Design for 3D Printing

Design for 3D Printing

Brent Seidel
modestconsulting@gmail.com
Modest Consulting

Modest Consulting

Outline

Introduction

A number of items need to be considered when designing an object for 3D printing. Some of them are generic while others may depend on the 3D printing technology used.

Measurements (units)

A good question is, "What does one unit in the CAD program translate to when printed out?". Depending on your application, the answer may range from mildly interesting to vitally important. If you are creating a stand alone decorative object, you may not care much as long as the result is a reasonable size. If, however, you are designing a part to physically interface with some existing items, the answer is vitally important.

Measurements (accuracy)

There are three different items that impact the detail and accuracy of the print.

- What is the resolution for printing bits of material? (note that x and y resolution may be different)
- What is the smallest bit of material that can be printed?
- What is the layer height?

Some or all of these may be adjustable on your printer. In general, reducing the resolution makes the print go faster.

Measurements (Resolution)

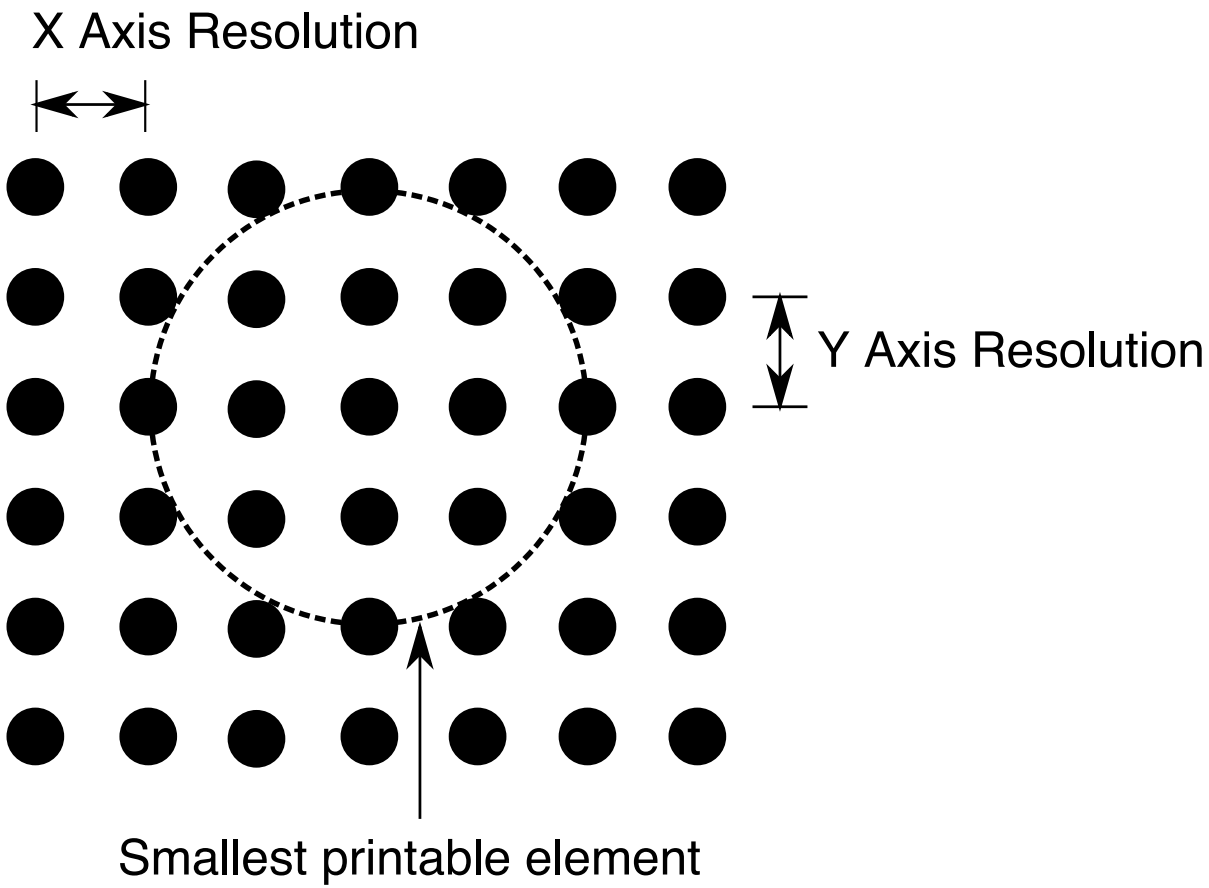

Figure: Measurements and Resolution

First Layer

Depending on the type of printer and its settings, the first layer printed may be thinner or thicker that the subsequent layers. This is usually accompanied with the material extending beyond or within the subsequent layers. This effect is sometimes called “elephant foot”. The slicer program often has a setting for “elephant foot compensation”.

