Understanding Fits

Look at your smart phone case. It snaps on and off well because there is a slight gap between the outside of the phone and the inside of the case. What if the width of your case is *exactly* the same as the width of your phone? You would need a hammer to put it on, and good luck getting it off!

This gap is called ***Fit***. When designing any parts that go together, it is important that the fit is not too tight or loose.

There are generally two types of fits: ***clearance*** and ***interference***.

With a *clearance fit* (Figure 1), there is some space (clearance) between mating parts. This allows them to slide nicely. With an *interference fit* (Figure 2), the internal part is too large and must be deformed to fit inside, causing an extremely tight connection. For something like a puzzle, clearance is key.

Notice that the dimensions are given in a range. This range is called the ***tolerance***. Don't worry too much about this now, just understand that the dimensions of a manufactured part will vary slightly because no manufacturing process is perfect (3D printing especially!)

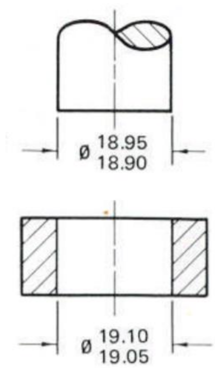


Figure 1: Clearance Fit

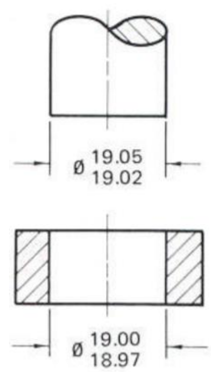
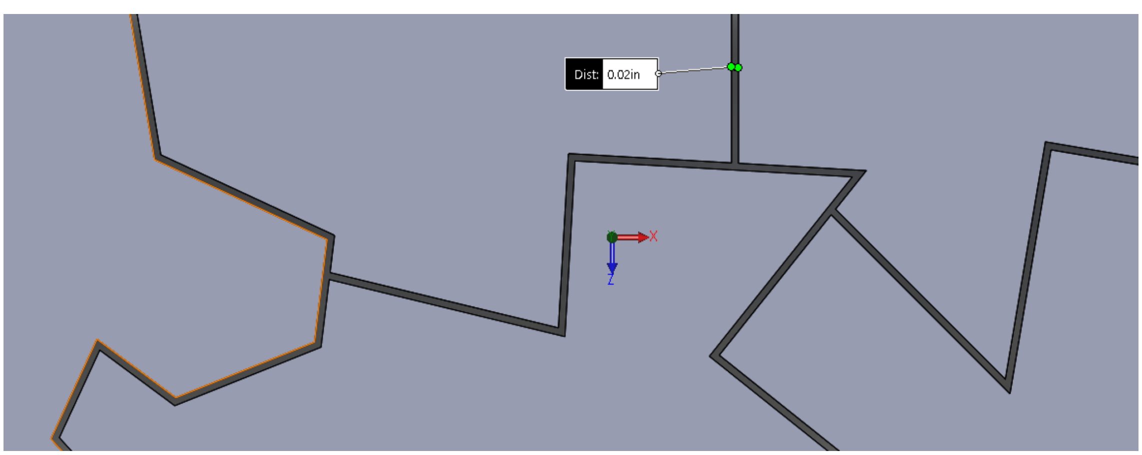


Figure 2: Interference Fit

Here are some general rules to keep in mind when designing parts that must fit together:

- Always make sure the outer part (e.g. hole) is larger than the inner part (e.g. rod)
- As a rule of thumb, start with a **0.5mm (0.02in)** gap between mating parts
  - If the fit is a bit too snug, you can always sand things down. Prototyping is not an exact science!
- Example: Even with strangely shaped pieces, there is a consistent gap between each part.



Design for 3D Printing

The type of 3D printing done at University of New Haven is called Fused Deposition Modeling (FDM). In this process, layers of melted plastic are stacked on top of one another to form 3D geometry. Think of a computer-controlled hot glue gun.

There are lots of things to know when designing parts to be 3D printed and it would take you many hours of hands-on experience to understand everything. This website gives a good overview:

https://www.3dhubs.com/knowledge-base/how-design-parts-fdm-3d-printing

Here are the basics:

Overhangs:

Since 3D printers build up vertically, if there are features that have no material underneath them, they will collapse and look like Figure 3. For this part, a better way to print it would be to lay it down flat.

Instead of a 90 degree overhang, one can generally get away with angles shallower than 45 degrees (Figure 4). More than that and the geometry will need to be supported, or one can simply find another way to orient the part.

Figure 3

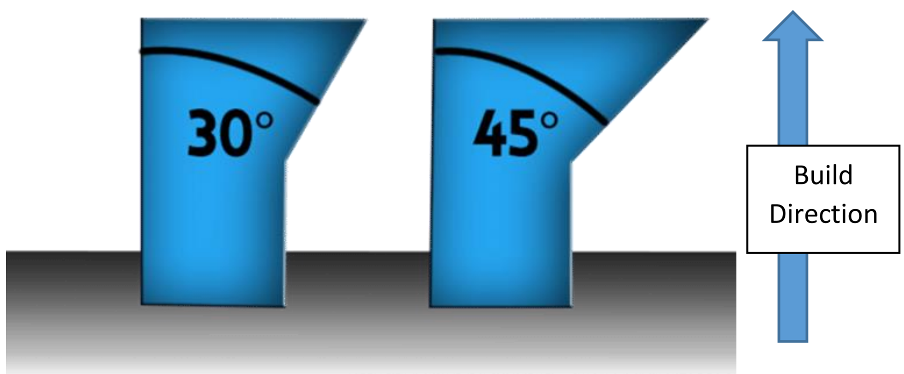


Figure 4

Feature Size/Resolution:

It is generally a good idea to avoid small features in 3D printed parts. Due to limits in resolution, some small details may not be visible in the final product, and small features may be incredibly fragile.

