3D Printing: Quick tips for going from CAD design to printed object

Siemens PLM Software

Designing for 3D printing: quick tips

Whether you are using 3D printing for your amusement or your income, at the core of the printing process is the design.If you're used to traditional manufacturing, you'll discover that your approach to design as well as manufacture needs to change for 3D printing.

When your design is complete, there are post-design steps, including model orientation and parameter setting, which can help make sure your print comes out as you intended.

As an additional consideration, most printers can print with a percentage of infill. Understanding infill can help you print an object that won't warp or break, while using a lot less material and enabling faster printing.

A final factor in the success or failure of your 3D print is whether it securely attaches to the print bed; a print that detaches during printing will make all of your hard work for naught.

In this eBook, we'll help you understand the 3D printing process, and offer easy-to-apply tips for designing for 3D printing, post-design techniques to make printing easier, and strategies for making sure your print attaches to your print bed.

This eBook primarily deals with tips for fused deposition modeling (FDM) style printers, although users with other printer types may find it useful as well.
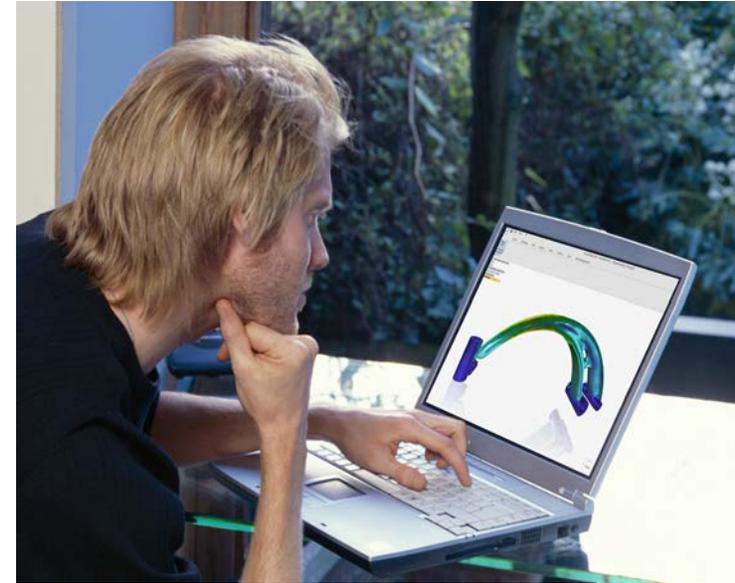
The 3D printing process

Regardless of the printing technologies and methods you ultimately use, the process of getting to an actual 3D print is the same.

Design your object

All 3D prints start with a design. When the design you want to print is your own, you'll need to use 3D modeling, or computer-aided design (CAD), software to bring your idea to reality. The object can be as simple or complex as you want, although it's best to avoid models that are very thin or small.

Save the file in a print-friendly format

In order to get any object to print, it must be saved as a print-friendly file, like a .stl file extension (STL). STL is the de facto standard format for printing. This file type breaks an object's surface into triangles to produce the shape. Easier shapes require fewer triangles, whereas the more complex the shape, the more triangles may be needed. There are other formats that are used for 3D printing today, for example, the *.3MF format developed by Microsoft, but STL is still the most common.

Saving a (CAD) file is as easy as save-as. However, the quality of the print can be improved by adjusting a few settings when saving to STL, such as conversion tolerance and plane angle. The lower the conversion factor and the better the angle, the smoother your print will become.

Open the file in a slicing engine

Most, if not all, 3D printers come out of the box with their own slicing engines. A slicing engine will take the STL CAD file and slice it into individual layers, and then generate the code the printer uses to print the file.

Orient the model in the build space

Once build parameters have been set, the model or models must be nested onto the build plate. Multiple items can be nested onto a single build plate; this means the build time is longer than printing a single object, but should take less time overall. We'll discuss tips for orienting your model later in this eBook.

