

# Clint's Simplify3D Settings Manual

This is my collection of information and documentation on the Simplify3D profile – the parameters and settings that control how Simplify3D (S3D) transforms the surface geometry of a 3D model (typically a .stl file) into the numerical control language used by most 3D printers (most typically a .gcode file).

I have collected this information from a wide range of sources. Organizing them into this single document has helped me to better control the complex transformations that S3D performs. However, this information is not authoritative nor an official voice of the authors of S3D. Also, the information pertains to a range of version of S3D from v3.1.1 through v4.0.1, so it is possible that there are inconsistencies across these versions.

This documentation follows the outline of the user interface and menu structure in S3D v4.0.1. Some of the parameters have been in other location in earlier versions of the application, and I have tried to relocate the documentation to the appropriate place in the v4.0.1 outline.

This document is available at [http://www.BreathFlute.com/pdf/S3D\\_SettingsCG.pdf](http://www.BreathFlute.com/pdf/S3D_SettingsCG.pdf). Use this link to access the most recent release of this PDF document.

You may also be interested in the document at [http://www.BreathFlute.com/pdf/S3D\\_ProfilesCG.pdf](http://www.BreathFlute.com/pdf/S3D_ProfilesCG.pdf), which examines twelve standard profiles for the Prusa i3 Mk3. That document is also bundled with copies of the 12 core profiles into a ZIP archive, available at [http://www.BreathFlute.com/zip/S3D\\_Pi3Mk3\\_CoreProfiles\\_CG.zip](http://www.BreathFlute.com/zip/S3D_Pi3Mk3_CoreProfiles_CG.zip).

Finally, on 5/8/2018 I release a copy of my own working core profile, based on what I've gleaned from all this profile research. You can access it, along with a description document, at [http://www.BreathFlute.com/zip/S3D\\_Pi3Mk3\\_CG\\_BF\\_20180504.zip](http://www.BreathFlute.com/zip/S3D_Pi3Mk3_CG_BF_20180504.zip).

*I hope you find this document helpful!*

— Clint Goss, Ph.D. [[clint@goss.com](mailto:clint@goss.com)], as of 5/8/2018

## Revision History

---

4/24/2018: Initial version made available on Prusa and S3D forums.

4/26/2018: A few additional images, and export to PDF with bookmarks to provide an outline for quick section access

4/30/2018: Add **Rafts Exclusively for Support Structures** (thanks to forum user Airscapes for pointing this out) and a substantial section on **Strength, Speed, Cost, and Quality**.

4/30/2018: (yes, two releases in one day) Regularize the Short IDs across this document and the corresponding *Profiles Comparison* and *History* documents.

5/8/2018: Incorporate Matt Harrison's *Extruder Calibration* procedure, Tom Sanladerer suggestions, and the **Metes and Bounds** section. Released along with the new Clint Goss Breath Flute core profile.

TBD: Navigation enhancements: Bookmark each setting and provide PDF links from SIDs and Setting names to the correct section.

## Sources

---

This documentation is drawn from several sources. In some cases, I have modified the content from these sources for clarity.

### Units, XML, and SID

% <firstLayerWidthPercentage> [FWid]

A box near the beginning of each section, like the one shown at the right, shows three items:

**Units.** The units of measurement (as shown on the user interface).

**XML Tag.** The ID of the parameter / setting entry in the .fff file which maintains the profile. This file is in XML format, and the tag in the file might look like this:

<firstLayerWidthPercentage>100</firstLayerWidthPercentage>

**SID.** “Short ID” developed by Clint Goss for display of profile settings in a reasonably condensed way. The example above would be listed as “FWid 100%”. See the **Brief Profile Display and SIDs** section below for an example.

### Tooltips (TT)

Pop-up “balloons” that appear when you rest the mouse / pointer over fields in the application.

Tooltips are shown in a box like this.

### Tip of the Day (ToD)

A series of posts on the S3D User Forum (<https://forum.simplify3d.com/>) during January–March 2015. All the ToD postings were made by S3D Forum user **KeyboardWarrior**. They provided detailed explanations of the various parameters implemented at that time. However, note that this documentation (apparently) precedes S3D v3.0, which was announced on [www.Simplify3D.com](http://www.Simplify3D.com) on June 23, 2015.

Most of the images that appear in this document are borrowed from these ToD posts. However ... many of these original images are screenshots of the S3D user interface, which is quite different than the current version of S3D. These screenshots have not been included, to avoid confusion.

Here are links to those specific posts:

- Extruder Tab – <https://forum.simplify3d.com/viewtopic.php?t=1980>
- Layer Tab – <https://forum.simplify3d.com/viewtopic.php?t=1941>
- Additions Tab – <https://forum.simplify3d.com/viewtopic.php?t=2100>
- Infill Tab – <https://forum.simplify3d.com/viewtopic.php?t=1953>
- Support Tab – <https://forum.simplify3d.com/viewtopic.php?t=2135>
  - Support Gap – <https://forum.simplify3d.com/viewtopic.php?t=1973>
- Temperature Tab – <https://forum.simplify3d.com/viewtopic.php?t=2365>
- Cooling Tab – <https://forum.simplify3d.com/viewtopic.php?t=2364>

- G-Code Tab – <https://forum.simplify3d.com/viewtopic.php?t=2366>
- Scripts Tab – <https://forum.simplify3d.com/viewtopic.php?t=1959>
- Speeds Tab (I can find no ToD for this tab ... maybe the tab did not exist before S3D v3.0?)
- Other Tab – <https://forum.simplify3d.com/viewtopic.php?t=2363>
  - Horizontal Size Compensation – <https://forum.simplify3d.com/viewtopic.php?t=2042>
- Advanced Tab – <https://forum.simplify3d.com/viewtopic.php?t=2089>

### ***ToD Discussions***

Posts by S3D Forum users in response to the ToD posts. I have chosen posts that shed additional light on the semantics of the settings.

### ***S3D Articles***

Information from web pages in the support section of [www.Simplify3D.com](http://www.Simplify3D.com). In particular, the articles that are provided at <https://www.simplify3d.com/support/articles/> can be extremely helpful!

### ***S3D Print Quality Troubleshooting Guide***

An on-line resource for diagnosing common issues with 3D-printed projects. This document shows images of issues that are tied directly to a particular setting, along with links to the on-line guide at <https://www.simplify3d.com/support/print-quality-troubleshooting/>.

### ***S3D Support***

Direct communications I have had with S3D support staff.

### ***Naver Blog Posts by 산이아빠***

Content from series of December 2017 blog posts on Naver.com by user 산이아빠 (pronounced “san-iappa” according to Google Translate) have been incorporated in this document. Although the text on these pages is in Korean, the images provided in these posts shed light on some of the more arcane parameters. I have cropped the images in some cases to zoom in on the key elements.

In some cases, I have been able to use the Google Translate service to get useful information from these posts. Since they are automatically (and possibly imprecisely) translated, I highlight these paragraphs with a light blue background (such as this paragraph).

Here are links to the specific posts:

- Advanced Tab – <https://m.blog.naver.com/blah82/221163773102>
- Other Tab – <https://m.blog.naver.com/blah82/221157164225>
- Speed Tab – <https://m.blog.naver.com/blah82/221156944385>
- Scripts Tab – <https://m.blog.naver.com/blah82/221156347415>
- G-Code Tab – <https://m.blog.naver.com/blah82/221156029522>
- Cooling Tab – <https://m.blog.naver.com/blah82/221155179969>

- Temperature Tab - <https://m.blog.naver.com/blah82/221155031794>
- Support Tab - <https://m.blog.naver.com/blah82/221153324594>
- Infill Tab - <https://m.blog.naver.com/blah82/221151466503>
- Additions Tab - <https://m.blog.naver.com/blah82/221151117319>
- Layer Tab - <https://m.blog.naver.com/blah82/221146713020>
- Extruder Tab - <https://m.blog.naver.com/blah82/221144580857>
- Process Menu - <https://m.blog.naver.com/blah82/221144035028>

### ***Notations by Clint Goss***

In-line notations that I have added are shown with a thin dashed line in the right margin of the text. These are typically things I've stumbled on or read about in newsgroups, so take them with a grain of salt since I'm still very new to 3D printing.

## Strength, Speed, Cost, and Quality

---

We generally choose slicer settings to balance the competing goals of strength, printing speed, cost of materials, and quality of the printed model. This section looks at a **very** general overview of this topic ... with some suggestions garnered from several cited sources.

### A Very Brief Summary

If your primary consideration is strength, I believe (after working through the sources listed in the next section) that these are the basic principles to remember:

- Fused filament is strongest along the axis that it was extruded / deposited. That means that setting **Internal Fill Pattern** to **Rectilinear** is typically the best choice for strength.
- Bonds between neighboring filament extrusions on the X-Y plane are weaker, and those between layers along the Z axis are weaker still. If you want a strong part, set the **Infill Angles** to be parallel to the axis along which you expect the greatest loads.
- The optimal number of **Outline/Perimeter Shells** for strength appears to be 3.
- The higher the **Infill Fill Percentage**, the stronger the part ... with one exception: Parts with 100% **Infill Fill Percentage** will take the most stress before they break, but they will “Yield” (permanently deform) at a lower strain than a 90% **Infill Fill Percentage**. Additionally, 100% infill can reduce the quality of the surface of the part. So, an **Infill Fill Percentage** setting of 90% may be the best all-around choice when you want a strong part.
- A **Primary Layer Height** of 0.2–0.4 mm is somewhat stronger than 0.10–0.15 mm.
- Printed parts are significantly weaker under loads along the Z axis versus the X and Y axes.

