

DESIGN FOR ADDITIVE MANUFACTURABILITY:

CAD Output for FDM

Your Computer Aided Design (CAD) file tells the story of your potential part. It holds the data and information about material, tolerances, geometry and features. CAD applications use a universal set of surface and solid definitions that make up a part. Additive manufacturing systems (3D printers) require a different set of data, basically made up of triangles forming the surface of a part. In order for an additive manufacturing system to build a part, the CAD file needs to be exported into an STL file. Often information can get lost in this translation from CAD to STL. It is important to prepare your files for the export to ensure quality expectations and design intentions are maintained from CAD to final part production.

There are a number of ways to optimize and prepare your CAD data to guarantee clean files ready for processing and production. There are adjustments you can make to complex geometry data, such as wall thicknesses, or even small changes to file size and features that will help create a pristine STL file and accelerate processing. This article will explain some important considerations specifically for the Fused Deposition Modeling (FDM) process. We recommend reading *Design for Additive Manufacturability: FDM Basics* first to learn the differences between designing your part for 3D printing vs. traditional manufacturing processes.



TRIANGULATION

The STL file format is used to exchange information on the part geometry between CAD and a 3D printer. The STL format approximates the surfaces of a solid or surface model or point cloud with triangles. In an STL file, each and every triangular ‘facet’ is defined in terms of the coordinates of its three vertices. The triangles are tessellated, arranged in a gapless, repeating pattern of non-overlapping figures, to create the STL file. The triangulation of the surface causes faceting, in which the triangles or facets deviate from the design intent. When the faceting is too coarse you can see flat spots on curved surfaces when viewing the STL file. These flat spots will show up when the part is produced. To avoid this, it is important to find the right file resolution for your application.

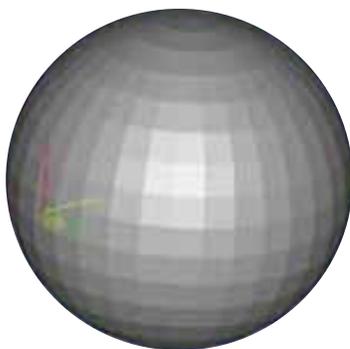
FILE RESOLUTION

Faceting can be controlled by the following settings when exporting the STL file from the CAD system:

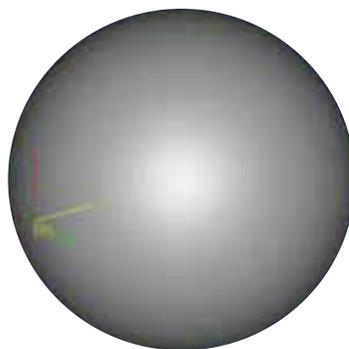
- Angle control – the maximum angular deviation between adjacent triangles
- Chordal tolerance/deviation – the maximum distance between the surface of the original design and the tessellated surface of the STL model

The majority of parts can be produced to meet the application’s requirements simply using the default export settings. However, more demanding applications often require specific settings to achieve the goals of the application. Coarse faceting is nearly always caused by the angle setting being too high or the deviation/chord settings being too high, or a combination of both. For example, a deviation/chord height setting around 0.0004 inch in SolidWorks will usually produce a good quality FDM part, but increasing the resolution may be necessary for parts with fine or rounded features.

But increasing the resolution will also increase file size. Large files take longer to transfer and process which could increase lead time and the cost of part production. A good rule of thumb is to keep file size below 15 Megabytes because going above this value will rarely improve part quality. The goal is to produce a file detailed enough that the features will build to the file dimensions while maintaining a manageable file size.