Set the parameters

The slicing software allows the user to control parameters such as speed, flow, nozzle temperature and build plate temperature.

Most slicing engines have easy settings meant specifically for beginners. However, most also have more advanced settings to allow experienced users to get the best possible prints. You can adjust the percentage of infill, how much support material is needed, and what type of raft you want (a small, thin base that is printed for a good foundation for the print that can be snapped off). The variables are endless, depending on brand of printer, but are easy enough to get through.

Send the G-code file to the printer

Once parameters have been set and you are happy with the nesting, orientation and quality, then it's time to print! Simply hit "send to printer" or print, then find something to do while you wait for your physical part to be printed, which could take a few minutes to a few hours, depending on the complexity of your design.

Postprocessing

Postprocessing consists of separating the print from the build plate and removing any support material by melting, snapping or dissolving (based on your printer's specifications). A light sanding or a good cleaning may be needed, but a print done correctly should look pretty good already. Postprocessing can also include placing plastic prints in a bag of acetone for smoothing, gluing to other objects (if your design is larger than your 3D printer, or certain parts or object require different orientation to the 3D printer bed), drilling holes and painting.

| Design object in 3D CAD | Save object as STL file | Open file in slicing engine | Orient object for best printing | Set print parameters | Send object to 3D printer | Post process print (cleanup) |
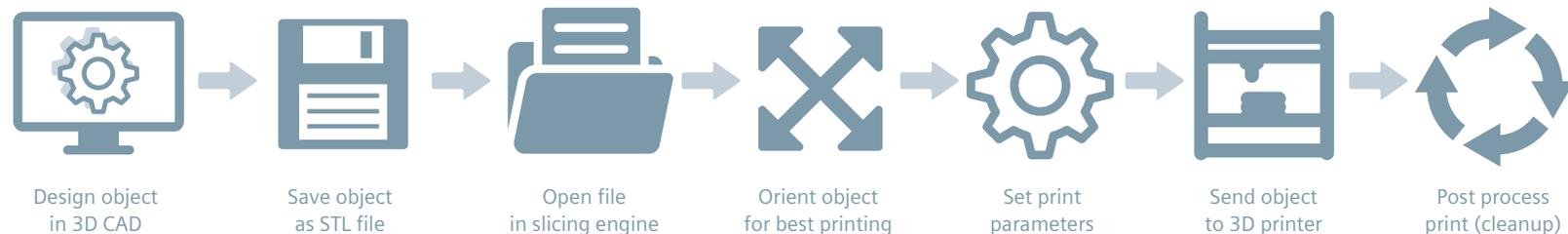
Design with your 3D printer in mind

Avoid sharp corners

When your geometry includes abrupt changes, like a vertical wall intersecting a horizontal floor, the printer can struggle to adapt, and end up creating rough internal surfaces with too much applied material. You can prevent that result with two easy fixes: add chamfers to ease transitions, or add fillets to ease the corner and gradually build up to the vertical surface. Adding rounds will also strengthen those points, as sharp corners are prone to breaking more easily.

Avoid thin walls/small features

FDM technology extrudes hot plastic through a nozzle to print each layer of your object. Extruded plastic can only be thinned to a limited extent, which is controlled by the orifice size and the speed of the moving head. Very thin-walled structures do not print well, and may end up in a stringy mess. If they are able to be printed, they tend to be fragile and easily broken.

Avoid walls that are too thick

If your walls are too thick, on the other hand, your print can become brittle, which can cause the walls to crack. This is especially important when you start to print with different materials, as excessive thickness can create internal stress during the printing process. Even when working with plastic, walls that are too thick also use an unnecessary amount of filament, and can take longer to print.

Avoid large overhangs

While they can create amazing shapes and surfaces, 3D printers cannot print into thin air. Anywhere there is a void with material above, support material must be used. Most slicers do this automatically, but parameters can be used to determine how the support structure is oriented and how much is used. Single nozzle printers create a series of thin columns that must be snapped off, which can lead to not-so-smooth surfaces. For this reason, it's generally best to reduce large overhangs when possible to minimize the need for support structures.