### Sources

Here are some research studies that have tested the effects of various settings and/or shed light on issues of choosing slicer settings for strength:

- [3DMATTER 2015] 3D Matter, *What is the influence of infill %, layer height and infill pattern on my 3D prints?*, March 10, 2015, <http://my3dmatter.com/influence-infill-layer-height-pattern/>
- [SMYTH 2015] Cliff Smyth, *Infill and Strength*, 2015, <http://3dprintingforbeginners.com/infill-strength/>
- [GRAHAM 2015] Michael Graham, *Mechanical Testing 3D Printed Parts: Results and Recommendations*, September 2, 2015, <https://engineerdog.com/2015/09/02/>
- [EMBURY 2016] Robert Embury, *Mechanical Properties of Toner Plastics Filament*, November 23, 2016, <http://toner-plastics.com/tesile-testing-results-of-toner-plastics-3d-filament/>



## Basic Metrics of Loads

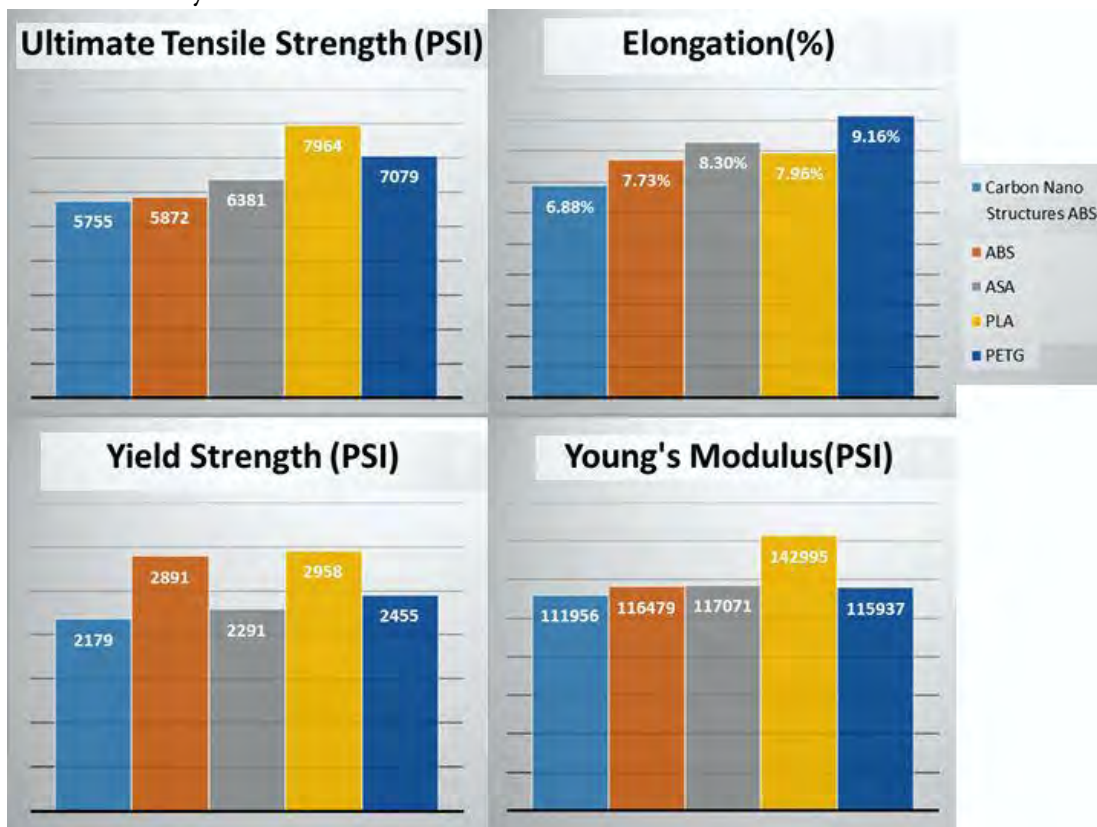
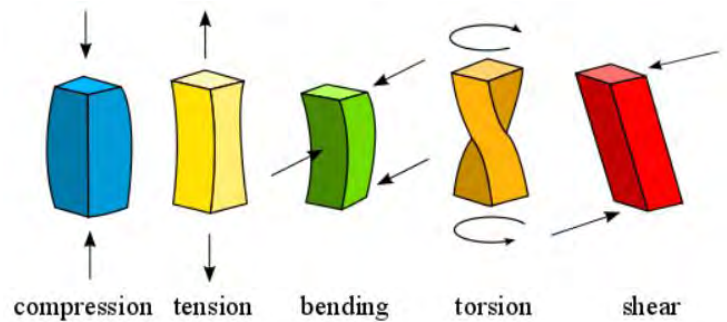
The basic orientations for loads in Stress-strain analysis are shown at the right. “Bending” is also called “Flexing” or “Flexure”, and “Torsion” is often called “Twisting”.

Destructive testing of parts under the various loads often report these metrics:

- **Maximum Stress** (“Max Stress”, “Ultimate Stress”, “Ultimate Tensile Strength”, UTS, or “Stress at Break”) is a measure of maximum stress at the breaking point of the part.
- **Yield Stress** (“Stress at Yield”) is a measure of strength at the point where the part experiences a permanent deformation of 0.2% of the original dimension.
- **Elongation at Break** is the percentage that the material is stretched at the point where it fails.

Other metrics of Stress-strain analysis that you may encounter (but which we do not typically use in this document) are:

- **Young Modulus** (“Young’s Modulus”, “rigidity”, or “Modulus of Elasticity”) is a measure of the stiffness of a given *material* (not a particular shape created from that material). This is basically a measure of a material’s ability to resist permanent deformation.
- **Geometric stiffness** is a measure that depends on the shape of the object.
- **Harness** is a measure of the resistance that the surface of the material imposes against penetration by a harder body.



- **Toughness** is the amount of energy that a material can absorb before it fractures.

## Material Characteristics

The summary diagram of the characteristics of materials shown at the right was composed from images in ([EMBURY 2016]).

## Layer Height and Infill

Here is a nice summary of effects of **Primary Layer Height** and **Infill Fill Percentage** on the four primary considerations ([3DMATTER 2015]):



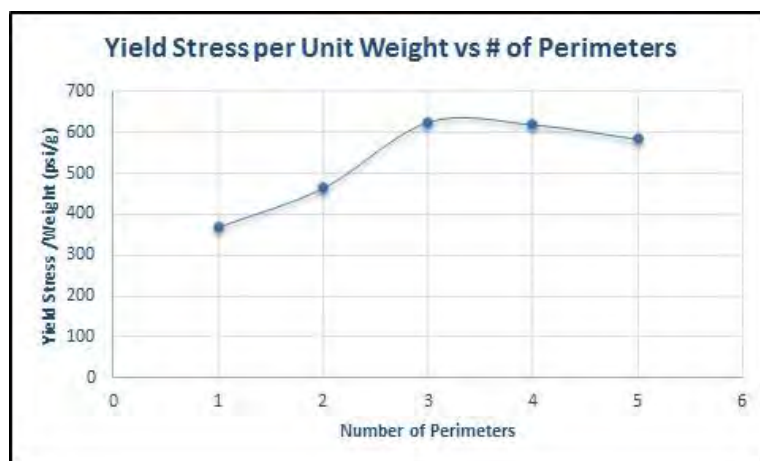
## 100% Infill and Quality

One counter-intuitive result from the summary charts above is the effect if **Infill Fill Percentage** on Quality. From footnote 2 of [3DMATTER 2015]:

*Infill % does not matter with regard to quality (it is inside) except at 100% infill where we have observed that the prints were not as smooth due to an excessive amount of material extruded.*

## Outline/Perimeter Shells and Bending Loads

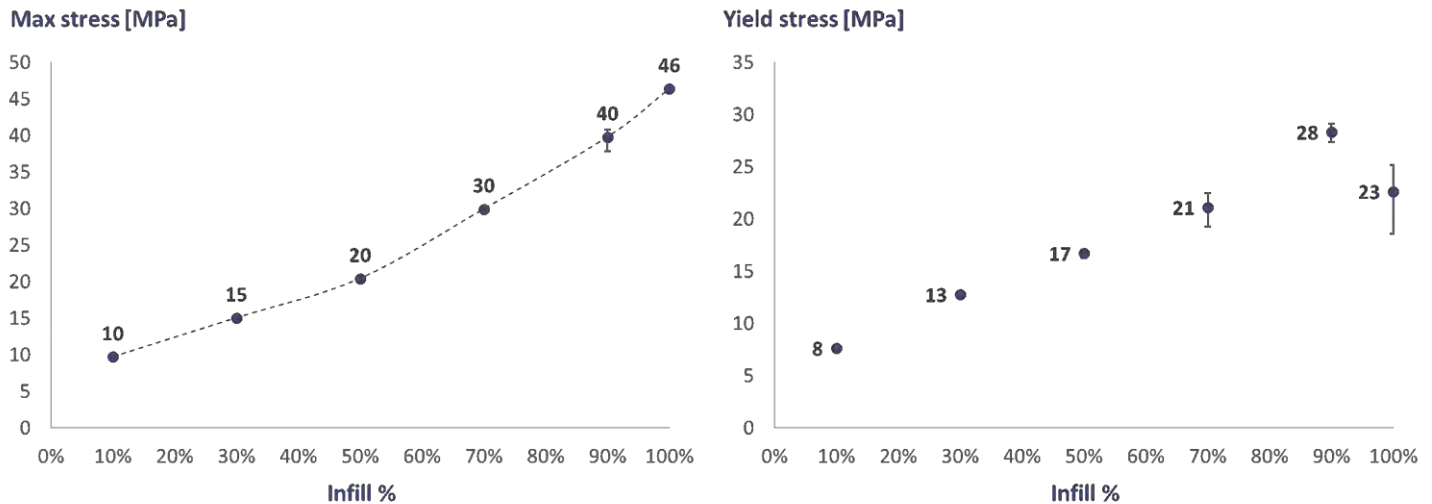
The [3DMATTER 2015] study applied bending loads parts printed with different settings for **Outline/Perimeter Shells** the measured the Yield Stress of each. It appears that there is a significant improvement to using 3 perimeter shells versus 1 or 2, but the benefit does not extend to higher number of shells.





## Infill Fill Percentage and Tension Loads

The [3DMATTER 2015] study compared the strength under tension loads across a range of **Infill Fill Percentage** settings. These tests were done with an infill pattern of “Linear”, which would correspond to an **Internal Fill Pattern** setting of **Rectilinear** with the **Infill Angles** parameter set to 0/90. This causes the extruded filament of the infill to line up with the direction of stress for these strength tests.



The Maximum Stress measurement (a measure of strength at the breaking point of the part) increased monotonically from 10% to 100% **Infill Fill Percentage**, as shown in the left chart. The relationship is not quite linear, with the higher percentages having a greater incremental effect on Maximum Stress.

The Yield Stress (a measure of strength at the point where the part experiences a permanent deformation of 0.2% of the original dimension) is plotted on the right chart. It shows an optimal **Infill Fill Percentage**, of 90%, with the 100% infill having significantly less strength. The authors opine that:

*For infill around 90%, the filaments touch and form a continuous 3D material, but it is porous because there are lots of small air voids in it (~10% of the specimen). In this case, the stress concentrates around the voids so the strain is localized around the void areas.*

*For 100% infill, the plastic filaments also touch but there are (nearly) no more air voids in the material. Therefore, the plastic deformation is not localized anymore and the whole specimen behaves as a single plastic filament would.*

*Yield stress increases from 8 MPa at 10% infill to 28 MPa at 90% infill, before decreasing back to 23MPa at 100% infill. The fact that yield stress is higher at 90% than at 100% infill is in line with our hypothesis ... [that] ... the stress is localized around the air voids at 90% so at a macro level, the material yields at a higher stress.*

### Internal Fill Pattern and Strength

The [3DMATTER 2015] study compared the strength under tension loads of three standard Infill patterns at 10% **Infill Fill Percentage**, and found them to be comparable – effectively indistinguishable within the error bars of the study. Note that:

- “Linear” corresponds to an **Internal Fill Pattern** setting of **Rectilinear** with the **Infill Angles** parameter set to 0/90,
- “Diagonal” corresponds to an **Internal Fill Pattern** setting of **Rectilinear** with the **Infill Angles** parameter set to -45 / 45, and
- “Hexagonal” appears (from the images provided in the study) to most closely correspond to an **Internal Fill Pattern** setting of **Full Honeycomb**.

