Creating Functional Parts

Creating functional components with your 3D printer can be a cost-effective way to save money. However, optimizing your printer's capabilities requires an understanding of its tolerances.

If you design and print a 10mm hole and cylinder both measured at 10mm, you may discover that they don't fit together properly. This is because most consumer-grade printers extrude along the center of the path generated by the slicer and as the layer is squished against the build plate or the previous layer it is forced outward. (see figure 1).

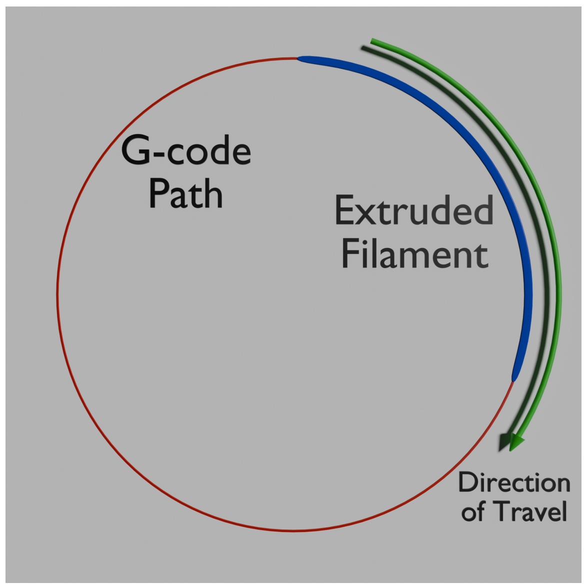

Figure 1

In this image you can see how the extruded filament is wider that the g-code path the slicer produced. In this instance the inside diameter would be undersized and the outside diameter would be oversived

Here's where tolerances come into play. How closely do the physical dimensions match the digital ones? Depending on the feature being printed, some physical measurements may be slightly undersized, while others may be oversized.

To address this, a simple process using a caliper and basic math is required. I've created a simple worksheet to assist in this. It also can be used to track multiple printers tolerances for reference later. You can find it here Start by printing a test piece and measuring its X, Y, and Z dimensions, as well as the inside diameter (ID) of the hole and the outside diameter (OD) of the cylinder with your calipers. Record these values. Then, subtract the physical measurements from the digital measurements for each dimension. Record these differences in the "Difference" row.

Now, when designing your functional parts, simply add or subtract these difference values. For example, if the test piece measured 20.1mm on the X-axis, subtract this from the 20mm digital measurement, resulting in a -0.1mm difference on the X-axis. Adjust your functional part's digital X-axis measurement accordingly by decreasing it by 0.1mm.

Note that when designing a part with a tight fit into a hole or slot, you don't necessarily need to compensate on both parts. You can adjust the hole size, the part fitting into the hole, or split the difference and adjust both by half as much. Keep in mind that this will create a tight fit. If you want a looser fit for parts to spin or slide freely, consider increasing the difference by 0.2mm.

Update:
Thanks to **u/Gabriebr95** and **u/Bluewing** over on Reddit for bring these factors to my attention which would have otherwise gone overlooked

If you really want to dial your printer in for the most accuracy possible you should also consider shrinkage due to part cooling and the print orientation.

Part shrinkage: As your filament is heated and liquified it has the tendency to expand and is it cools it does the reverse. Some materials have pretty negligible shrinkage amounts while others are considerable. Look at PVDF in the chart below, it can have up to 4%! At the same time even materials with low shrinkage can have noticeable effects on larger parts. If you have a PLA part that is designed to be 100mm long, after the part cools (and shrinks) it could be up to 0.5mm shorter. This could lead to holes and other features being misaligned.

Material	Shrinkage %	Average Shrinkage %
PLA	0.3 - 0.5	0.4
PETG	0.2 - 1.0	0.6
Nylon 12	0.7 - 2.0	1.35
Nylon 6-6	0.7 - 3.0	1.85
ABS	0.7 - 1.6	1.15
ASA	0.4 - 0.7	0.55
PP	1.0 - 3.0	2.0
HIPS	0.2 - 0.8	0.5
PC/ABS	0.5 - 0.7	0.6
Fiber-reinforced Nylon	0.5 - 1.0	0.75
PEEK	1.2 - 1.5	1.35
Fiber-reinforced PEEK	0.5 - 0.8	0.65
PVDF	2.0 - 4.0	3.0

*Percentage shrinkage of various plastics used in FFF 3D printing. Source: SpecialChem.com

Print Orentation: Print orientation plays a part in knowing your printers and designing around it weaknesses. Due to gravity any parts of your model printing in air (without anything below) will sag. The amount of sag is dependant on the size of the overhanging part. You should try to design or orientate your models in a way that minimizes the amount of overhangs. Ofcourse there will be times when this is not possible. In those cases, supports are one of the easiest way to reduce sagging.

Little design tip: If you are designing a hole that is going to be printed paralle to the build plate, the top of the hole should be pulled upward into a “teardrop” shape. This will reduce sagging inside the hole and you would need supports for that feature.