

Xometry

Design Guide: Fused Deposition Modeling (FDM)

VERSION 2.0



Table of Contents

Overview.....	3
Printing Process.....	4
Tolerances.....	5
General Tolerances.....	5
Wall Thickness.....	5
Holes.....	6
Text & Small Details.....	7
Gaps.....	7
Tabs.....	8
Features.....	9
Fillets.....	9
Ribs.....	9
Infills.....	10
Orientation.....	10
Surface Finish.....	11
Finishes & Post-Processing.....	12
Resources at Xometry.....	13

Overview

Fused Deposition Modeling (FDM) is an additive manufacturing process in which a machine precisely extrudes melted plastic filament to create a part. Parts are very rigid, especially compared to Selective Laser Sintering (SLS), which makes them a great fit for projects with a rigidity requirement.

Given that the filament has a thickness of 0.010" (0.254mm), some fine features may not resolve properly. Knowing this, it is important to follow guidelines set forth in this guide to ensure all features are properly resolved.

FDM also requires support material (soluble or break-away) for certain features, so it is important to keep this in consideration when designing parts to allow support structures to be removed during post processing.

NOTE:

Designing for FDM is an iterative process. Features will not always print as-desired the first time. Realize that producing parts in FDM is a combination of iterating through prototypes and designing for the process, so be sure to have patience and work with the support team at Xometry to design parts correctly.



Printing Process

Producing parts in FDM is broken up into 3 steps, which includes pre-processing, production, and post-processing.

PRE-PROCESSING:

Printer software slices 3D CAD file into layers. For each slice, the software then converts the data into machine code that determines tool paths for the machine to follow.

PRODUCTION:

An extruder head extrudes liquefied plastic filament along the tool path layer by layer until the part is completed bottom-up.

POST-PROCESSING:

Support material is removed by either dissolving it in water, or breaking it off. More custom finishes such as tapping, inserts, etc. are then applied.



Printed parts on build sheet

Tolerances

General Tolerances

- $\pm 0.004"$ or $\pm 0.002"$ per inch, whichever is greater
- Build area up to 24 x 36 x 36"
- Prints in 0.010" layers (0.254mm), or 0.013" (0.331mm) layers for parts over 16" (406.4mm) max dimension
- Minimum feature size at least 0.035" (0.889mm), 0.045" (1.143mm) is safest



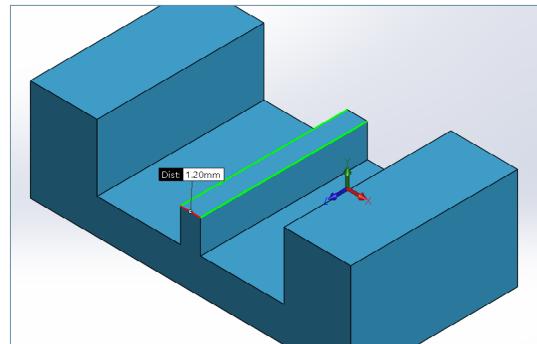
Wall Thickness

Wall thickness is limited by the filament size, and features thinner than twice the filament's thickness usually do not print successfully. Features with only one layer of filament are weak and inaccurate. For supporting structures, walls need to be at least 0.047" - 0.06" (1.2 - 1.5 mm) thick for best results by allowing filament to fill between layers.

Walls without closed profiles such as lines are also less resolved compared to those such as circular and closed loops. As a result, thinner, more circular walls are closer to design intent than those with straight walls.

NOTE:

Walls designed below tolerance may not register, or might print with unexpected errors, such as the "bulging" effect on the side wall below where the machine attempted to compensate for the thin wall.