The study also included two “non-standard” infill patterns that were significantly weaker.

Strength under bending loads was studied by Michael Graham in [GRAHAM 2015]. He tested various orientations of Rectilinear and Full Honeycomb and also found that the optimal **Internal Fill Pattern** is Rectilinear with the **Infill Angles** parameter set to 0/90.

### Primary Layer Height and Strength

The [3DMATTER 2015] study compared strength of printed parts across a range of **Primary Layer Heights** at 80% **Infill Fill Percentage** using a Linear pattern (corresponding to an **Internal Fill Pattern** setting of **Rectilinear** with the **Infill Angles** parameter set to 0/90).

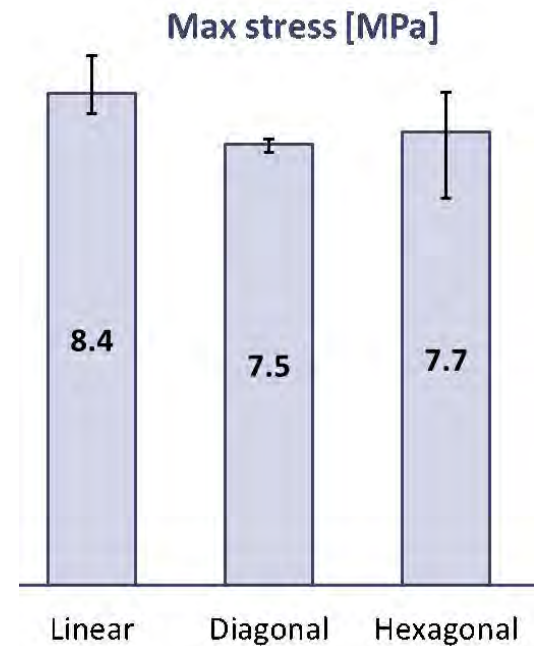
The Maximum Stress increased from 29MPa at 0.10mm **Primary Layer Height** to 33MPa at 0.15 and then ranged from 35–37MPa for **Primary Layer Height** of 0.20, 0.25, 0.30, and 0.40 mm.

The Yield Stress ranged from 22–26MPa for **Primary Layer Height** of 0.10–0.30 mm with no clear trend.

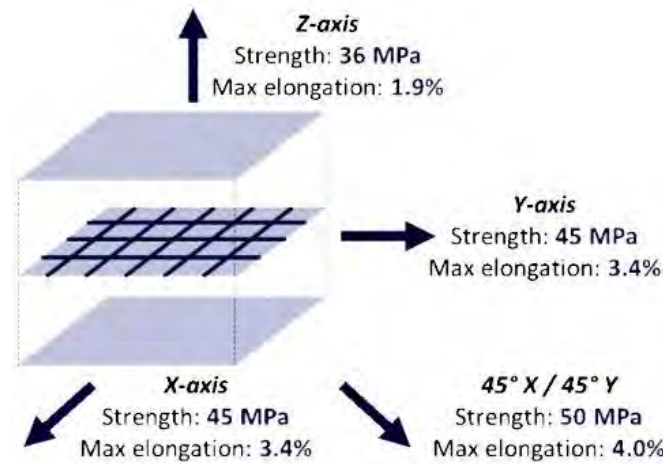
### Z-Axis Loads

It is the opinion of Cliff Smyth ([SMYTH 2015]) that higher **Infill Fill Percentage** improves strength for tension and compression loads in the Z-Axis. In other test done in that article (not directly Z-axis loads), lower **Infill Fill Percentage** settings accommodate more bending prior to failure, but though they failed at lighter loads.

Based on this – and failing more direct evidence of z-axis compression loads vs. infill percentage – I printed these feet shown at the right that support two of four legs of a weight-bearing rack using 100% **Infill Fill Percentage**.



The [3DMATTER 2015] study showed that there is an inherent weakness along the Z-axis, because the interface between layers is not as strong. For 100% infill parts under tension, the Z-axis direction is 20% to 30% weaker than other directions. The maximum elongation at break in the Z-axis is about half of the X-axis and Y-axis. Under tension loads 45° to the X and Y axes showed the highest strength:



### On the Horizon

And finally, two articles that look far beyond the static, repetitive patterns offered by the current generation of slicers:

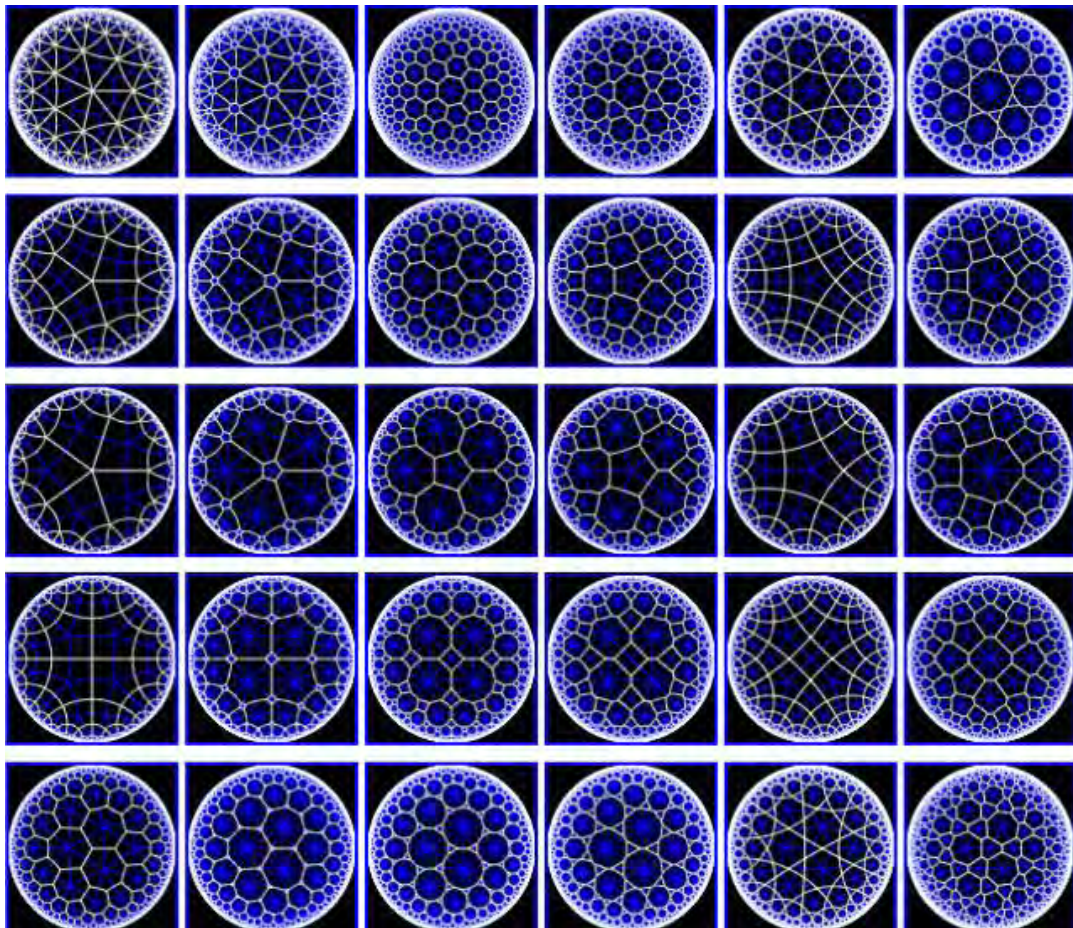
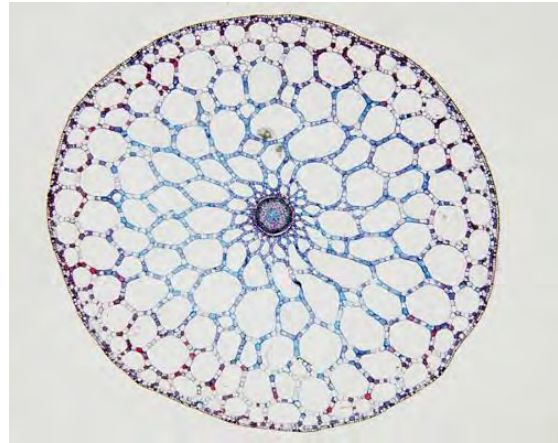
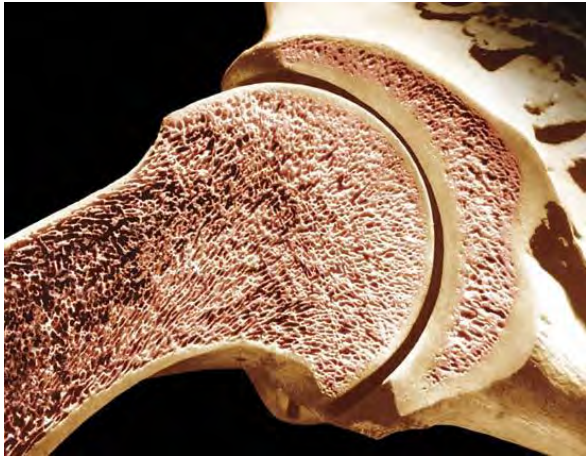
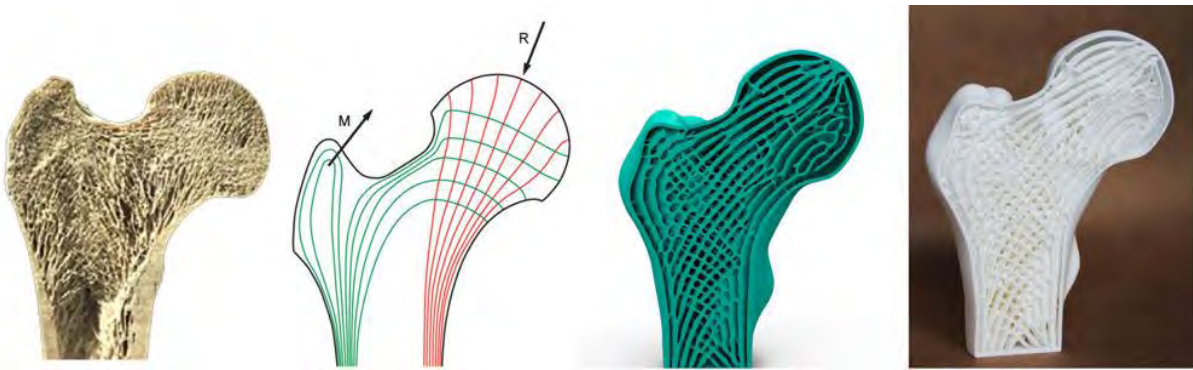
- [CROCKETT 2016] Michael Crockett, *3D Printing & Internal Geometry*, March 15, 2016, <http://blog.michael-crockett.com/html/InternalGeometry.html>
- [WU 2016] Jun Wu, Niels Aage, Rüdiger Westermann, and Ole Sigmund, *Infill Optimization for Additive Manufacturing—Approaching Bone-like Porous Structures*, November 19, 2016, <https://arxiv.org/abs/1608.04366>

They use generative geometry algorithms that produce optimum internal structures to improve the functional requirements of the part, while reducing weight, material used, and also print time. In this approach:

- strength is optimized by placing more material along outer shells, with support becoming progressively become less dense towards interior,
- special attention is focused on load-bearing areas, considering load path axes for internal geometry optimization,
- print preparation receives part geometry and desired indication for relative or absolute strengths (load conditions or specific regions for increased stiffness).