First Layer

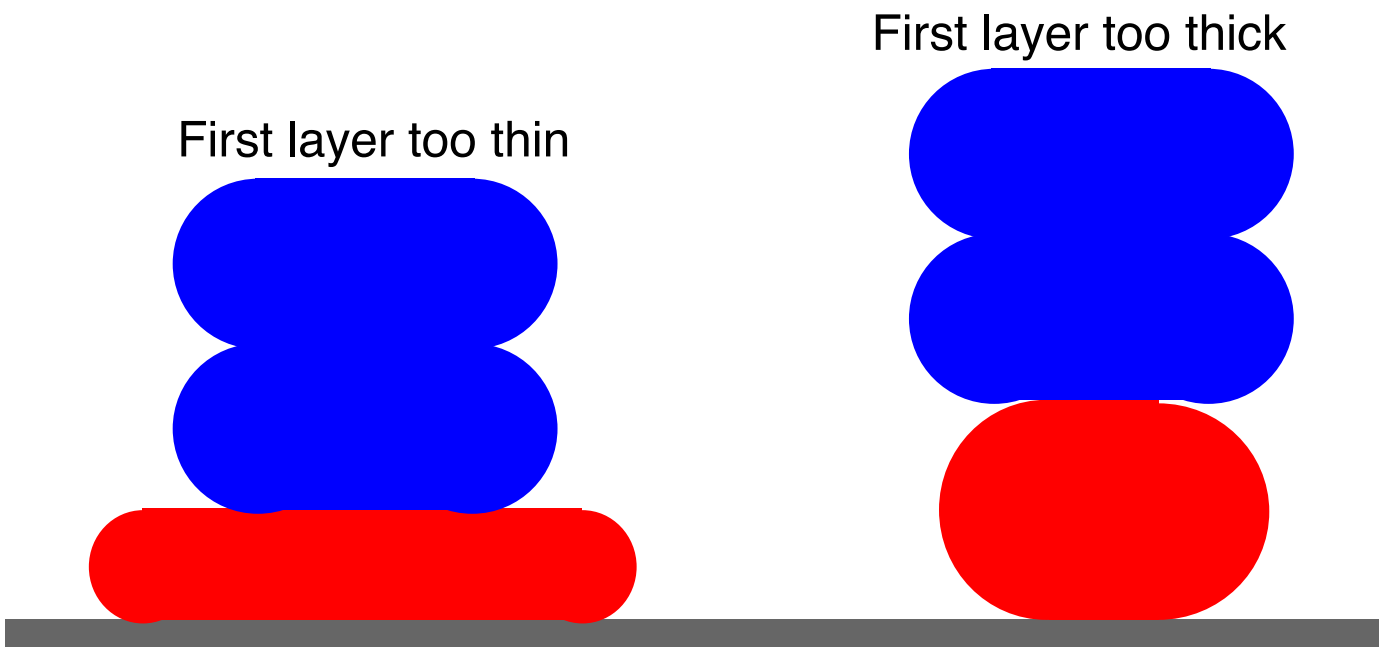

Figure: First Layer Problems

Holes and Voids

The size of holes in a 3D printed object are generally less than specified because holes are printed as polygons. The effect is more pronounced when the polygon has a small number of sides.

