

# 7 Tips for Solving Common 3D Printing Challenges



1

## Thin Walls

When outside perimeters and infill setting create weak and thin parts, you can solve it by increasing the number of outside perimeters to match the intended wall width and/or increase the infill percentage by 100% to create a solid thin part. Note that software matters. Gap Fill settings from Simplify3D provide assistance for thin wall models.

2

## Support Structures

When designing a part, remember how it will be built. The first layers will be placed onto the build plate directly, which becomes the structure on which to build the next layer. When your part design deviates out from the initial layer, you'll need a structure to build it on. Note that the structure, too, begins from the build plate and must be layered along with the part itself.

3

## Damaging Supports

Whether your supports are made from the same material as your printed part or a soluble material will matter in the end. For same-material supports, you must allow for an easy breakaway from the part. If your model walls are thin, you'll need strong support structures, so if the process settings aren't adjusted properly the support structures will be too dense, without enough separation from the part. To avoid breakage when removing the supports, create two dense support layers with 55% density, and use one upper and one lower separation layer to aid in the removal of the support structures.

4

## Warping and Curling

Warping and curling of parts when they come off the build platform is a common frustration. The solution is to first prepare your machine by leveling and calibrating the build plate, then producing a test chip to confirm first layer height and consistency. Also, remove all plastic residue and clean the plate with water, beer, or isopropyl alcohol. Then apply an adhesion agent of your choice—hair spray or glue stick. For process settings, be sure you have a consistent and steady build plate temperature for proper part adhesion and retention. Then use a rectilinear or Grid infill pattern to reduce warping compared with Honeycomb patterns. You can also enable a brim to add surface area to the part and to ensure proper part adhesion.

Sponsored by



# 7 Tips for Solving Common 3D Printing Challenges

5

## **Blobs and Zits**

Pretty much every material is susceptible to blobs and zits, which causes a poor overall surface finish. These anomalies can occur when the retraction settings are not set correctly. The solution is to pay attention to proper settings by setting your retraction distance at 1.10 mm and by choosing a start point closest to a specific location.

6

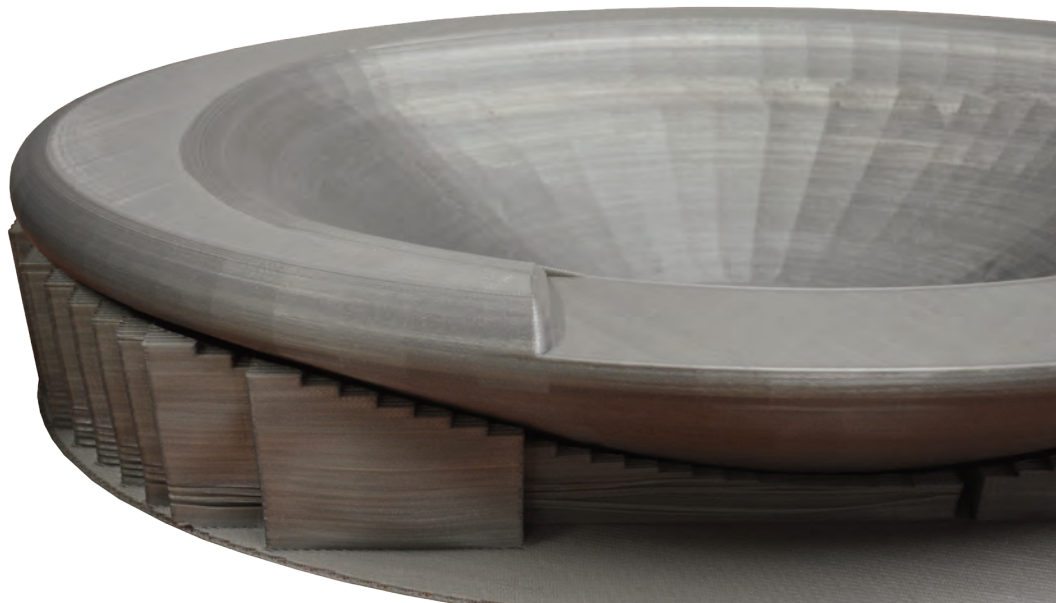
## **Oozing Filament**

If your print isn't finishing as smoothly as you'd like and is bulging and oversized, it's due to a combination of factors. Your nozzle temperature may be too high, in which case you can reduce the temperature of the nozzle. To verify this, raise the z-axis, heat the nozzle, and extrude a length of filament. The bead should resemble a smooth lasso. Next, check print speed and increase it to 2,100 mm/min so the material won't get too hot and start to ooze. If flow rate is too high and pushing too much material through the nozzle, decrease the rate a little at a time. Be sure to make only one adjustment at a time so you can evaluate your progress as you go.

7

## **Start Small Before Printing Big**

Often parts will have certain sections or one area where the geometry creates difficult printing conditions. In these cases, you can save time and solve problems quickly by printing a small section of the part, testing different setting or support strategies before you print the full piece. Starting small before you print big may seem simple, but in the long run it will save time and material, getting you to a successful print faster.



Sponsored by

**3DP**  
3D PLATFORM