Figure 5 shows examples of holes in a 3D printed part. Notice how the small holes are increasingly distorted. As a general rule, holes should not be designed smaller than **2mm (0.08in) in diameter.**

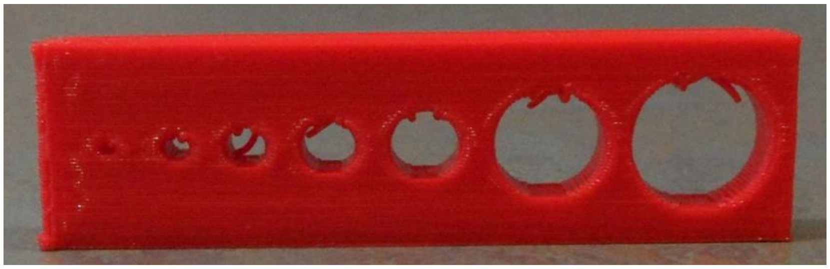


Figure 5

Figure 6 shows a designed part next to the final printed version. Notice that the thinner pieces were too fragile and snapped off. As a rule, make sure that every feature is at least 1mm (0.04in) thick. Even at this thickness it will still be fragile, but at least the part will likely print successfully.

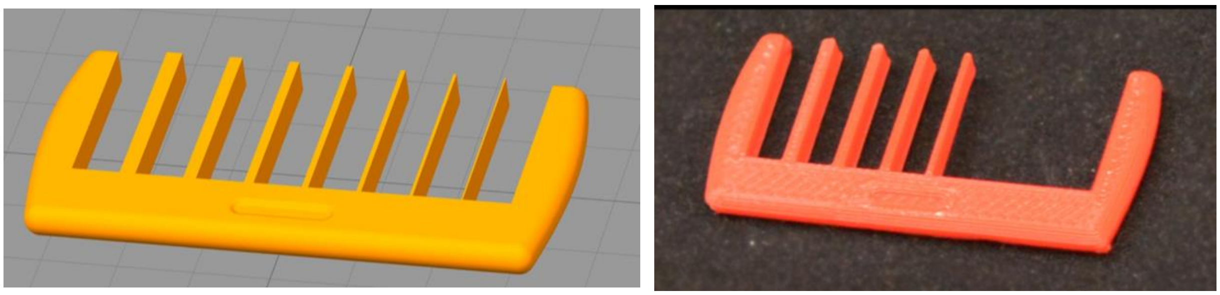


Figure 6

Warping:

When plastics heat up and cool down, they tend to expand and contract unevenly, leading to warping.

The best way to design around this is to avoid making parts that are wide and flat, such as in Figure 7.

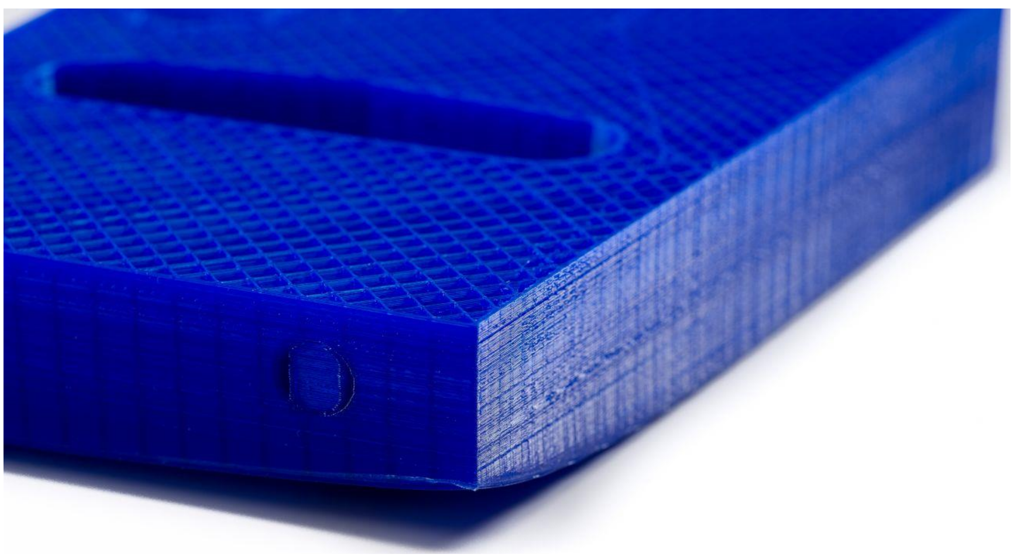


Figure 7

This guide is not meant to be exhaustive, but to make you aware of some general principles.

Questions/Concerns/Need Help?

For prototyping assistance: email prototype@newhaven.edu