Minimum wall thickness

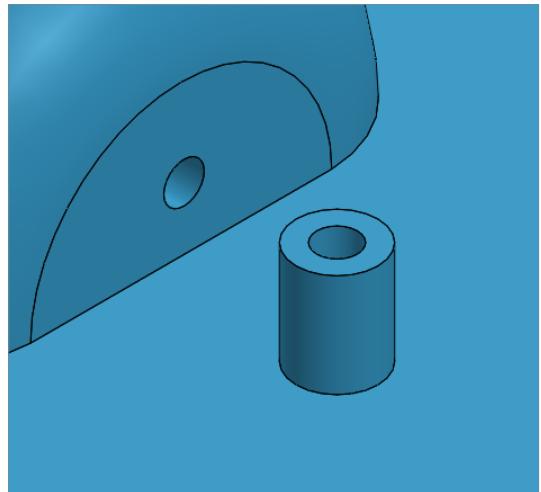
Holes

Xometry recommends that holes for FDM be designed with a diameter greater than 0.04" (1mm) to retain a circular shape. Orientation of holes is very important and resolution tends to be best when printed parallel to the xy-axis. This is because holes are outlined with concentric circles of filament versus staggered stops between layers.

To increase the accuracy of through holes, Xometry may drill out holes during post-processing.

NOTE:

Post-process drilling is only possible for solid infill parts.



Hole orientations



Warped hole

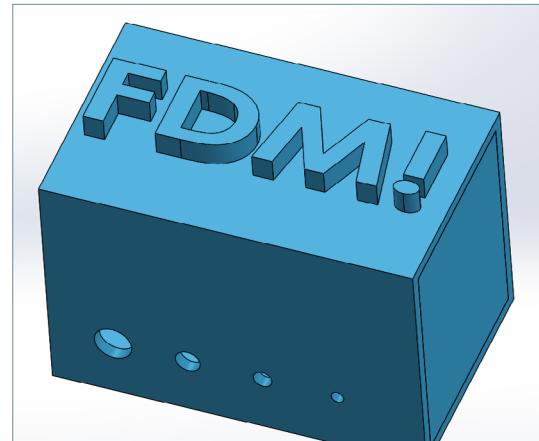
Text & Small Details

Text and small details are very difficult to print in FDM because many small features do not register in the printing process when they are below certain tolerances such as the filament's thickness.

Recommended thickness that protruding text must maintain is at least 0.04" (1mm) to register properly. Text must also have a height of at least 0.04" (1mm), but 0.047" - 0.06" (1.2 - 1.5 mm) is recommended to be legible and avoid unexpected errors, which will allow multiple layers of filament to stack on top of each other and reduce the risk of breakage. Cut-out text must also hold these tolerances to resolve properly.

Be sure to use regular or larger text fonts, but stay away from more condensed and smaller fonts.

All text must also be oriented parallel to the xy-plane for best results, because it will allow multiple layers to stack cleanly and avoid "squished" characters and details.

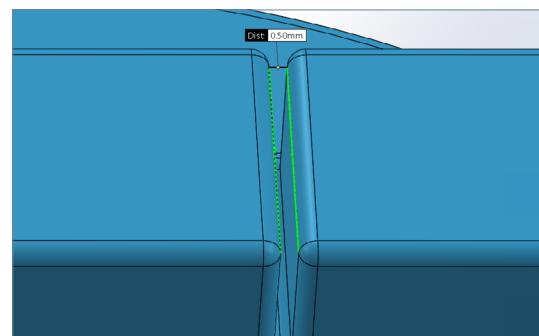


Text resolution

Gaps

A good tolerance to follow for thin gaps is maintaining a width greater than 0.2" (5mm) to ensure that all support material can be removed.

Orienting gaps so that the opening is parallel to the xy-plane is a good way to make sure that parts have resolution. This also reduces the need for support material on that feature.

