Getting Started with 3D Printing

Introduction

This document is intended to get you started printing on your printer as quickly as possible. Once you understand how to get started printing, you will learn 3D best practices to help you get the best out of your designs. For more information, please visit NVBOTS.Zendesk.com.

How to Print

Here are a few simple steps to get started printing right away:

1. Make sure your printer is set up. Follow all instructions in the Equipment Manual to set up and bring your printer online.
2. Open your Internet browser (Chrome recommended) and navigate to https://my.NVBOTS.com and login using your credentials.
3. If you have your own .STL file, click on the Upload tab and upload your file here.

Figure 1. Top bar on NVCloud. The Upload tab is outlined in red.

Print Settings

General

Quality

The Quality setting affects the layer height. Fast quality will print quickly with thick layers that are 0.3mm in height. Standard Quality will print more slowly with thin layers that are 0.2mm in height.

Strength

The Strength slider affects the infill density of the part. The infill pattern can be configured between honeycomb, rectilinear, line, and concentric.

Support Material

This button can be pressed to turn on or off support material generation.

Support Angle

This value selects the angle threshold of overhangs to be supported on the printed part. This setting should typically left on Auto.

Support Pattern

Various patterns of support material may be selected.

Copies

The Copies setting affects the number of copies to be printed.

Transforms

Scale

The Scale slider affects the scale of the part. The Lock Scaling Ratio button can be turned off to allow for independent scaling in the X, Y, and Z axes.

Rotate

The Rotate slider affects the orientation of the part. Note: Holding the SHIFT key will snap the rotation angle to the nearest 45 degree increment.

Advanced

Top Layers

The Top Layers setting affects the number of solid layers to be printed on top surfaces.

Bottom Layers

The Bottom Layers setting affects the number of solid layers to be printed on bottom surfaces.

Perimeters

The Perimeters setting affects the number of perimeters to print on each layer.

XY Compensation

The XY Compensation setting grows/shrinks the object in the XY plane. Negative values shrink the part and positive values will grow the part. This feature is helpful for fitting assemblies together and for expanding models with thin walls.

3D Printing Best Practices

Part Orientation

Print orientation affects bulk and local mechanical properties. Printed parts are strongest in tensile loading along the layers in the XY plane. The lowest strength is across the layers in the Z direction. One of the most common failure modes of 3D printed parts is delamination between layers.
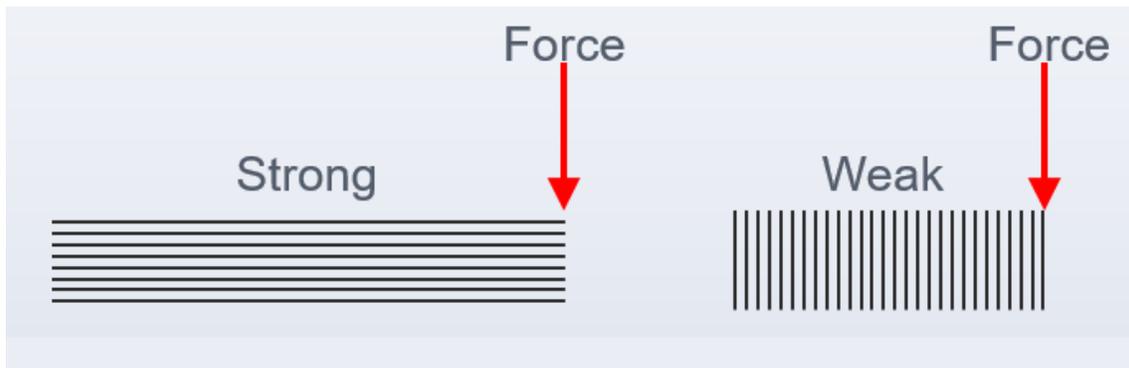

Figure 2. The effect of layer orientation on final part strength.

