



Surface Finish

Surface finish will be determined by several factors including printer used, slice height, filament diameter, build orientation, part shape and supports required. Lower end printers may have clearly visible layer lines that could require extensive post-processing in the form of sanding. To ensure that the part being designed meets the required surface finish requirements, the designer should understand the printer being used, and the diameter of the filament used, and design accordingly with the appropriate CAD software.

ABS - M30

Next to PLA, ABS has long been a standard for FDM printers. **ABS-M30 is 25 to 70% stronger than standard ABS** depending on part design. ABS-M30 is used for concept modeling as well as prototyping and can be used for manufacturing tools such as jigs and fixtures. Parts made with ABS-M30 are generally smoother with excellent surface appeal exhibit significantly greater strength due to improved layer bonding. It has excellent tensile, impact, and flexural strength and support removal is hands free when using soluble support.

ASA

ASA (Acrylonitrile Styrene Acrylate) is an engineering plastic and will retain its appearance and impact resistance after long term exposure in adverse conditions. ASA can be used to build UV-stable parts and has superior aesthetics compared to other FDM materials. ASA has excellent mechanical properties and strength which makes it highly desirable for outdoor production parts and prototypes. It is used in roof coverings, electrical installations and enclosures, auto parts, toys and other outdoor uses. ASA also has a good surface finish and good chemical resistance

Nylon 12 – Chopped Carbon-Filled

Nylon 12CF combines nylon and **up to 35% chopped carbon filament**. It is considered one of the strongest FDM thermoplastics and has excellent structural characteristics. The presence and random orientation of the chopped carbon filament allows more cross bonding between layers and as a result, Nylon 12CF has a very high strength to weight ration and excellent tensile strength. It is used within aerospace, automotive, industrial and recreational industries.

Polycarbonate

Polycarbonate was the original base material for bulletproof glass and those mechanical characteristics make it an excellent FDM material as well. 3D printed Polycarbonate parts are extremely tough and heat resistant. It has a tensile strength of 9,800 psi, exhibits high impact resistant and is heat resistant and flame retardant. Polycarbonate is used within automotive, construction and transportation for parts such as data storage media, sunglasses, phone and computer cases and food containers.

PC/ABS

Another engineering grade thermoplastic, PC/ABS is used in automotive, electronics, medical and telecommunication. The blend of Polycarbonate and ABS can yield parts that are 5-60% stronger than traditional ABS alone. PC/ABS is highly heat resistant and the blend results in filament that captures the strength of Polycarbonate and the flexibility of ABS. Parts made with PC/ABS also exhibit excellent feature definition and surface finish.

ULTEM 9085

ULTEM™ 9085 is similar in characteristics to ULTEM™ 1010. It is a high- performance thermoplastic that is flame retardant and has a high heat resistance. ULTEM™ 9085 has met the meticulous testing criteria required by aerospace industries and is often used as a lightweight alternative to metal. In addition to being flame retardant, it is FST-compliant, meaning it possesses excellent flame, smoke and toxicity characteristics. It is often used to print 3D parts such as electrical enclosures, ductwork and interior components and can also be used to print functional prototypes and customized tools.

Figure: ULTEM 9085 Aero ULA Vent

ULTEM 1010

ULTEM™ 1010 (Polyetherimide) is an extremely strong FDM thermoplastic. It has a high glass transition temperature of 217°C and can perform in continuous operation of 170°C. This excellent heat resistance makes it usable in autoclave operations for sterilization processes. ULTEM™ 1010 is naturally flame retardant as well. It has excellent chemical resistance and a high strength to weight ration making it useful within automotive and aerospace applications. ULTEM™ 1010 is both bio-compatible and food safe. It is often used for molds, prototypes and production parts.

Figure: ULTEM 1010 Air Intake

Printers for FDM Materials

Printing for a wide range of FDM applications requires stringent and controlled printer specifications. This is especially true when using high performance and engineering grade FDM materials. These applications often require a larger build envelop to meet the requirements of part sizes for various industries. At GoProto, we utilize Stratasys' F900 and 450mc FDM printers.

Stratasys F900

The Stratasys F900™ is a robust, industrial-sized FDM machine that combines size and controls into a single unit. The Stratasys F900™ is designed to use a wide range of engineering grade thermoplastics as well as more common materials such as ABS. This is supported by dual build zones that allow dual parts builds or a combined build for larger parts. It has a large build envelop of 36" x 24" x 36" and can produce parts to within an accuracy of ±0.09mm. The F900 can operate at a throughput of 2.1 times the speed of standard mode.

The F900 is integrated with GrabCAD software that allows the machine to print directly from CAD designs. This is combined with onboard monitoring of the workflow to manage material usage, history and other real-time performance factors.

Stratasys 450mc

Although smaller than the F900, the Stratasys 450mc™ can handle the same engineering grade materials with a superior level of control. The 450mc has a build envelope of 16" x 14" x 16" and can produce parts to within an accuracy of ±0.127mm. The 450mc can also operate at a throughput of 2 times the speed of standard mode.

Let’s Get This Done

GoProto is a full-service manufacturing partner, with a focus on customer service across the country. We are organized unlike any other rapid manufacturing solutions provider, with dedicated work groups in territories across the US. Each territory is staffed with an expert Business Development Manager living and working in that region, matched up with dedicated project manager experts to offer consistent and amazing service from initial consultation through production. People matter, relationships matter, and consistent expertise is critical to help you achieve your goals of speed, cost-effectiveness and the best possible process to meet your project needs.

We know you like to win. We do too. We’ll work together to make it happen

ON-SITE TRAINING

GoProto will gladly come on site for training and presentation via lunch and learns.

DESIGN HELP SERVICES

Looking for DFMA (Design for Manufacture and Assembly) guidance We can help to ensure this process is smooth.

WE’RE FAST AND GOOD

We understand that time is priceless and will never sacrifice our commitment to superior quality in the production of your parts.

Additive Manufacturing

CNC Machining

Sheet Metal

Cast Urethane

Injection Molding

Finishing

Go to **www.GoProto.com/GoQuote** to quote and order today!

GoProto

www.GoProto.com

1-888-GoProto (467-7686) | quotes@GoProto.com