COARSE



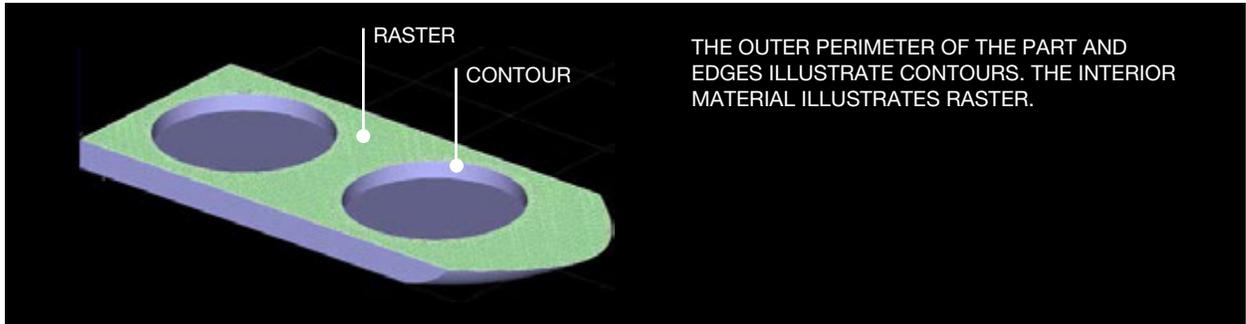
EXCESSIVELY FINE



GOOD

CONTOURS

A contour is the material deposited along the perimeter curve on each layer that makes up the outer extent of the part on that layer. Most parts have just a single contour but multiple contours can be used to increase the stiffness of the part and wall thickness. The thickness of the contour is equal to the extrusion path width, which is determined by the diameter of the tip used to produce the part. For example, a T16 tip has an inside diameter of 0.016 inch, a minimum extrusion path of 0.016 inch and a maximum extrusion path of 0.032 inch. The thickness of vertical walls should be at least twice the slice thickness in order to avoid creating flimsy walls that are prone to buckle.



RASTERS AND FILLS

The material used to fill the interior of the part between the contours is called a raster. The raster can be set to fill the interior with a sparse (honeycomb) or solid fill. A sparse fill requires a minimum of two contours to define the exterior of the layer and has an air gap between extrusions in the interior, reducing the cost and weight of the part. A sparse fill also adds porosity to the part which can be useful in situations where you need a lightweight part or certain tooling applications such as thermoforming and paper pulp molding. The sparse fill air gap can range from 0.001 to 0.5 inch while 0.080 to 0.150 inch is most common. An overall part wall thickness of 0.200 inch or greater is where you'll see cost savings from sparse fill since it uses less material than if you were to fill it solid.

No changes need to be made to the part geometry to use a sparse fill. Work with your project engineer or designer to determine if sparse fill is a good option for your part or application. Another option is to use a negative air gap on top level surfaces to compress the raster fill to a higher level of density than a solid fill. This approach provides a particularly good surface finish. Different fills can also be used in different areas of the part. For example, a thermoforming tool might have a normal air gap on the surface to impart a smooth finish on the produced part and a positive air gap in others so that air can be drawn through the tool.



TOLERANCES

In designing parts to be produced by CNC machining, a general tolerance such as ± 0.005 inch is often specified for areas of the part where no other tolerance is noted. This does not usually cause problems because tolerances of this magnitude are relatively easy to hold with CNC machining. For FDM parts, close tolerances should be specified when necessary. Precise tolerances are achievable with FDM, but secondary operations may be required which then requires leaving extra stock on your part design that can later be machined down to your specifications.