If a large overhang must be used, try orienting the print upside down. Most printers can handle up to about a 45-degree overhang. At certain heights, the tip of the overhang may string up a bit. This may require some trial-and-error testing to determine what your printer can handle.

Holes will shrink

Bear in mind that your part will be printed using heated plastic, which does shrink due to cooling process. For that reason, holes and critical features may need to be oversized to make sure they are close to the correct size after printing.

If your object requires a hole with a tight tolerance, however, it's better to 3D print the hole smaller than required, and then enlarge the hole with the proper-size drill. This is true especially for holes that are parallel to the print bed.

Make use of elephant feet

For objects that do not have a lot of surface area coming in contact with the build surface, you risk the print coming loose mid-build. Prevent this by incorporating elephant feet for any legs of the model that go down to the build plate:flare out the geometry of the model to add material as the leg reaches the buildplate.

There are other ways to make sure you are properly attaching your object to the build plate and we'll talk about those in a few pages.

Know these post-design tricks

Keeping design strategies in mind will make printing easier, but there are also some post-design tricks that you should know.

Orient round objects round face down

Orient the model to use the least amount of support material, ideally with any large flat planes in contact with the print bed. Also orient round or cylindrical objects with the rounded sides facing up towards the printer nozzle; if you looking down from the top of the printer, you should be able to see the circle the round object forms. Printing this way confirms the print comes out as symmetrical as possible, and delivers a structurally sound round shape.

Print voids and holes vertically

If there are any voids in your model (like a square tube), try orienting it vertically to reduce the amount of support material. If you lay a tube down and print it, the entire inside shape will need support, but standing on end, it doesn't need any.

This also applies to holes: To get the most concentric hole, it is best to have the printer create the hole vertically by making stacked rings. This prevents your holes from sagging or flattening out into ovals.

Set quality parameters

Quality parameters, such as conversion tolerance for the STL and slicer settings can give your print a surface quality that rivals a manufactured part. But higher quality means longer printing times. Think about your purpose when determining print quality: is this the final product or a prototype? Will it be visible or covered?

Quality also impacts the shape of any holes in your design. Holes in CAD files are really just a bunch of tiny straight lines at angles to each other. The higher the quality of the saved STL, the less the hole will look like an octagon.

Decrease layer thickness

For the best quality, especially for FDM printers, reducing layer thickness results in a finer quality print. This does increase print time, but the end results are well worth it!

Optimize infill

Objects do not need to be printed as a solid form for structural integrity. Much like the honeycomb structure produced by bees, printers can create a pattern of infill that balances strength and valuable filament savings. However, if you are using your print as a prototype to test the strength of an object that will be traditionally manufactured, or plan to put it under certain types of stress or pressure, solid printing may be the way to go.

Consider your materials

Carefully consider your material for a successful print. Each material has different properties; for example, thermoplastic polyurethane (TPU) and polylactic acid (PLA) have lower melting points than acrylonitrile butadiene styrene (ABS). Material type should also be a factor when considering support structures. An object printed in PLA can have supports made from PLA as they're relatively easy to snap off of the finished print. But ABS needs a separate support material and TPU is best left with no supports.

A few words about infill

Solid is not always better in 3D printing. Although printing a solid object can have its advantages, infill can save you precious filament and time.

The ability to print an object with a percentage of infill is a capability unique to 3D printing. And you don't have to design infill into your object as the slicing engine will take care of this for you. Generally, the only input it needs is for you to set a percentage – the closer to 100 percent, the more solid your object will be – and select a pattern, if applicable, for your printer.

In addition to saving time and filament, infill has a host of advantages.

Infill prevents warping
Printing large objects as one large chunk of plastic will make your print prone to warping. Using a lower percentage infill allows air to flow over the part as it's printed, resulting in more uniform cooling and preventing warpage.