In the end, these articles suggest that using biological patterns for inspiration when developing internal geometry patterns optimized for strength:





# Background Information

---

## A Note on Speed Settings

There are several quirks about the way S3D handles speeds that caused me a fair bit of confusion.

By default, S3D used mm/min. If you are coming from Cura, which uses mm/sec, you might consider going to the Tools → Options → Preference menu and changing **Speed Display Units** from **mm/min** to **mm/s**.

A larger and more pervasive issue is the setting of many of the speeds as a percentage of another speed.<sup>1</sup> The primary speed setting – **Default Print Speed** – is specified directly as mm/min or mm/sec. However, these other speeds are done as percentages – *but it is often unclear of what they are a percentage*.

In practice, I often calculate the actual speeds for a profile, using a calculator or a spreadsheet. This helps tremendously to get a handle on what is actually happening during the print.

This is a list of all percentage-based speed settings and what is (to the best of my knowledge) is the hierarchy of how they are derived. The top level of the hierarchy are all given as mm/min or mm/sec. Lower levels (more indented) depend on the higher levels in the hierarchy above them. For example, the speed when printing the first layer will be **Default Print Speed** × **First Layer Speed** (%).

- **Default Print Speed**

- **First Layer Speed** (%)
- **Outline Underspeed** (%)
  - **Allow speed reductions down to** \_\_\_\_ %.

This is fairly complex. S3D seems to calculate a speed that will cause the layer to print in at least as much time as the number of seconds specified for the **Adjust printing speed for layers below** \_\_\_\_ **seconds** parameter. The calculations considers **Default Print Speed** and, if the area being printed is an outline, also **Outline Underspeed**. However, This Allow speed reductions down to \_\_\_\_ % parameter puts a lower floor on how slow the extruder is allowed to move – as a percentage of **Default Print Speed**.

- **Solid Infill Underspeed** (%)
- **Support Structure Underspeed** (%)
- **Above Raft Speed** (%)
- **(Prime Pillar) Speed Multiplier** (%)
- **(Ooze Shield) Speed Multiplier** (%)
- **Bridging speed Multiplier** (%)
- **X/Y Axis Movement Speed**
- **Z Axis Movement Speed**
- **Retraction Speed**

---

<sup>1</sup> This discussion applies to the speed that the extruder moves. It does not apply to fan speeds. Although fan speeds are set as percentages, that is a percentage of the maximum fan speed of the device.



## Metes and Bounds

There are physical limitations that we have to consider in any given hardware setup. There may also be limitations imposed by the firmware. This section has several “metes and bounds” that I have come across for my own situation, and how they relate to the slicer settings:

### **Maximum Print Speed and Melt Volume**

**Print Area** is the cross-sectional area (mm<sup>2</sup>) of filament that is being extruded. We often simplify the calculation of Print Area using a simplified formula:

$$\text{Print Area} = \text{Extrusion Width} \times \text{Layer Height}$$

**Print Volume** is the volume (mm<sup>3</sup>) of filament that the extruder prints per unit time (sec):

$$\begin{aligned}\text{Print Volume} &= \text{Print Speed} \times \text{Print Area} \\ &= \text{Print Speed} \times \text{Extrusion Width} \times \text{Layer Height}\end{aligned}$$

Since the printer must melt the filament to print it, this metric is often called **Melt Volume**. Specific printers and extruder hardware have a limit on amount of material that can be heated, and this rating is called the **Maximum Melt Volume** of the extruder.

$$\text{Melt Volume} = \text{Print Speed} \times \text{Extrusion Width} \times \text{Layer Height}$$

So, the parameters **Extrusion Width**, **Layer Height**, and **Maximum Melt Volume** place a limitation on the print speed. For “coarse” prints using large extruder nozzles and layer heights, the print speed could be significantly limited.

For example, I have been told that the **Maximum Melt Volume** of the Pi3Mk3 is “just over 10 mm<sup>3</sup>/sec” (Prusa forum user PJR, 5/2/2018, <https://shop.prusa3d.com/forum/general-discussion-announcements-and-releases-f61/mk3-0-8mm-nozzle-t17563.html>). In a typical printing scenario, you set **Extrusion Width** to 0.4mm and **Layer Height** to 0.2mm. Converting the formula above to solve for **Maximum Print Speed**:

$$\begin{aligned}\text{Maximum Print Speed} &= \text{Maximum Melt Volume} / (\text{Extrusion Width} \times \text{Layer Height}) \\ &= 10 \text{ mm}^3/\text{sec} / (0.4 \text{ mm} \times 0.2 \text{ mm}) \\ &= 125 \text{ mm/sec}\end{aligned}$$

If you were printing with an over-sized nozzle, setting **Extrusion Width** to 0.8mm and **Layer Height** to 0.6mm, your **Maximum Print Speed** would be far more restricted:

$$\begin{aligned}\text{Maximum Print Speed} &= \text{Maximum Melt Volume} / (\text{Extrusion Width} \times \text{Layer Height}) \\ &= 10 \text{ mm}^3/\text{sec} / (0.8 \text{ mm} \times 0.6 \text{ mm}) \\ &= 20.8 \text{ mm/sec}\end{aligned}$$

## Brief Profile Display and SIDs

In situations where you want to display profile settings in a reasonably condensed way, I use this “**Brief Profile Display**” format. It uses the Short IDs (SIDs) to identify the settings. Here is an example Brief

Profile Display from a profile for the Prusa i3 Mk3 using PLA at High Detail that was provided by Simplify3D on 4/12/2018:

**Extruder:** E-List [PrimExtr], Index Tool 0: Noz 0.40, ExtMult 1.00, ExtWid Man 0.40, Ooze Control: YES Retr, RetrDist 1.00, ExRestart 0.00, RVertLift 0.00, RSpeed 2400, Yes Coast, CoastDist 0.20, Yes Wipe, WipeDist 2.00

**Layer:** L-Extr PrimExtr, LHt 0.10, TSolid 4, BSolid 4, Shells 2, Dir: InOut, No PISeq, No Vase, FHT: 150%, FWid 100%, FSpeed 50% (2400mm/m = 40mm/s), StartPts: FastPrint

**Additions:** Yes Skirt, Sk-Extr PrimExtr, SkLayers 2, SkOffset 4.00, SkOutlines 2, No Raft, R-Extr PrimExtr, Top 3, Base 2, ROffset 3.00, SepDist 0.14, TopInfill 100%, SpAbRaft 30%, No Pillar, P-Extr AllExtr, PPWidth 12.00, PPLoc North-West, PPSpMult 100%, No Ooze O-Extr AllExtr, OOffset 2.00, OPerims 1, OShape Waterfall, OAngle 30°, OSpMult 100%

**Infill:** I-Extr PrimExtr, IntPat Rect, ExtPat Rect, Infill 30%, OutOvr 20%, InWid 100%, MinLen 5.00, Combine 1, No IncSolid, DiaphEvery 20, IntAng: 45 / -45, ExtAng: 45 / -45

**Support:** No GenSupp, Sp-Extr PrimExtr, Infill 40%, ExInflDist 0.00, BaseLayers 0, CombEvery 1, DenseSupp D-Extr PrimExtr, DenseLayers 0, DenseInfill 70%, APTType Normal, APRes 4.00, APAngle 45°, HSep 0.30, UpLay 1, LowLay 1, SuppAng: 0

**Temp:** T-List [PrimExtr] T0, TType Extruder, No Layer, No Loop, Yes WaitStab, SetP 1:200°C, T-List [HBed] T0, TType HBPlat, No Layer, No Loop, Yes WaitStab, SetP 1:60°C

**Cooling:** FanSpeed 1:0 / 2:100, No Blip, No Incr, ITime 45sec, MxFSp 100%, No Bridge, BrSpOvr 100%

**G-Code:** Yes 5D, No RelDist, Yes AllowZ, No Indep, No M101, Yes Sticky, No Offsets, GOffsets X:0.00 / Y:0.00 / Z:0.00, Yes UpMDef, MType Cart, Build X:250 / Y:210 / Z:210, Orig X:0 / Y:0 / Z:0, Home X:Min / Y:Min / Z:Min, Flip No X, Yes Y, No Z, THeadOffsets [Tool 0] X:0 / Y:0, Yes UpdFirm, FType RepRap, GPX: Replicator 2, Baud 115200

**Scripts:** Starting Script and Ending Script (see below), others blank  
ExpFmt: Standard G-Code, No AddCeleb, AddTermCmd: (none)

**Speeds:** SpDefault 4800 mm/min (= 80mm/sec), SpPerim 50% (2400mm/m = 40mm/s), SpSolidIn 80% (3840mm/m = 64mm/s), SpSupp 80% (3840mm/m = 64mm/s), SpXY 12000 (= 200mm/s), SpZ 1000 (= 16.67mm/s), Yes AdjBelow, AdjBelowSec 15, AdjDown 20% (SpDefault: 960mm/m = 16mm/s, SpPerim: 480mm/m = 8mm/s)

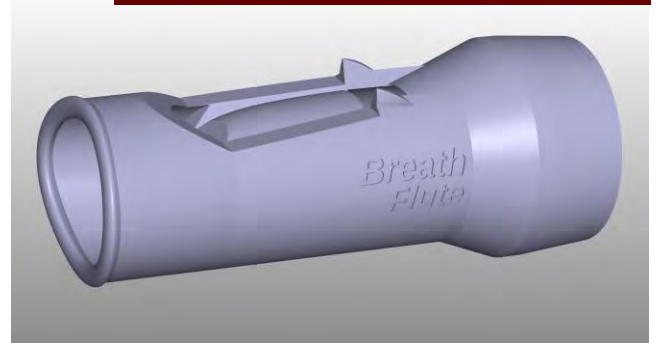
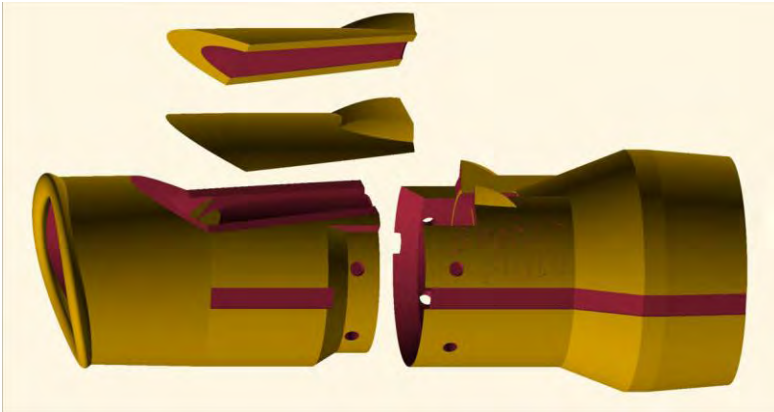
**Other:** Area 50, ExInflat 0.00, BrExMult 100%, BrSpMult 100%, No FixedAngle 0°, No BrPerim, HComp 0.00, F-List [Tool 0] FilDiam 1.75, FilPrice 46.00, FilDen 1.25, ChgRetDist 12, ChResDist -0.50, ChRetSp 600

**Advanced:** No Start 0.00, No Stop 0.00, ExThinType PerimOnly, InThinType GapFill, Overlap 25% MinExLen 1.00, MinPWide 50%, MxPWide 200%, EndExtDist 0.20,

Yes Open, Yes ForceRet, No MinTRetr 3.00, Yes RetWipe, Yes WipeOuter,  
No AvoidCross, MaxDetour 3.0, NonManSeg Heal, No Merge

### Impetus

This work was done as part of my main 3D printing effort – to develop a new style of flute called the **Breath Flute**. The flute consists of a 3D-printed headjoint and four feet of 1¼" tube. The OpenSCAD model currently runs about 7,000 lines of code (with comments). Here are some images from the current state of development:



## Extruder Tab

---

The parameters on this Tab can be set for each Extruder in the Extruder List.