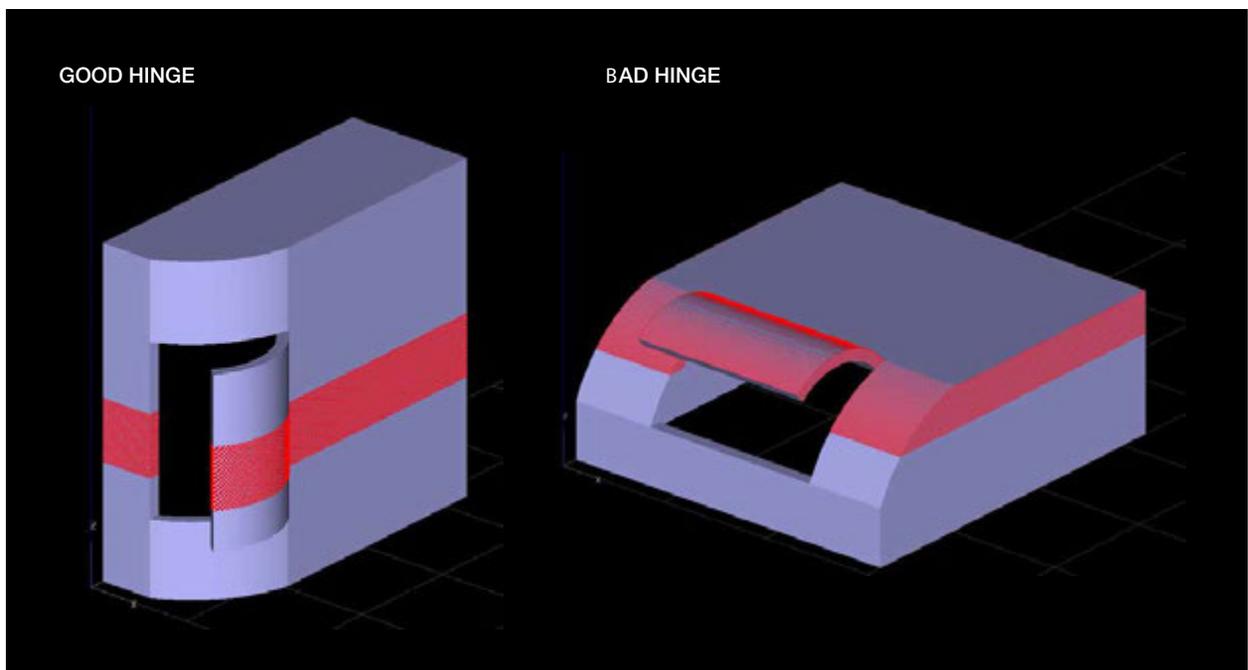


Close tolerances may also make it necessary to use a thinner slice thickness, which in turn gives you a more accurate part, but will drive up the build time and increase cost. Both cost and accuracy of an FDM part can be optimized by carefully considering the tolerance requirements and application, selecting the largest slice thickness that can be used to achieve these tolerances, then dimensioning the part in increments of this layer thickness. For example, if the decision is made to use a 0.010 inch layer thickness then whenever possible the z-axis dimensions should be defined in multiples of 0.010 inch, such as 0.050 and 0.100 inch. In that case, the part will be produced with near-perfect accuracy in the z-axis.

LIVING HINGES

A living hinge is a unique feature that is traditionally complicated to manufacture. Flexible materials, like FDM Nylon 12 which provides twice the elongation at break of SLS Nylon 12, make hinges easier to produce and can deliver scores of full 0 to 180 degree opens and closes. However, a living hinge should only be produced with FDM for prototyping or concept modeling applications with a short service life.

Living hinges should be designed and optimized with the best build orientation to last through a small amount of cycles – see an example below. The hinge wall needs to be thick and strong enough to withstand mechanical stress and bending. The hinge itself should be kept thin and will normally consist of two contours. If a 0.010 inch tip is used then the hinge can range from 0.032 to 0.040 inch total thickness.