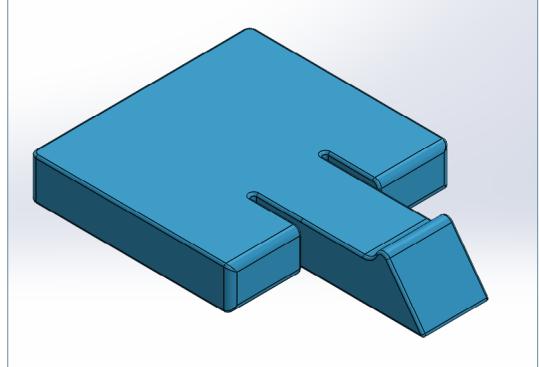


Gap distance

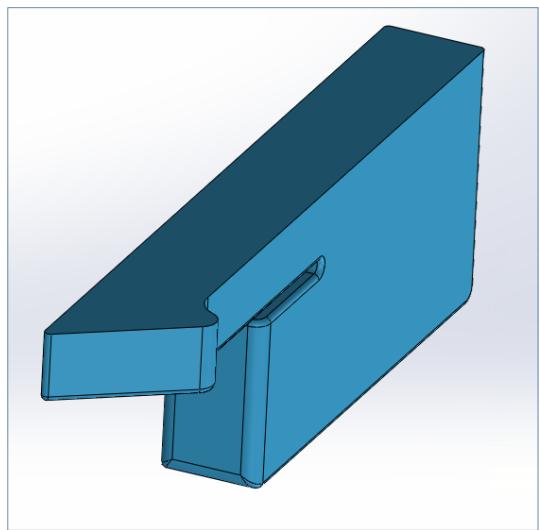
Tabs

Tabs are very susceptible to breaking in FDM, especially for larger parts. The first thing to do would be to increase the size of the tab as large as possible (size depends on the application and size of the overall part).

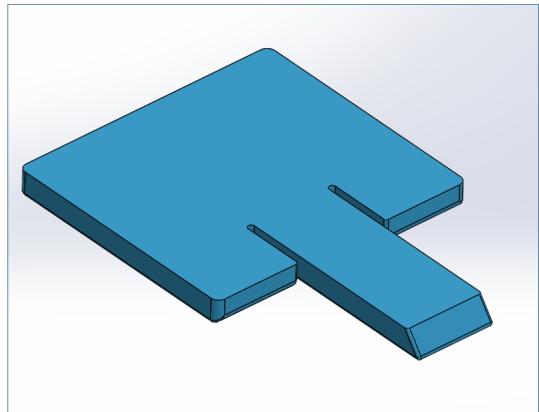
Orienting tabs parallel to the xy-plane will also increase the strength of the tab and reduce the risk of breakage. Integrating a replaceable tab design is another great way to use tabs in FDM, so that tabs may be printed separately in the best orientation then inserted post-processing.



Tab orientation



Stronger orientation



Weaker orientation

Features

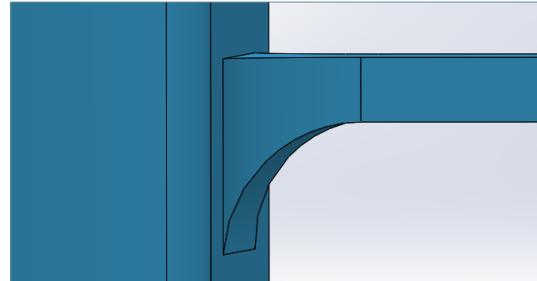
Fillets

Fillets are a helpful design feature to implement in every additive manufacturing process, especially FDM, in order to reinforce fragile features. Fillets will greatly increase structural rigidity, and help surfaces build naturally and avoid the need of support material.

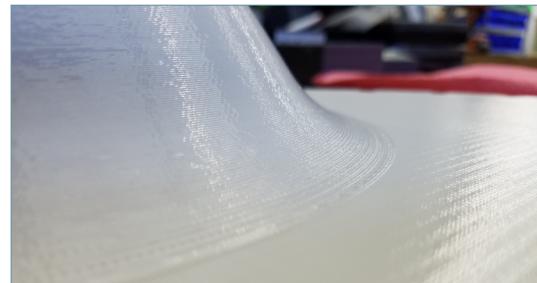
Support material is required for overhanging surfaces greater than 45 degrees from vertical, so adding a fillet will help the surface gradually grow, and continually support itself.

NOTE:

It is important to use the largest fillet possible to increase the strength by having a larger amount of surface area between features.