### Extruder List

Choice <extruder> [E-List]

List of available extruders. Double-click to edit extruder name.

### Overview

#### Extruder Toolhead Index

Integer <toolheadNumber> [Index]

The index of the selected extruder toolhead. This is required for firmware toolchange commands.

This is the Identifier you associated with your extruder head that is decided in your firmware. The general convention is your first tool will be T0, if you have multiple extruders general convention also is that T0 will be the right extruder and T1 will be the left extruder.

#### Nozzle Diameter

mm <diameter> [Noz]

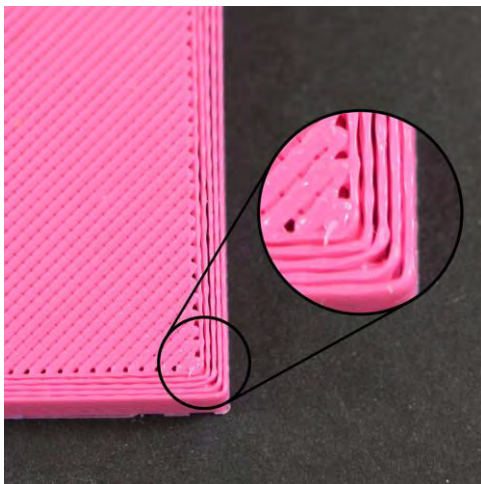
The diameter of your extruder nozzle.

This is the diameter of the tip of your nozzle where plastic is extruded from. Traditional values range from 0.3 mm all the way up to 1 mm for some of the more customized printers. A smaller nozzle is great for thin walls, for printing small miniatures and other things that require fine precise details. Large 1 mm nozzles are great for rapid prototyping, creating large parts quickly and efficiently by pushing out lots of plastic.

#### Extrusion Multiplier

Factor <extrusionMultiplier> [ExtMult]

Multiplier for all extrusion movements (allows simple flowrate tweaking).



The extrusion multiplier will multiply the amount of filament extruded for your entire print. This includes skirts, rafts, supports, perimeter and infill extrudes. The default values in the software are 0.90 for PLA and 1.0 for ABS. If you notice that your **Top solid layers** are not as filled-in as you'd like them to be, I would recommend increasing the number of **top solid layers**. If you still think there's an issue, then this is the setting I'd recommend changing.

This parameter is often used to correct under-extrusion problems. See the **Not Extruding Enough Plastic** section of the *Simplify3D Print Quality Troubleshooting Guide* at

<https://www.simplify3d.com/support/print-quality-troubleshooting/>.

## Extruder Calibration

Some of the more experienced makers take pains to calibrate their extruders and set the Extrusion Multiplier accordingly. The calibration can be done by material (e.g. PLA vs. ABS), brand of material (e.g. MakerGeeks PLA vs MatterHackers PLA), or even down to the specific roll of filament being used – it depends on your goals and level of precision.

Thanks to Matt Harrison for providing an excellent article on Extruder Calibration, dated April 19, 2017, on his page at <https://mattshub.com/2017/04/19/extruder-calibration/>. This text is taken from that page:

Having your extruder properly calibrated is essential for perfecting your print quality. Often when you go to calibrate it for the first time, you'll find that your calibration is actually a long way off from what it is supposed to be. You've been printing with it set incorrectly all along!

There are two parts to calibrating your extruder:

- tuning your extruder steps/mm value in your firmware,
- and tuning your **Extrusion Width / Extrusion Multiplier**.

It is important to do it in this order. First, we want to sort out how much plastic is being fed in to the hot end (regardless of any values like **Extrusion Width**, **Extrusion Multiplier**, **Filament Diameter**, etc.), and then move on to how the printer is behaving in terms of the plastic actually being pushed out and laid down.

### **Calibrating extruder steps/mm**

To calibrate the steps/mm value, we tell the printer to extrude 100mm of plastic. Then we measure how much plastic was actually extruded.

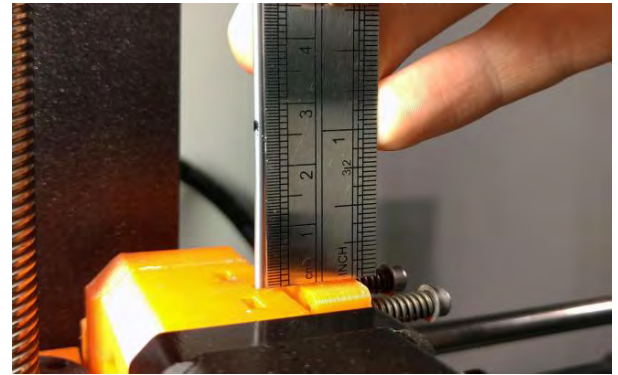
#### **Measuring**

To do this, first measure out 120mm of filament from where it enters your extruder and mark it with a pen or marker. This will be the reference point. Then, connect your computer to your printer and open up a program that allows you to send individual G-Code commands to it (like Pronterface, Simplify 3D, Repetier Host, or Octoprint web interface). Heat up your extruder to your regular printing temperature. Then we want to make sure that the extruder is in relative mode, so send the command **M83** using the textbox in the bottom right hand corner if you're using Pronterface. In Simplify 3D you'll want to open the machine control panel window and go to the communication tab. All the commands that I'll be referring to later in this post should all be entered in this way.

Then, tell the printer to extrude 100mm of plastic with the command **G1 E100 F100**. This will take 60 seconds until it's finished extruding. Why are we extruding it so slow? Because extruding slowly in this step ensures that the resistance of the plastic further down when it is being pushed into the nozzle, does not affect how much is fed in by the stepper motor. It also helps to take the temperature out of the equation, again by reducing the effects of pressure in the nozzle to virtually nothing.



Once the machine has finished extruding the 100mm, switch off the extruder heater and measure the distance between the point that you marked before we started, and where the plastic enters the extruder (the same point from which you measured the initial 120mm). If this is exactly 20mm, congratulations! Your extruder steps/mm are perfectly calibrated. If it is more or less than 20mm, then it means that your printer is over or under extruding. Fortunately, the way to solve this is fairly easy.



## Tuning

In order to calculate what our new steps/mm value will be, we need to know the existing steps/mm value, and the under/over extrusion rate. To get the existing steps/mm value, send the command **M503**. This will print out all the current values saved in your printer's EEPROM (storage that persists when it is powered off), including all your axis steps/mm values. We're only interested in the E value, highlighted in the picture at the right.

```
SENDING:M503
echo:SD card ok
echo:Steps per unit:
echo: M92 X100.00 Y100.00 Z400.00 E161.30
echo:Maximum feedrates (mm/s):
echo: M203 X500.00 Y500.00 Z12.00 E25.00
echo:Maximum Acceleration (mm/s2):
echo: M201 X9000 Y9000 Z500 E10000
```

Now, the math: We need to work out how much plastic your printer actually extruded, which can be calculated from: **120 mm – left over filament distance** (measured in the previous step). For example, for me this was **120 – 26 = 94**.

So, my printer was actually only extruding 94 mm when I asked it to extrude 100 mm – that's a 6% under extrusion! So, to calculate the new, correct extruder steps/mm value, we do the following:

Original extruder steps/mm value × 100 mm = total steps taken:

$$161.3 \times 100 = 16130$$

From that, we can extrapolate that calibrated extruder steps/mm value (y) × actual extruded distance = total steps taken:

$$y \times 94 = 16130$$

Therefore, calibrated extruder steps/mm value (y) = total steps taken / actual extruded distance:

$$y = 16130 / 94 \\ = 171.6$$

```
>>>M92 E171.6
SENDING:M92 E171.6
>>>M500
SENDING:M500
echo:Settings Stored
```

This is our new calibrated extruder steps/mm value! To enter and save it to your printer use the commands **M92 E###.#** (replace the hashes with your calibrated extruder steps/mm value) and then **M500** to save it. The Prusa i3 MK2 must be running a recent firmware version to enable saving to EEPROM.

To make sure that this has all worked out as intended, turn your printer off and on or reset it and then send the command **M503** again to check if your new extruder steps/mm value is shown. To do a final test to make sure it's correctly calibrated, measure out another 120 mm of filament, mark it, and then extrude 100 mm. You should have exactly 20 mm left over. If not, recalibrate using the steps above and your new steps/mm value as the original.

### Calibrating extrusion multiplier

Now that we know the right amount of filament is being fed into the hotend by the extruder mechanism, we need to make sure that the filament being extruded is the same amount as what our slicer thinks it is. We can do this by printing a single perimeter cube and checking if the widths of the walls are the same as our **Extrusion Width**.



To begin, accurately measure your **filament diameter**, preferably with some digital calipers. Enter this into your slicing software. Then, make sure that your **Extrusion Multiplier** is set to 1. Check what your **Extrusion Width** is and remember it – that's what we'll be comparing to later on.

Load a 25 mm cube (<https://www.thingiverse.com/thing:2267549>) into your slicer and set the infill to 0%, perimeters to 1, and top solid layers to 0. You'll also want to print it at a fine resolution – I chose 0.15 mm and it did make a small (0.02 mm) difference in the wall thickness as opposed to 0.3 mm. Print it out and then use digital calipers to measure the thickness of the walls. Your aim is to get this to be the same as your **Extrusion Width** set in your slicer. Adjust your new extrusion multiplier to:



$$\frac{(\text{extrusion width} / \text{measured wall thickness}) \times \text{extrusion multiplier}}{\text{extrusion multiplier}}$$



For example, since my walls first came out as 0.5 mm even though I set my **Extrusion Width** to 0.45 mm, my extrusion multiplier would need to be changed to  $(0.45/0.5) \times 1 = 0.9$ . Enter this into your

slicer and print the model again and re-test. If your measurements and your calculations were correct it should only take one adjustment, but sometimes it might take a few tries to get it right. This extrusion multiplier value should be calibrated and set on a per material basis due to the different flow characteristics of different materials when extruded (e.g. viscosity).