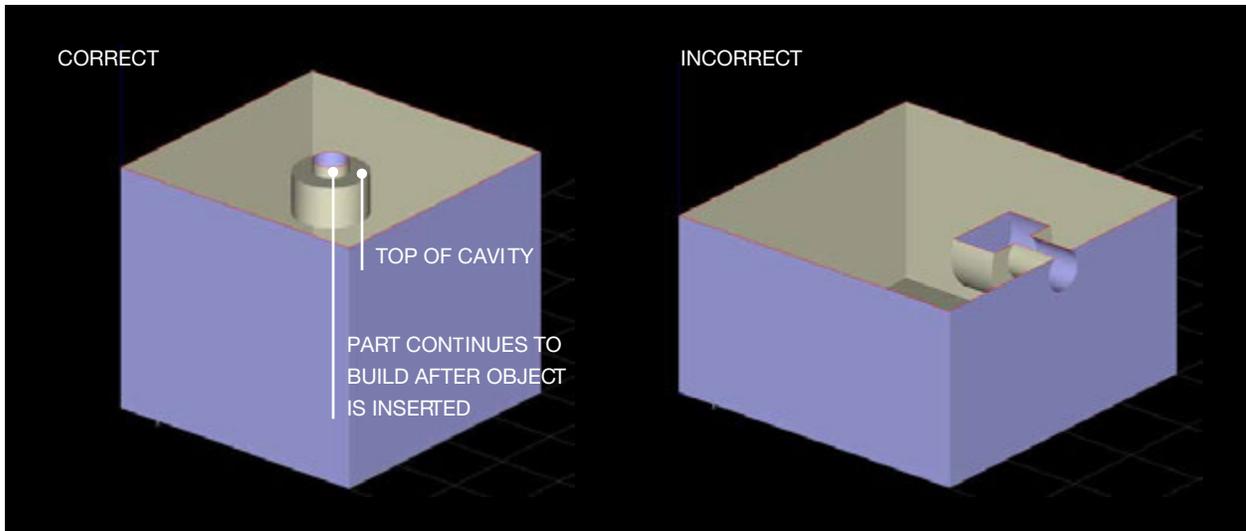


THE HINGE FEATURE, WHEN SLICED, IS ON THE SAME LAYER AS THE BODY OF THE PART. IT GIVES IT THE MOST STRENGTH.

IN THIS ORIENTATION, THE HINGE WOULD ONLY BE CONNECTED TO THE MAIN BODY WITH ONE LAYER.

FASTENING HARDWARE

Mechanical fasteners and inserts are common among prototyping and end use applications. With FDM, washers, nuts, bolts, threaded rods, or other objects can be inserted mid-build at the specified area without secondary operations, accelerating the process and decreasing delivery time. In order to embed hardware into the build, create a cavity in the design file with no clearance where the insert will be placed and let your project engineer know to adjust the clearance in the file based on tip size, depth and insert call out. The build can only be paused in the Z orientation in order to embed the insert in the clearance, therefore build orientation needs to be on the X/Y plane (see the illustration below).



THIS ILLUSTRATES THE CORRECT BUILD ORIENTATION FOR MID-BUILD INSERTS. ONCE THE PARTS BUILDS TO THE TOP OF THE CAVITY FOR THE OBJECT, WE PAUSE THE BUILD, INSERT THE OBJECT AND THEN RESUME THE BUILD.

THIS ORIENTATION IS IMPOSSIBLE FOR MID-BUILD INSERTS BECAUSE THE INSERTED OBJECT WOULD BE STICKING OUT ABOVE THE LAST LAYER PRINTED, PREVENTING THE PRINTER HEAD FROM BUILDING THE NEXT LAYER ON TOP OF THE OBJECT.

FDM parts can also be drilled and tapped, but the threads will only last through a few fastenings so heat staked inserts are preferable for most applications. Heat staked helicoils provide a high torque limit and greatly reduce the potential for the thread to be stripped with repeated use.

BOSSSES, RIBS AND GUSSETS

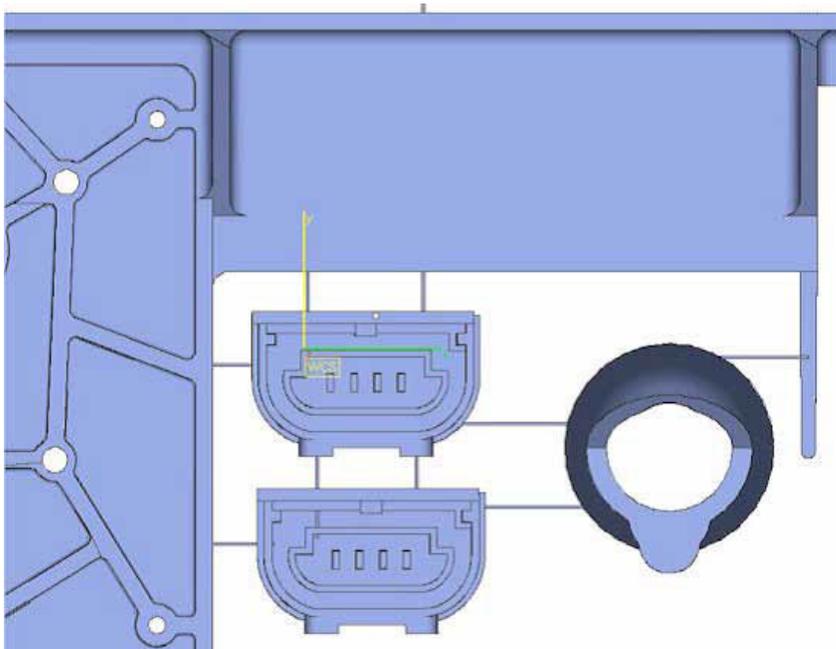
Bosses are usually added to a part to later be drilled and tapped. If the wall thickness of the boss is less than other areas of the part, it may be necessary to use a smaller tip size which will produce a more accurate hole. To ensure the correct diameter, consider reaming the hole to the correct size. For optimum performance, the boss diameter should be minimum 3x that of the insert diameter. A larger boss diameter decreases the chance of fracture when using interference fits and increases the strength of the boss.