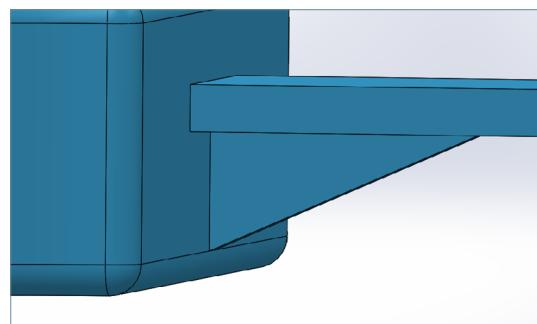
A fillet on a CAD model



Example of a printed fillet

Ribs

Another great alternative to fillets for increasing the strength of support structures is by adding ribs. This is because ribs distribute the force applied to a structure to a greater surface area. It is important to follow tolerances outlined in the wall thickness section when creating ribs to ensure that they aren't too thin and either break or don't register during printing. For this reason, a safe minimum rib thickness to follow is 0.06" (1.5 mm) to allow multiple layers of filament inside of each rib.



A rib on a CAD model

Infills

A great way to save money for FDM solid parts is changing the infill type. Xometry offers FDM parts in 3 different infill options:

- **Ultralight:** Consisting of a single cross-hatch pattern throughout the parts, this infill is the most cost-effective for parts that don't need to be very strong.
- **Light:** This is the next best option for parts that need to retain some strength - a double cross-hatch pattern throughout the part to increases some strength but keeps the price low.
- **Solid:** The standard, strongest option - material is completely filled throughout the part.



Ultralight infill

Orientation

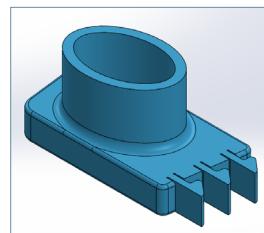
Orienting parts in FDM has an enormous impact on its overall strength and appearance, especially for fine and concentric features. Concentric features resolve best when layers print parallel to the xy-axis. Many fine features (e.g. tabs) are stronger when printed parallel to the xy-axis as well.

Designing a part so that fragile and concentric features grow in the same direction will help determine the best orientation for a part to print.

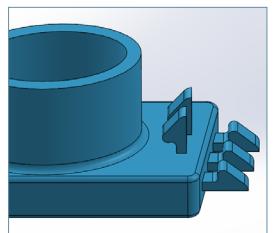
NOTE:

Long surfaces will also save a lot of money and time if cross sections orient parallel to the xy-plane.

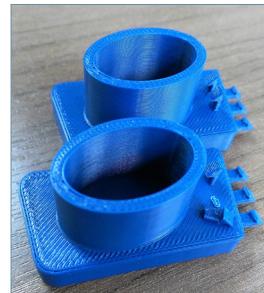
Printing the part shown from the bottom up, vertical tabs are extremely weak and may break off between layers. However, concentric features are more resolved and have no layer stepping. When printing the same part on its side, tabs are much stronger, but concentric features have significant stepping.