Infill Density

Increased infill density correlates to increased loading capacity.
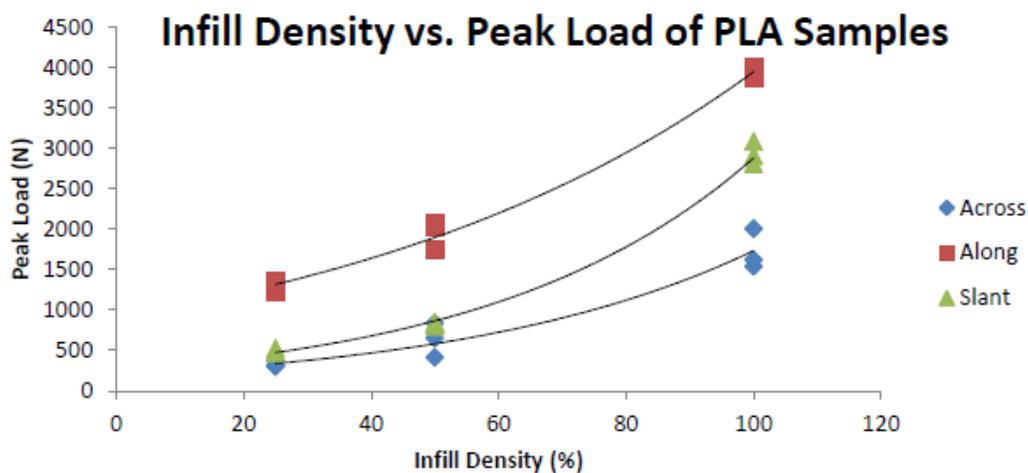

Figure 3. Infill density vs. peak tensile loading capacity of PLA samples. Tensile loading data is included for loading along the layers, on a 45 degree slant to the layers, and at 90 degrees across the layers.

Overhangs & Support Structures

Support structures allow for the printing of overhangs without compromising part quality. Typically, overhangs further than 45 degrees from vertical are supported. Bridges are unsupported features that are supported at two ends with a span in between. The span of bridges is typically unsupported.
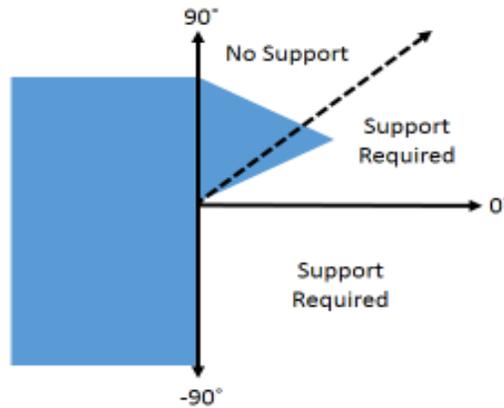

Figure 4. 3D printed overhangs typically require support material for an overhang angle of 45 degrees or greater from vertical.

Tolerances

Clearance Fit: shaft rotates or slides freely	Clearance = 0.5mm – 2mm
Transition Fit: shaft is held precisely but can be disassembled	Clearance = 0.25mm – 0.5mm
Interference Fit: shaft requires force for insertion	Interference = 0.0mm – 0.25mm

Note: 3D printed parts are oftentimes slightly oversized. Setting the "XY compensation" value to approximately -0.1mm will typically allow parts to fit together as designed.
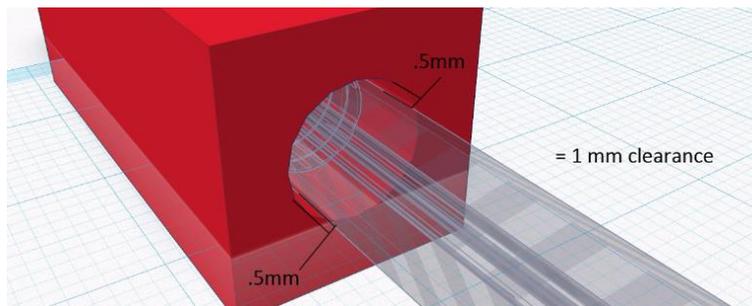

Figure 5. Altering clearances can allow printing of slip fit, transition fit, and interference fit components.