## Finalisation

Your extruder should now be fully calibrated! Enjoy printing knowing that you have everything dialed in perfectly. If you're finding that your prints look like they're over extruding or under extruding, first check your **Filament Diameter** and make sure it's the exact same as what you have in your slicer. If it is and you're still having minor issues, don't be afraid to change your extrusion multiplier up or down a few %. The tests above are all prone to errors (most likely in measurements), and what's most important is how the prints actually come out. So, if you need to change it a little bit then that's fine!

## Extrusion Width

0 / 1 <autoWidth> [ExtWid]  
mm <width> [ExtWid]

**Auto:** Automatically choose an extrusion width to maintain similar flow rates across different layer heights.

**Manual:** Define a fixed extrusion width that will remain constant for different layer heights.

The slicing engine doesn't use your nozzle diameter, but instead uses your **Extrusion Width** setting. For most cases, keeping this setting on **Auto** ( $1.2 \times$  your nozzle diameter) is the best route. However, for certain cases where you have extremely thin nozzles, you may find it's best to go to **Manual** and play with lowering the **Extrusion Width** setting.

Prusa Forum user PJR indicated that “*generally extrusion width should be from 1 to 1.25 times the nozzle diameter*” (May 2, 2018, <https://shop.prusa3d.com/forum/general-discussion-announcements-and-releases-f61/mk3-0-8mm-nozzle-t17563.html>)

## Ooze Control

### Retraction

Yes/No <useRetract> [Retr]

Reverse filament direction at the end of a loop to help prevent stringing.

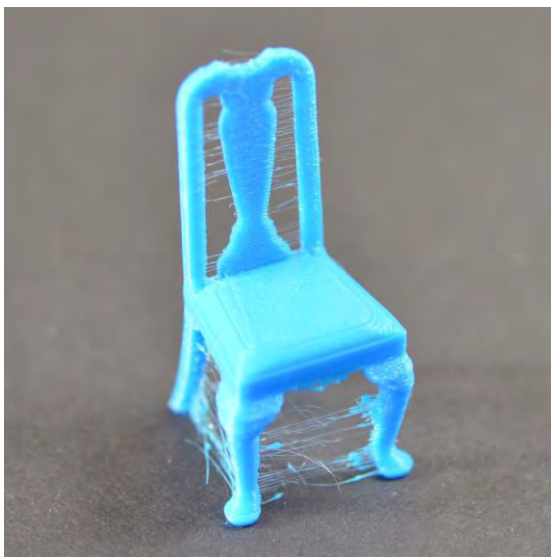
A flag that indicates whether to perform retraction.

### Retraction Distance

mm <retractionDistance> [RetrDist]

How much plastic to pull back into nozzle (in terms of raw filament).

How much filament will be retracted when the software does a retract. For standard Direct Drive, usually 0.5 to 2.5 mm will work well. For Bowden extruders, 5.0 to 8.5 mm seems to work well.



This **Retraction Distance** parameter and the **Retraction Speed** parameter (below) are sometimes implicated in situations where the print comes off the bed with lots of “hair”. See the “Stringing and Oozing” section of the *Simplify3D Print Quality Troubleshooting Guide* at <https://www.simplify3d.com/support/print-quality-troubleshooting/>.

### Extra Restart Distance

mm <extraRestartDistance> [ExRestart]

Extra extrusion distance on top of initial retraction amount. Negative values are allowed (in terms of raw filament).

When the software does a prime (after retracting), it will prime the same amount as retracted. However, you can modify this by placing an Extra Restart Distance, if you notice that your prints have minor blobs at the starting points, you may find that placing a small negative Extra Restart Distance can help, such as -0.2 mm.

## Retraction Vertical Lift

mm <retractionZLift> [RVertLift]

Nozzle will lift from the surface of the part during a retraction move.

The nozzle will move up in the Z-axis whenever you do a retract. This is particularly helpful with Delta printers and prints that have a lot of islands, since the delta printers can be prone to running into parts when doing rapids.

## Retraction Speed

mm/min <retractionSpeed> [RSpeed]

Extruder speed for the retraction movements. Typically use the highest speed your extruder can support.

The speed at which the filament will be retracted/primed. I think 1800 mm/min or 30 mm/sec is about as slow as I'd want to go for standard retracts, that going quicker will help for print quality in most cases.

## Coast at End

0 / 1 <useCoasting> [Coast]

Turn off the extruder a short distance before the end of a loop to relieve pressure in nozzle and prevent blobs

## Coasting Distance

mm <coastingDistance> [CoastDist]

Distance that nozzle will stop extruding prior to the end of a loop.

Setting a coasting value can be good if you want to empty out your nozzle before doing a retract. Let's say you're printing a single line that is 100 mm long. If you set a coasting distance of 5 mm, the extruder will be pushing out plastic for the first 95 mm of traveling, but then stop extruding and while it will move over the rest of the line, it will not actually extrude anything for the last 5 mm. It will instead depend on the filaments momentum, and gravity to let the rest of the filament ooze out and fill in the region for the last 5 mm of the line.



## Wipe Nozzle

Yes/No <useWipe> [Wipe]

Wipe the nozzle at the end of a loop.

A flag that indicates whether to perform the Wipe Nozzle processing.

## Wipe Distance

mm <wipeDistance> [WipeDist]

Total distance for the wipe movement.

Wiping happens after you've printed your outer most outline (Advanced tab, there's an option that's usually enabled for this). When you print your outer most outline, when it's time to do a retract, there's a good chance that not all of the filament in the extruder nozzle head is going to rise up, that a blob of molten liquid plastic is going to be at the tip of the nozzle still. For this reason, instead of rapidly moving to the next spot right away, you can use the Wipe function, which will wipe over the perimeter and let that filament ooze out, similar to Coasting. However, Coasting is not extruding over areas that need filament (you risk under-extrusion voids if coasting value is too high), Wiping is extruding over areas that have already been printed on (much lower risk of part quality being negatively affected).

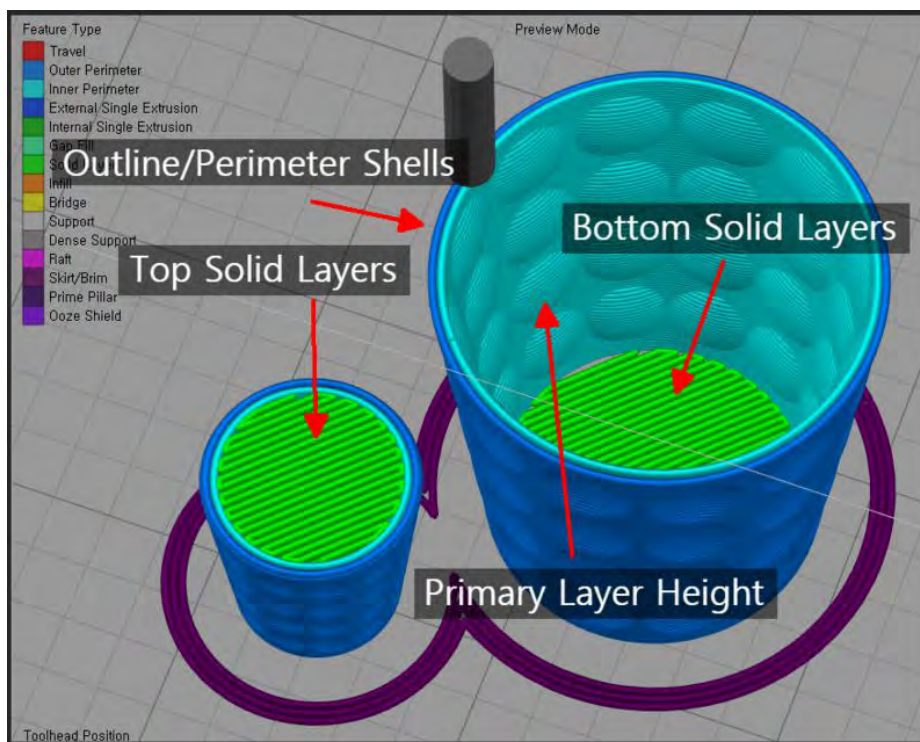
## Ooze Control Summary

There's a lot of terms in there, but here's the order they go in for an outer perimeter:

- 1) Coasting starts \_\_ mm before outer most perimeter is done
- 2) Finishing printing outer most perimeter
- 3) Retract \_\_ mm at \_\_\_\_ Retraction Speed
- 4) Wipe \_\_ mm over the outer most perimeter
- 5) Vertical lift up \_\_ mm
- 6) Rapid move at Rapid Move Speed (the Other Tab)
- 7) Vertical lift down \_\_ mm
- 8) Prime filament. Amount primed is Retraction Distance + Extra Restart Distance

## Layer Tab

Here is an image from the Naver blog posts that shows some of the important parameters on this tab:



### Layer Settings

#### Primary Extruder

Index <primaryExtruder> [L-Extr]

Choose extruder for outline and exterior surfaces of your model.

Which extruder you'd like to have the Perimeters printed with.

#### Primary Layer Height

mm <layerHeight> [LHt]

Thickness of each printed outline layer.

How thick you want each layer to be on the Z-axis. Smaller means finer resolution and better print quality, but also means many more layers you'll have to print, which can dramatically increase print time.

This parameter plays a role for the strength of the printed part. See the **Strength, Speed, Cost, and Quality** section. A **Primary Layer Height** of 0.2–0.4 mm is somewhat stronger than 0.10–0.15 mm.

Most profiles use 0.3 mm for “Fast Print”, 0.2 mm for “Medium”, and 0.1 mm for “High Quality”.

Prusa Forum user PJR indicated that “generally ... layer height [is] a maximum of 80% nozzle diameter” (May 2, 2018, <https://shop.prusa3d.com/forum/general-discussion-announcements-and-releases-f61/mk3-0-8mm-nozzle-t17563.html>).

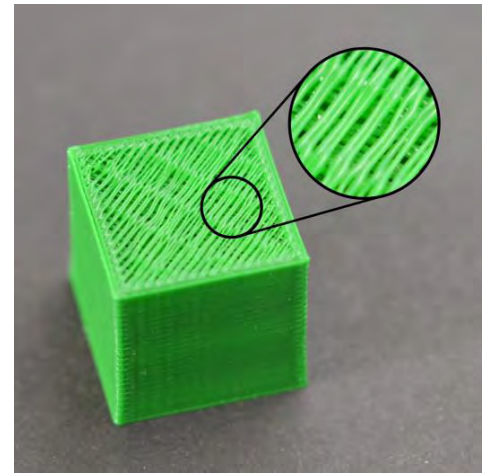
## Top Solid Layers

Count <topSolidLayers> [TSolid]

Number of solid layers to require at the top of the part.

The number of 100% infill layers that will be placed at the top of your part. If you notice that there's some incomplete filling your top layers, I would recommend raising this value from 3 to 5 and increasing the extrusion multiplier slightly under the Extruder tab if needed.