Infill doesn't compromise strength
Printing in an infill pattern does not equate to a weak print. In many instances, you will find that your print created with infill is strong enough for your needs, and is lighter and uses less material.

Function can determine infill pattern
Most slicers offer a variety of infill patterns – knowing which one is best for you can depend on the goal of your printed object. A standard rectangular infill is great for ease of printing, while honeycomb and triangle shapes create additional strength. And infill patterns like waves or wiggles can help your object bend or twist.

What's the right percentage of infill?
In general, the strength of your object increases as your infill percentage goes up. The default infill setting on most printers is around 20 percent, and that is fine for some applications, but too much or too little for others. Think about the stress that will be put on your object, and increase the infill for objects or areas that need more strength. If an object doesn't need to be very strong, reduce the infill as much as possible to speed printing and save filament.

You'll likely find that determining the right infill percentage for your needs comes from a bit of trial-and-error testing.

Decide how to attach your 3D print to the bed

Skirt, brim and raft: The terms may sound funny, but they describe three primary ways to attach your 3D print to the bed. Let's review each way and its uses.

Skirt

With a skirt, you simply create several racetrack laps around the object at the beginning of the printing step to make sure the plastic is flowing. The skirt does not actually attach to the object at all. The skirt surrounds your print, and helps prime your FDM-style printer. With a skirt, the nozzle on the printer gets a good flow of hot malleable thermoplastic before starting the build. This ensures good adhesion to the print bed and a nice, smooth object.

Brim

The brim is a wide printed area connected to your primary object, adjacent to the object footprint (like a hat brim). The brim is almost like a skirt, except that it attaches to the model. In addition to the benefits of a skirt, the brim has the added advantage of holding the edges of the print to the bed.

In addition, when printing an object, often the outside of the print will cool faster than the middle, which can cause the edges to curl up. Using a brim helps prevent curling by holding those edges down.

Raft

A raft is a detachable base– a thin lattice platform– under the entire object (so the object is sitting on the raft). To create a raft, the printer will lay out a two- or three-layer-thick flat plate before printing the object.

Rafts provide excellent bed adhesion as well as a sturdy foundation for the print. This is especially beneficial for very small prints, odd-shaped prints that would not stick well to a bed, as well as thin wall objects.

Most rafts snap off fairly easily when the print is complete.

A tip for printers that do not have heated beds

Rafts are also useful when working with a printer that does not have heated bed, making adherence to the bed an obstacle.

An alternate method is to use painter's tape on the bed platform, confirming the edges of the tape wrap down around the sides of the bed if possible (this helps preserve the print bed as well). You can also use kapton tape for this, but the price point is generally higher.

If warping is still happening or the prints are coming loose, use a washable glue stick on the painter's tape to add an extra bit of adhesion.
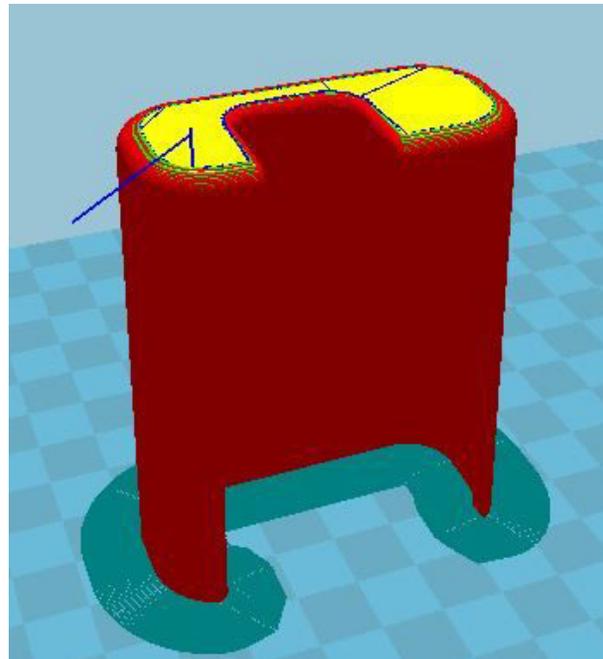
Wrapping it all up

Anticipate the design needs of your 3D printer and prepare your model with 3D printing in mind

3D printing is part science and part art. Effective design for 3D printing is a combination of understanding the printing process and adapting to it; and understanding the goals for your object and designing with those goals in mind. Practicing these design techniques and keeping the purpose of your print project in mind will help you optimize performance.