Holes

Holes are typically undersized on FFF parts. Drilling or reaming is recommended to generate properly sized holes.

Wall Thickness

A minimum wall thickness of 1mm will yield a strong surface. The minimum generated wall thickness is equal to the extrusion width at approximately 0.4mm.

Surface Finish

Surface finish is primarily affected by layer height:

Fast Quality (0.3mm Layer Height)	Surface has noticeable ridges
Standard Quality (0.2mm Layer Height)	Smoother surface finish

Note: Printing at Fast Quality will yield much quicker printing.
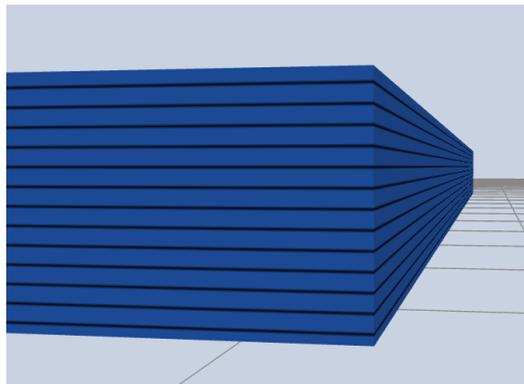

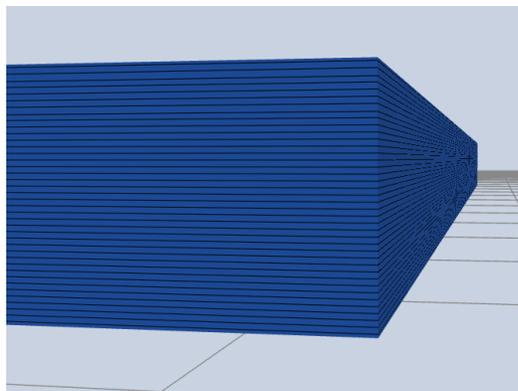*Figure 6. Print preview of a part set to Fast quality [0.3mm layer height].*

Figure 7. Print preview of a part set to Standard quality [0.2mm layer height].

Minimum Positive Feature Size

Minimum positive feature size is twice the nozzle diameter. Typical minimum positive feature size is approximately 1mm. Single walls may also be printed at 0.4mm.
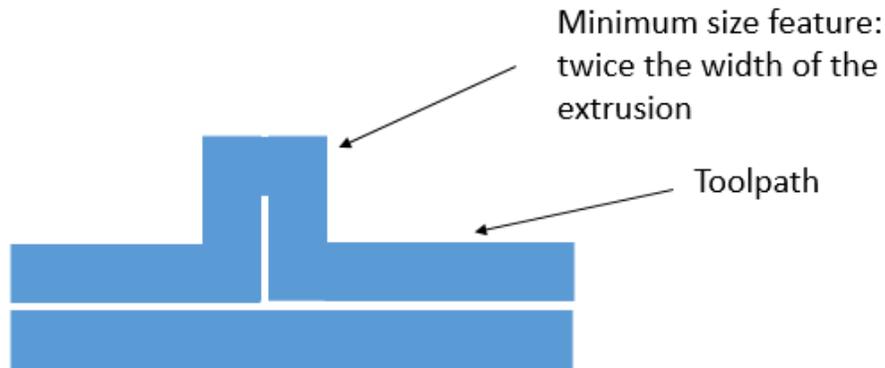

Figure 8. Minimum feature size is limited by the nozzle diameter.

Text

Text should be generated in 10 point font or greater. Text typically does not require support structures.

Post Processing

PLA can be worked similarly to ABS. The primary difference between ABS and PLA is that PLA has lower heat resistance, so low speeds should be used when performing machining operations to maintain part integrity. Options for post processing include:

- Machining operations
  - Turning
  - Milling
  - Cutting
- Sanding
- Painting
- Bonding