This parameter is often implicated in situations where there are holes or gaps in the top layer. See the “Holes and Gaps in the Top Layers” section of the *Simplify3D Print Quality Troubleshooting Guide* at <https://www.simplify3d.com/support/print-quality-troubleshooting/>.



## Bottom Solid Layers

Count <bottomSolidLayers> [BSolid]

Number of solid layers to require at the bottom of the part.

The number of 100% infill layers that will be placed at the bottom of your part.

## Outline/Perimeter Shells

Count <perimeterOutlines> [Shells]

The number of shells to use for the exterior skin of the part.

Outline shells will trace the outline of your part and extrude at your extrusion thickness. The printer will print the outline shells, then print Infill afterwards.

This parameter is important for the strength of the printed part. See the **Strength, Speed, Cost, and Quality** section. Based on that testing, the optimal number of **Outline/Perimeter Shells** for strength appears to be 3.

## Outline Direction

Choice <printPerimetersInsideOut> [Dir]

Print inner-most / outer-most perimeter first.

**Inside-Out:** It will print your perimeter shells from the inner shell to the outer most shell. This is very beneficial when printing overhangs, as the print is branching out in the X-Y direction for each layer.

**Outside-In:** It will print your perimeter shells from the outer most shell to the inner most shell. This is better for surface quality finish usually. For instance, if printing a cube, this may be the better route.

The value of the XML parameter <printPerimetersInsideOut> in the .fff file will be 1 for **Inside-Out** and 0 for **Outside-In**. The value for the **Dir** parameter in a Brief Profile Display is **InOut** for **Inside-Out** and **OutIn** for **Outside-In**.

## Print islands sequentially without optimization

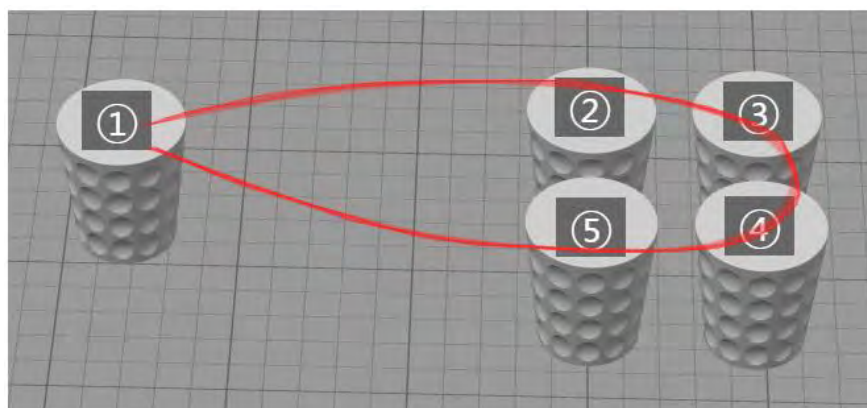
Yes/No <sequentialIslands> [PISeq]

This is typically disabled to optimize the travel time between layers for faster prints and minimal oozing. May need to be enabled for small parts with multiple islands to prevent overheating.

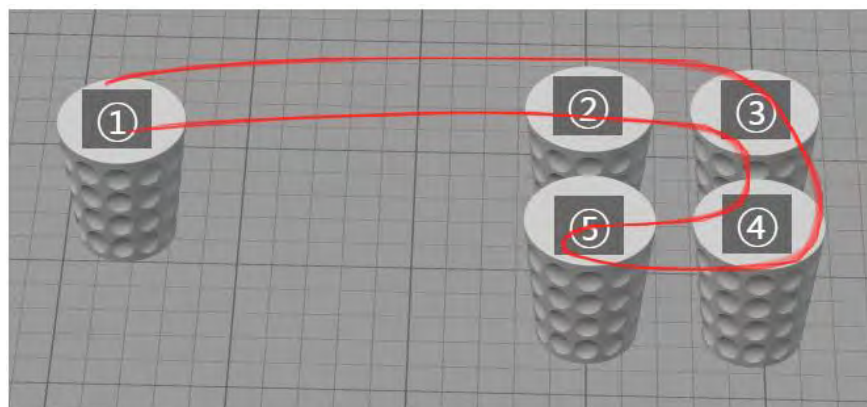
Let's say you are printing the part at the right. If you went from pillar to pillar in the most efficient order, you may get to the next layer so quickly there may be issues with heat-build up and the previous layer not being solidified. Therefore, you can turn off optimization, and then it will print the pillars in a random-order therefore helping you prevent from heat-build up.



These images from the Naver blog posts show how the print sequence is affected by this parameter:



When **Print islands sequentially without optimization** is checked, the output will be printed in the order: 1-2-3-4-5-1-2-3-4-5-1-2-3-4-5. This has the advantage of giving the material time to harden, but it has the disadvantage of taking more print time.



When **Print islands sequentially without optimization** is **not** checked, the output will be printed in the order: 1-2-3-4-5-5-4-3-2-1-1-2-3-4-5. The time is shortened because when 5 and 1 are finished, the next layer is output immediately. However, if the cooling becomes weak, the output will start immediately, and the output quality may drop.

## Single outline corkscrew printing mode (vase mode)

Yes/No <spiralVaseMode> [Vase]

Gradually increments the Z-axis to avoid any layer-change seams. Especially useful for vases, bracelets, and other hollow objects. (Note: Using this option will force 0% infill with only a single perimeter).

The extruder will print with one outline/perimeter shell and won't make any retracts. This means that it will slowly move up in the Z as it prints, imagine spiraling upwards, instead of printing a static layer, then moving upwards to do another layer. Traditionally with vases the best settings I've found are Zero top solid layers, 3 Bottom Solid Layers, and under the Advanced tab enable ***Merge all outlines into a single solid model.***



## **First Layer Settings**

These are applied to where the Bed touches your model. If you have a raft, that means the First Layer Settings will apply to your raft.

### **First Layer Height**

% <firstLayerHeightPercentage> [FHT]

First Layer Height is commonly modified to improve adhesion of first layer and account for uneven surfaces.

This % will take a % of your **Primary Layer Height**. If you are printing with a small **Primary Layer Height** such as 0.1 mm I would recommend a **First Layer Height** setting of 250%, to get about a resulting 0.25 mm first layer height.

If your **First Layer Height** is below 100%, the extrusion amount will remain the same, only the Z-will change, but if you increase **First Layer Height** above 100%, the extrusion amount will scale accordingly.

### **First Layer Width**

% <firstLayerWidthPercentage> [FWid]

The width of the first layer extrusion can be increased to help with adhesion.

The extrusion width of your first layer, you may find that your first layer sticks better with a thicker extrusion (100%+). I don't have too much experience with this, but I think 125% or 150% would be good starting points.

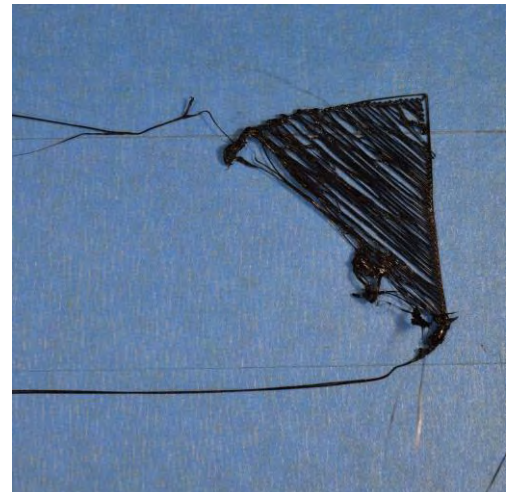
### **First Layer Speed**

% <firstLayerUnderspeed> [FSpeed]

Slower first layer speeds help improve bed adhesion.

Slows down the First Layer Speed to a percentage of your **Default Printing Speed** (Speeds Tab).

This parameter is often implicated in situations where the first layer is not sticking to the print bed. See the “Print not Sticking to the Bed” section of the *Simplify3D Print Quality Troubleshooting Guide* at <https://www.simplify3d.com/support/print-quality-troubleshooting/>.



## **Start Points**

Certain printers find that there are very small voids at the starting points. This can create a seam on the print. Start points can options can help for both controlling where these voids (the seam) goes, but also help for print time.

I am including several images from the Naver blog that highlight how the start points are chosen.

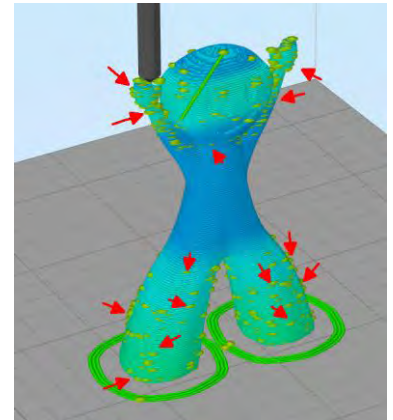
Choice  
<startPointOption>  
[StartPts]

### ***Use random start points for all perimeters***

Layer start points are randomly distributed all over the model.

Randomizes the starting points. See the top image at the right.

The value of the XML parameter <startPointOption> in the .fff file will be set to 0 for this option. The value for the **StartPts** parameter in a Brief Profile Display is **RandomStart**.

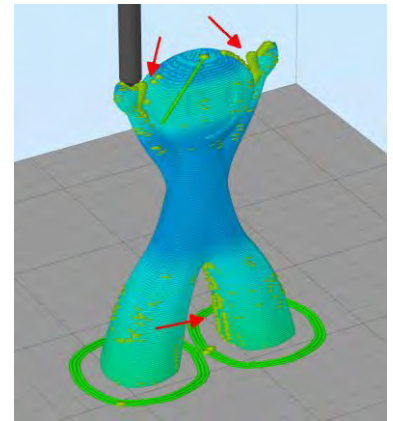


### ***Optimize start points for fastest printing speed***

Layer start points are chosen to optimize printing speed and minimize the travel distance between islands.

Optimizes starting points by finding the closest location for the next starting point. When generating G-Code you can see the travel moves, traditionally this option really minimizes the amount of red lines that signify travel moves you will see on the G-Code previewer.

The value of the XML parameter <startPointOption> in the .fff file will be set to 1 for this option. The value for the **StartPts** parameter in a Brief Profile Display is **FastPrint**.



## Choose start points closest to specific location

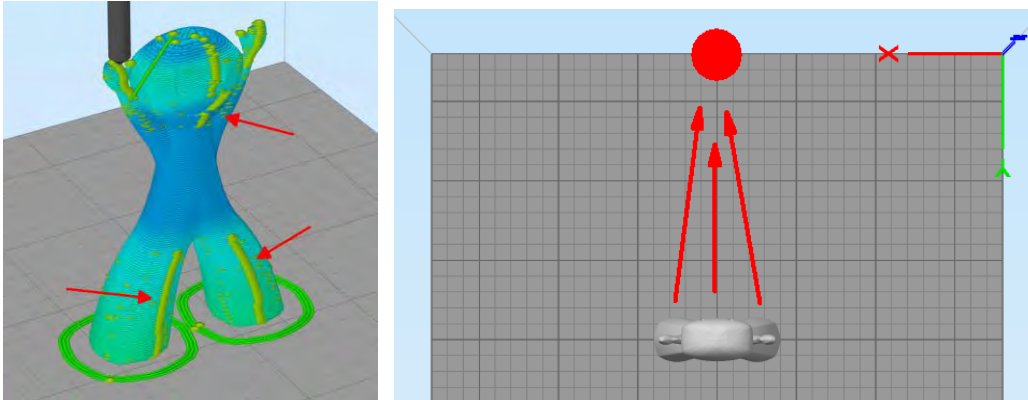
mm <startPointOriginX> <startPointOriginY>

All layer start points are aligned as close as possible to the specified X–Y location.