Often ribs and gussets are used in injection molding to avoid warp or sink marks that occur during cooling. These additional features aren't necessary when using FDM unless they're critical to the part function, but can increase the amount of stress a feature can endure.

MULTIPLE SHELLS, NESTED OR TABBED PARTS

To ensure accurate quoting and quick delivery of assemblies, tabbed or nested parts, you should always save each individual piece as a separate STL file. If files aren't separate, they often appear as one part and will most likely be difficult to quote and build because you can't optimize orientation for each separate part. Tabbed parts are also difficult to build because the break-away walls are too thin. An example of a tabbed part is a model airplane kit. Each individual part is connected together by tabs that when twisted, remove individual components.

Multiple shells are possible and should be used when your part contains moving components, such as gears or a flag hinge. Just make sure to include enough air gap clearance between moving parts that will be built up with removable support material and leave an appropriate air gap. We recommend discussing orientation with a project engineer when producing multiple shells.



EXAMPLE OF A FILE WITH NESTED PARTS

INVERSED NORMALS, EDGES AND SURFACES

A clean STL file is referred to as “water-tight”, meaning there are no missing surfaces in the file, the surfaces do not overlap, and there aren’t any inversed or bad edges. Typically most solids modeling CAD applications, such as SolidWorks, Inventor or ProE, produce clean, water-tight files. CAD applications based on surfaces, such as Rhino, or STLs that come straight from scanned data may not be clean and need to be repaired before processing.

You can easily find errors in your STL file by using an STL viewing tool. All tools provide visual information, but some also offer specific information on areas of concern within your native file. If surfaces aren’t trimmed correctly, the 3D printer will not be able to properly read the data, causing delays in production.

TEXT

Adding text can enhance the aesthetics of parts, whether it is to brand a part with a company’s name or to customize the look of a part. However, there are a few guidelines to follow when adding text to a part’s surface. The minimum suggested text is 16 point bold on the top or bottom build plane of an FDM model and 10 point bold on vertical walls. Text above the surface of the part is more efficient to produce than cut text because support material is easier to remove from the surface than it is in small

STL EXPORT UNITS

Each CAD program has unique STL export settings. As a rule of thumb, make sure to set your STL export units to inches or millimeters. FDM’s software only interprets inches or millimeters so if your STL export from CAD is in centimeters, the geometry will be correct, but the part won’t be built to scale.

Production doesn’t begin when we hit print; it begins in CAD with your design. Each CAD system is different, but understanding your data, designing your parts for 3D printing and properly exporting STLs accelerates the manufacturing process so your order arrives on your desk sooner than you needed and exactly how you designed it. Follow the suggestions in this guide and consult with experts at RedEye to start making better files and improving your production process.

StratasysDirect.com
28309 Ave Crocker
Valencia, CA 91355

- ISO 9001:2008 Certified
- AS9100 Revision C Certified
- ITAR Registered

© 2016 Stratasys Direct Manufacturing. All rights reserved. Stratasys Direct Manufacturing is a business unit of Stratasys, Inc. Stratasys, Stratasys Direct Manufacturing, FDM, FDM technology, Fused Deposition Modeling, ABS, ABS-ESD7, ABS-M30, ABS-M30i, ABSi, PPSF, FDM Nylon 12, and PC-ISO are trademarks or registered trademarks of Stratasys, Inc. and/or its subsidiaries or affiliates and may be registered in some jurisdictions. PolyJet is a trademark or registered trademark of Stratasys Ltd. and/or its subsidiaries or affiliates and may be registered in certain jurisdictions.

ULTEM is a trademark of SABIC Innovative Plastics IP BV.