Holes and Voids

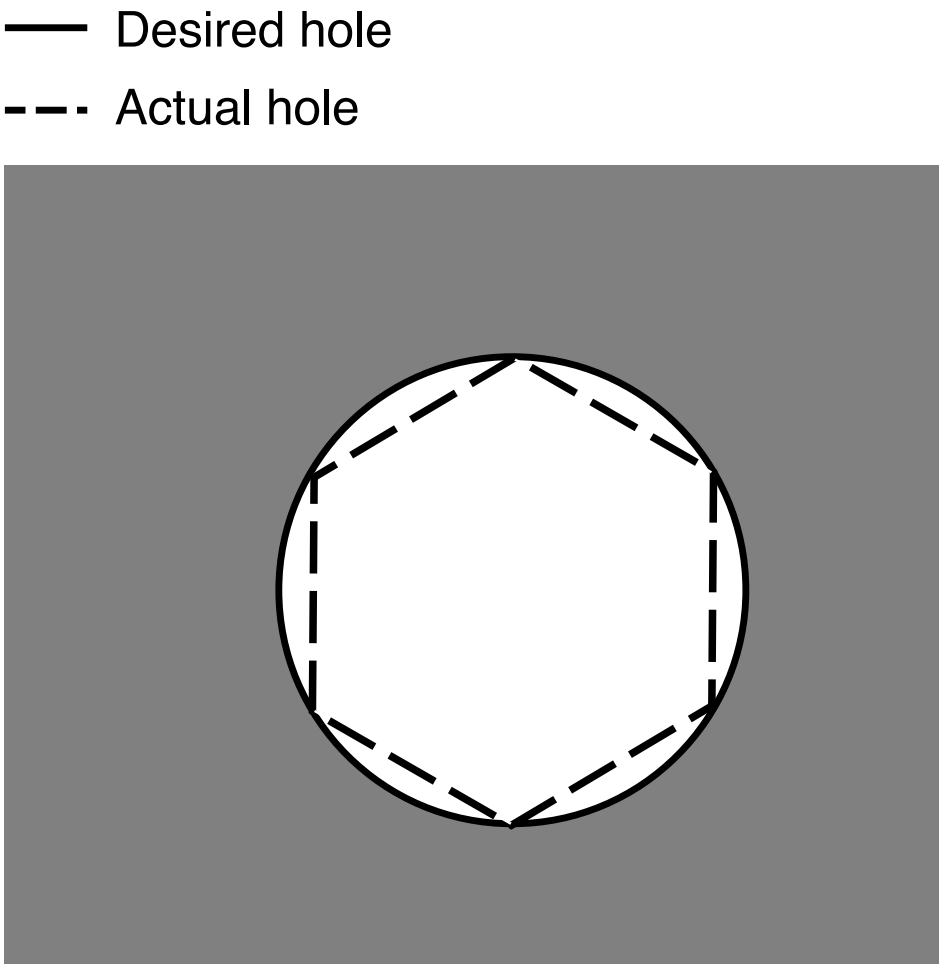

Figure: Desired Hole vs Actual Hole

Corners

Due to the nature of 3D printing, there are limits to how sharp corners can be made. In the discussion below, the following definitions are used:

- Corner Angle – The angle of the two sides that come together to make the corner.
- Sharpness – The size of the fillet formed at the corner.

Corners

Your CAD program may be able to specify infinitely thin lines. The printer is limited to the smallest printable element size.

- Thin dark line – path from design.
- Thick reddish line – actual path printed (width of line is the size of the smallest printable element).

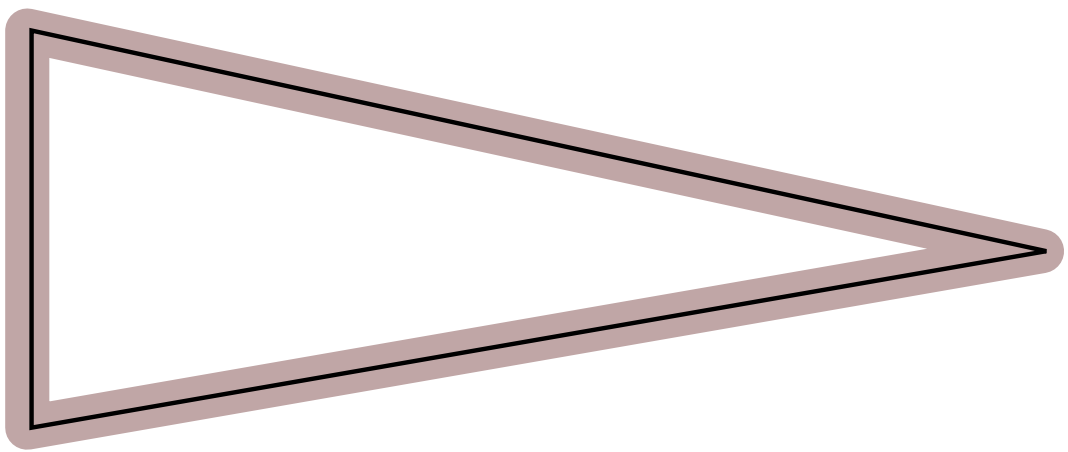

Figure: Design Path vs Actual Printed Path

Corners–Exterior

Exterior corner sharpness is limited by the size of the smallest printable element. The smaller the size, the sharper the corner. While drawing programs may offer several different options for joining lines at corners, the printer can only do a rounded corner.