When it comes to 3D printing, not all design software is equal

Don't let your design be hampered by the capabilities of your software. Solid Edge® software from Siemens PLM Software includes tools that help you design your components to take advantage of the latest 3D printing techniques, and to prepare and output your designs to different 3D printing hardware and services.

Take your ideas to the next level with new design techniques for 3D printing

Solid Edge enables you to explore new concepts with generative design: designers define a specific material, design space, permissible loads and constraints and a target weight, and the software automatically computes the geometric solution. The resulting complex shapes can be immediately manufactured on 3D printers.

You can also use 3D scanned data for your 3D print designs. Solid Edge allows you to seamlessly combine traditional solid modeling boundary representation (b-rep) with triangular mesh models without time-consuming and error-prone conversions, reducing rework.

If you've already downloaded an STL you want to print, our unique synchronous technology means that you can open it in Solid Edge and quickly and easily customize it and/or make changes before printing.

Print on your own machine, or seamlessly send to a printing service

From Solid Edge, you can easily print on your 3D printer, or upload your models to the cloud to receive instant quotes for manufacturing your part in different materials, and have the part shipped to your door.

Solid Edge supports output of your part models to 3D printers using the 3D print command. Write out your parts in STL and 3MF formats, or send your parts directly to the Microsoft 3D Builder app. If you don't have your own 3D printer, or if you want to experiment with different materials and finishes, Solid Edge allows you to send your designs directly to cloud-based 3D printing services like 3YOURMIND. You can get quotes for manufacturing the part in different materials and shipping times for delivery to your door.
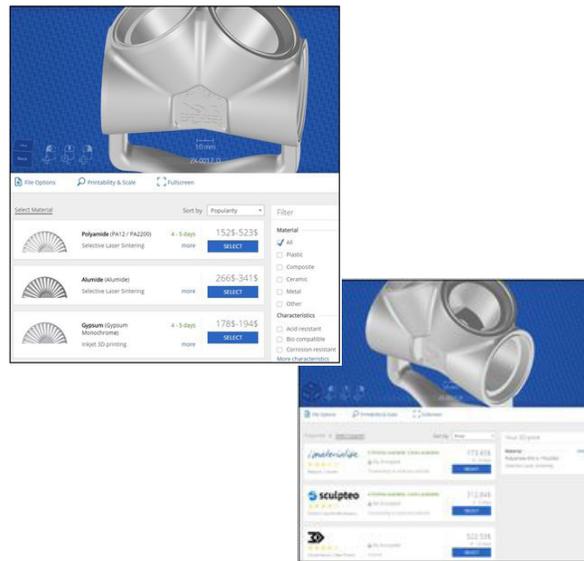
Start 3D printing with Solid Edge for free

Try Solid Edge for free
Start designing parts for 3D printing with a free trial of Solid Edge. Learn more: *www.siemens.com/plm/try-solid-edge*

Download Solid Edge for students
From high school students to self-taught learners, students of all types can take advantage of our free version for students, which includes advanced product development capabilities as well as the ability to output to an STL or 3MF file for 3D printing. Learn more: *www.siemens.com/plm/student*

Apply for Solid Edge for startups
Early stage startups that meet basic eligibility criteria can apply to get the professional version of Solid Edge for free. Easily design parts and prototypes for 3D printing. Learn more: *www.siemens.com/plm/startup*