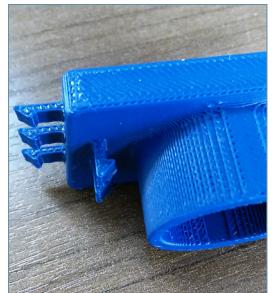
Successful orientation features



Unsuccessful orientation features



Weak tabs, good concentric resolution

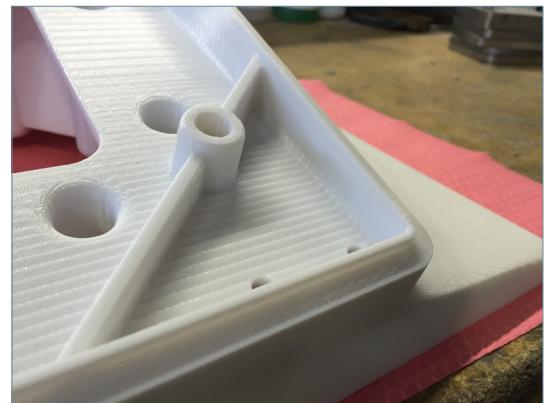
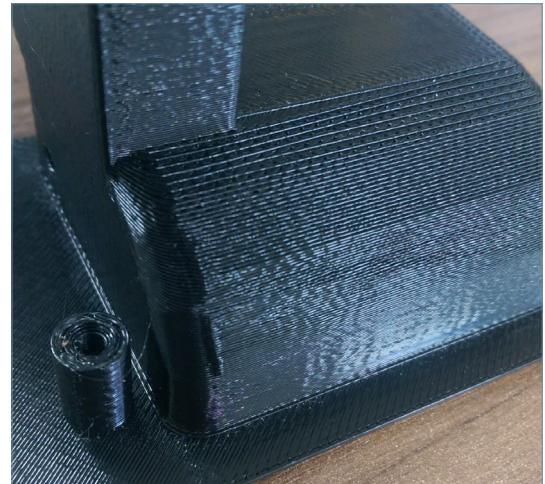


Strong tabs, layer stepping

Surface Finish

FDM tends to create a more rough and noticeably layered surface finish when compared to other manufacturing processes due to its layer by layer process of extruding plastic filament. This layer "stepping" is especially noticeable for shallow-angled and curved surfaces.

Designing parts so that curves and angled-surfaces can grow parallel to the build surface, or increasing angles or curvatures may reduce this stepping.



Examples of layer stepping in printed parts

Finishes & Post Processing

Xometry offers FDM parts in:

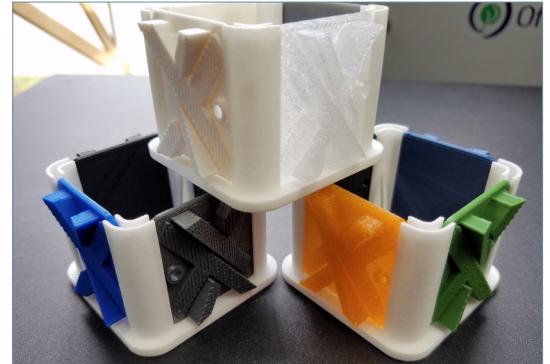
- **ABS**: strong, affordable, production-grade thermoplastic, many colors
- **Ultem**: the strongest, most heat resistant plastic available in FDM
- **ASA**: UV-resistant, affordable, durable plastic, many colors
- **Polycarbonate (PC)**: widely used industrial thermoplastic, offers high tensile and flexural strength
- **Nylon**: great impact strength, widely used in traditional manufacturing
- **PPSF**: sterilizable, strong, high performance plastic

There are currently 16 different color available in FDM depending on material to allow for tremendous customization for any part.

In ABS, PC, Ultem, and PPSF materials you also have the option to sterilize parts to further increase the part's applications.

FDM parts are known for their strength and rigidity, so many mechanical features can be applied to parts without risking structural integrity. Possible features include:

- Tapping
- Inserts
- Pins



Color options



Material options



A tapped hole in a printed part

Resources at Xometry

Online Instant Quoting

Web: Upload your CAD file at get.xometry.com/quote

CAD: Download the free Xometry Add-In for SOLIDWORKS: xometry.com/solidworks

Accepted File Types: .stl, .step, .stp, .x_t, .x_b, .sldpart, .ipt, .prt, .sat, .catpart (max file size: 300MB)

Capabilities: CNC Machining, Sheet Metal Fabrication, 3D Printing, Urethane Casting, Injection Molding

Live Engineering Support

Hours: M-F 8:00 AM - 9:00 PM EST

Email: support@xometry.com

Phone: (240) 252-1138

Online: xometry.com/support offers live chat, FAQs, and other helpful articles.

ITAR Registered

All uploads are secure and confidential. Contact us for assistance with large orders and existing Xometry quotes.

Need to use a PO? We offer same day credit application processing. Contact us for details.