Corners–Interior

Interior corner sharpness is limited more by the material and the printing process. Usually, one doesn't want an interior corner to be sharp as it causes a stress concentration thus making the part weaker. In most cases, one would be more likely to add a fillet to the corner.

Overhangs

- For some types of printers (typically those that fuse a powder), overhangs and unsupported sections are not a problem. For the rest, something may need to be done.
- A rule of thumb is that overhangs with an overhang angle less that 45° can print without too much trouble while larger angles will require support.
- There are test objects available online with varying overhang angles that you can use to see what your printer can do.

Angle of Overhangs

The angle of overhang is important in determining if your print will succeed or not.

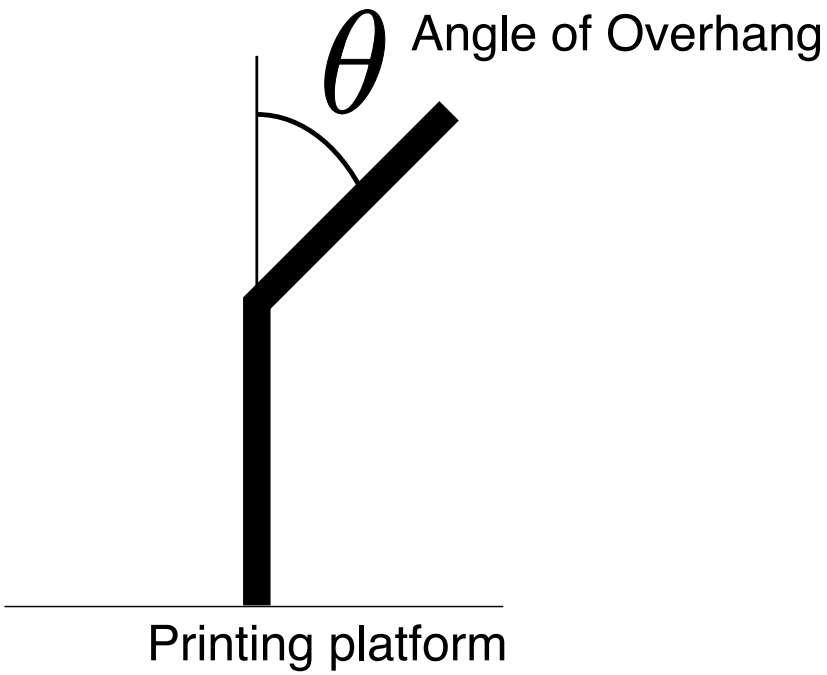

Figure: The Angle of Overhang

Unsupported Sections

If the Angle of Overhang exceeds 90° the printer will try to start printing a section on thin air (depending on the printer technology). Thus usually does not end well.

- You can choose a printer technology, such as powder fusing methods, where the section would be supported by unfused powder.
- You may be able to reorient your part so there is no longer an unsupported section.
- You may be able to use the slicer program to automatically generate a support structure.
- You may be able to add a support structure to your design.
- You may split your part into multiple pieces that get fastened together later
- Some combination of the above.

Unsupported Sections

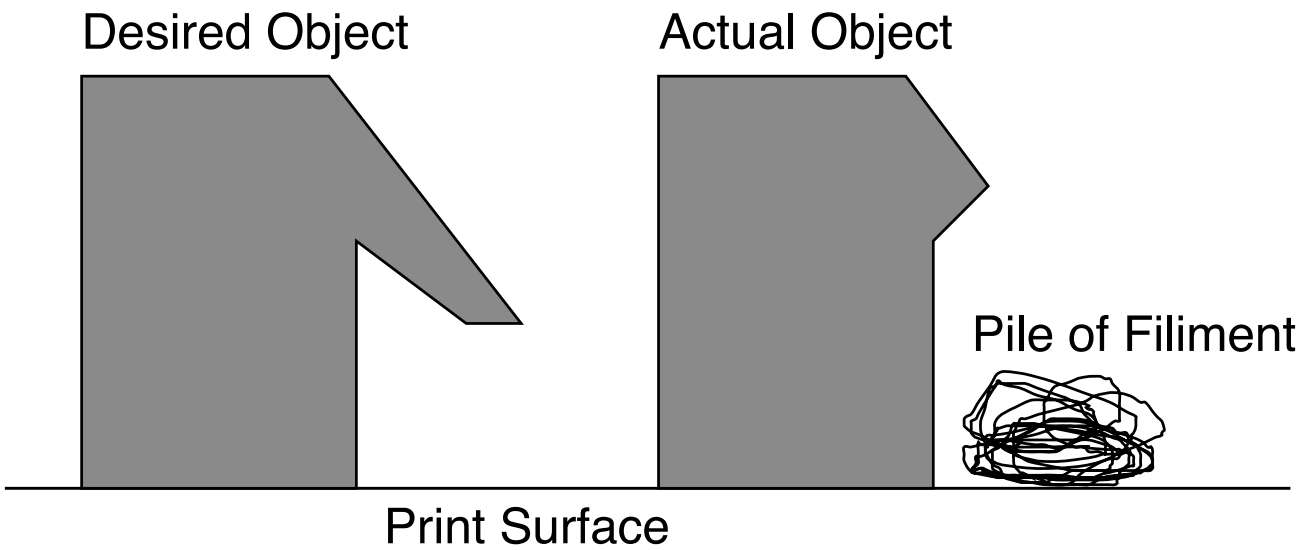

Figure: Unsupported Sections–Desired vs Actual

Unsupported Sections–Some Fixes

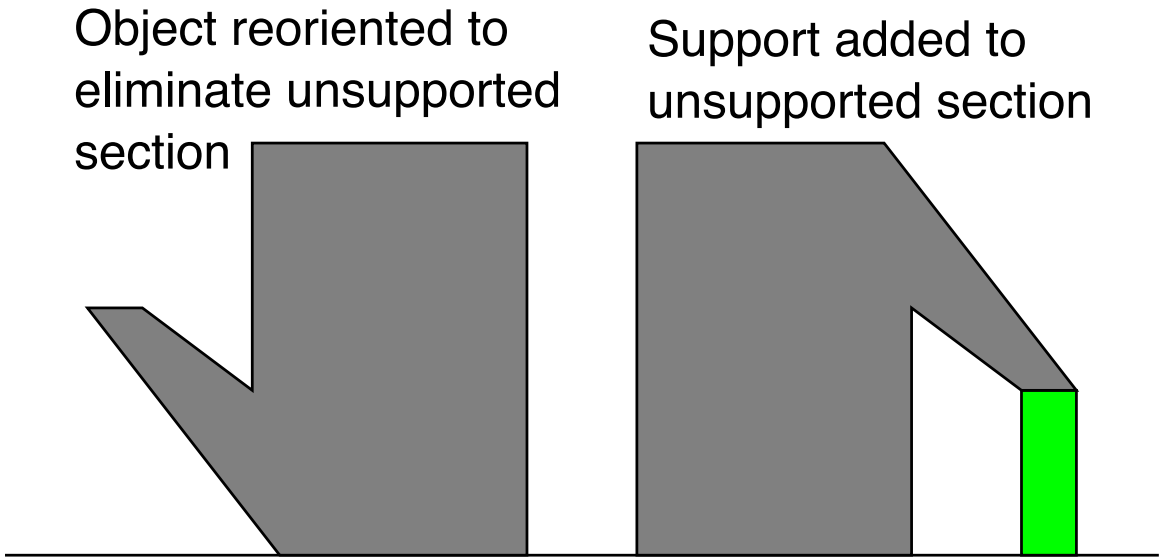

Figure: Some Fixes for Unsupported Sections

Dimensional Stability

It is likely that your 3D printer does not have the accuracy or resolution for this to be a major problem. Yet. As the technology improves, this may become more of a problem. If your design depends on tight tolerances, this will be something to consider.

The main reason that things change size is temperature. There is a good chance that the printing temperature will be different from the end use temperature. The same issues would apply if you were casting, forging, or machining a part.

In some cases, changes in temperature during the printing process can cause a part to deform. In this case, it would be a good idea to enclose the print area to stabilize the printing environment.

Tradeoffs

Generally there are a number of tradeoffs that can be made. Some of the more common ones are:

- Speed $\leftrightarrow$ Resolution
- Speed $\leftrightarrow$ Infill Density
- Cost $\leftrightarrow$ most anything else