X: X-coordinate where all layer start points are aligned.

Y: Y-coordinate where all layer start points are aligned.

This can be useful if you want to line-up your starting points on a certain part of your print, so the seam isn't obvious or is hidden.

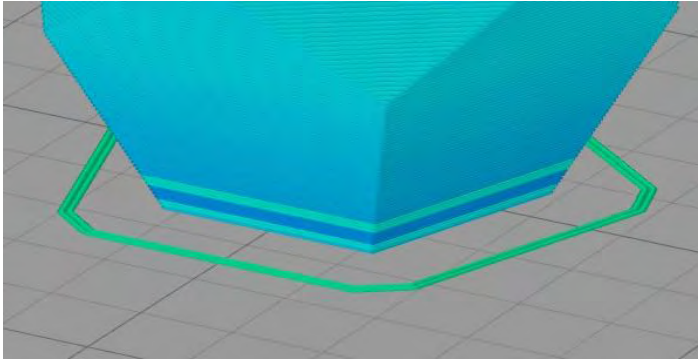


The value of the XML parameter **<startPointOption>** in the .fff file will be set to 2 for this option. The value for the **StartPts** parameter in a Brief Profile Display is **Choose**.

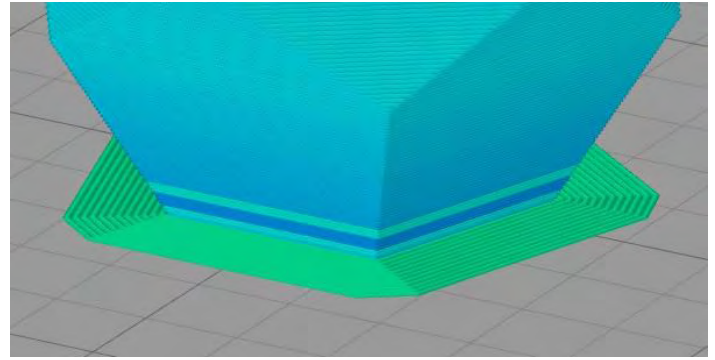
## Additions Tab

### Skirt Settings

Skirts are really useful for priming your extruder prior to printing. I generally print with a skirt offset of 4 mm and 2 outlines. However, changing the offset to 0 mm, offers the ability to create a “brim” that will connect to your part and help create surface adhesion. For more information on skirts, brims, rafts, this tutorial is very helpful: <http://www.simplify3d.com/support/articles/rafts-skirts-and-brims/>



Skirt



Brim

### Use Skirt/Brim

On/Off <useSkirt> [Skirt]

This option can help prime the extruder prior to beginning the print and also helps anchor the edges of the model to the build platform to avoid warping.

### Skirt Extruder

Choice <skirtExtruder> [S-Extr]

The extruder that will be used for the skirt / brim. If “All Extruders” is selected, each extruder will print one outline.

Selects the extruder to be used for printing your Skirt

### Skirt Layers

Count <skirtLayers> [SkLayers]

Number of vertical layers to include skirt.

How tall your skirt will be. If you want the skirt to be as tall as your part, you can just place 99999 in this field.

### Skirt Offset from Part

mm <skirtOffset> [SkOffset]

Skirt offset from the part outline. Offset of zero will just touch the exterior of the part (brim).

How far away your skirt will be from your part.

## Skirt Outlines

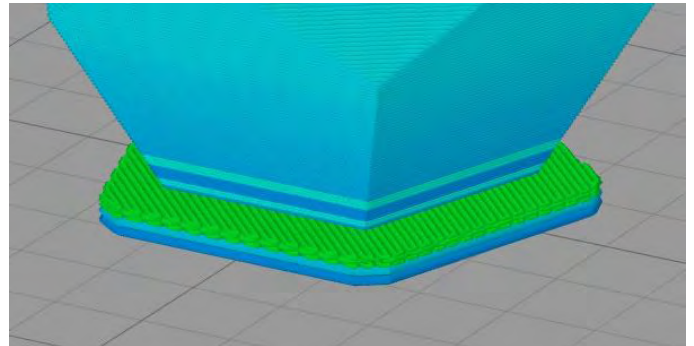
Count <skirtOutlines> [SkOutlines]

Number of skirt outline perimeters.

# of perimeter outlines for your skirt.

### **Raft Settings**

A raft offers a base understructure that will contour to the outline of your print, increasing the surface area to the bed. This in-turn will offer more adhesion to the print bed. However, because rafts take a long time to print and use a lot of material, it is only recommended to use rafts for prints that have a low surface area with the print bed or that generally don't have a good base will often benefit from a raft. Also, whether due to the design of the printer or calibration, those users with beds that are slightly uneven have often found that rafts can provide a good foundation.



When the print is completed, remove the entire raft and model from the build platform. You can then grab the raft and begin to peel it away from the part, leaving a high-quality surface finish on the bottom of your print. Usually, this can be done by hand, but for extra delicate parts, you may want to use a thin spatula or tweezers to help with the removal. If you find you are having a difficult time removing the raft from the part, you can try increasing the [Separation Distance](#).

### **Use Raft**

On/Off <useRaft> [Raft]

Creates a raft below the part that can help with adhesion and provide a clean level surface to begin the print.

### **Raft Extruder**

Choice <raftExtruder> [R-Extr]

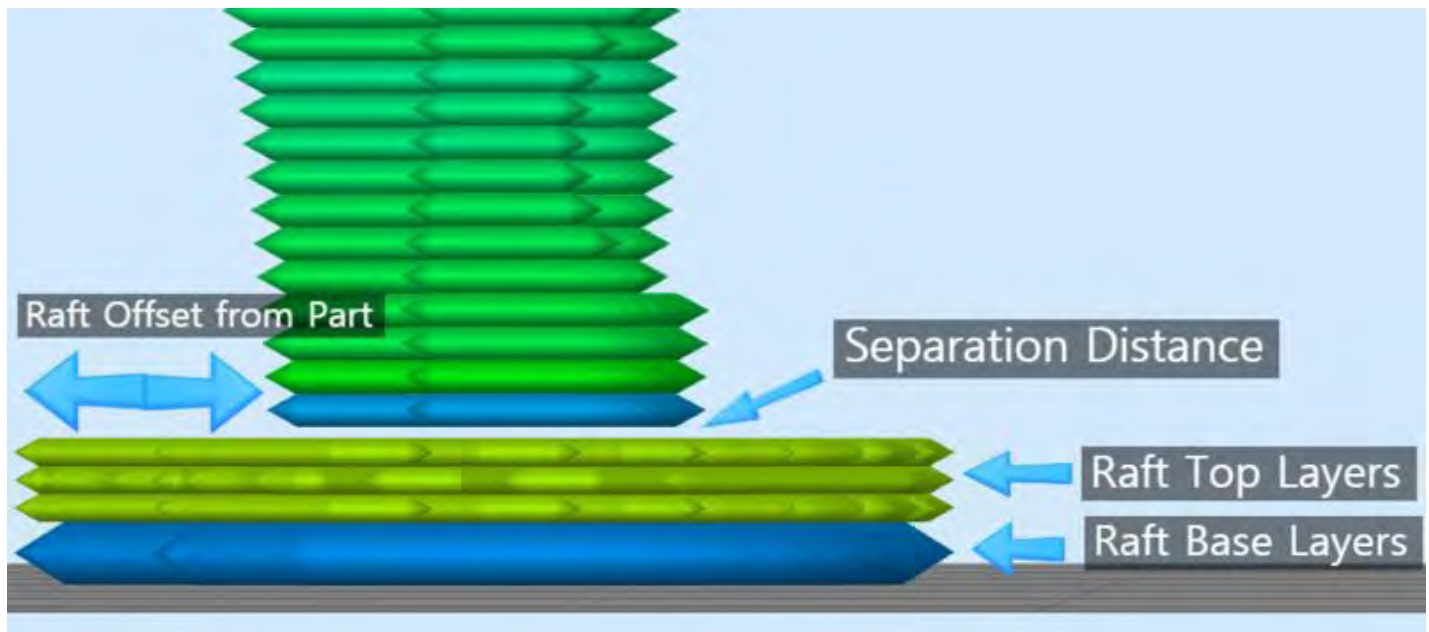
The extruder that will be used for the raft.

Selects the extruder to be used for printing your Raft.



## Raft Settings

I think this image from the Naver blog posts captures the important aspects of the next four parameters:



### **Raft Top Layers**

Count <raftTopLayers> [Top]

Number of interface layers at the top of the raft.

Your model will be printed on top of these layers, so you usually want at least 2–3 layers to ensure a smooth surface.

### **Raft Base Layers**

Count <raftBaseLayers> [Base]

Number of thick base layers at the bottom of the raft.

These layers are printed slow and thick to ensure a strong bond to the build platform.

Earlier versions of S3D used a checkbox setting called **Disable Base Layers**. This simply disabled the thicker layers at the bottom of the raft, which can be helpful if your rafts stick to the bed too much. In the current version of S3D, I believe that setting this **Raft Base Layers** to zero is the same as checking **Disable Base Layers** in earlier versions.

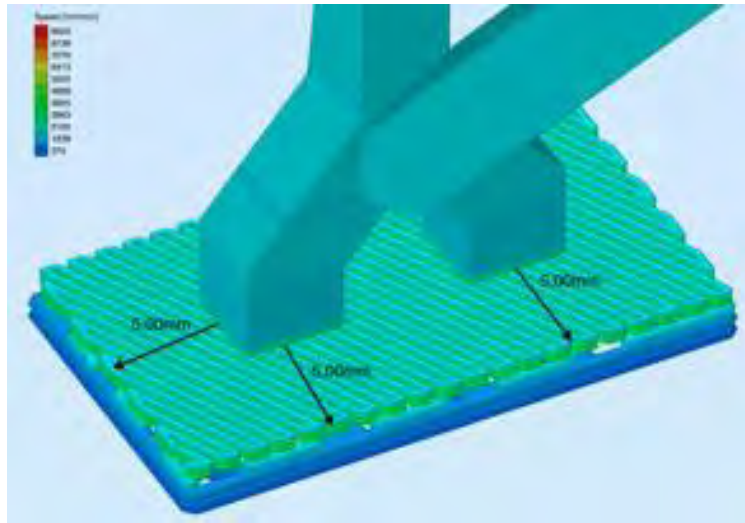
Note that, as of S3D v3.1.0, there was a single parameter in the .fff file called <raftLayers>.

### **Raft Offset from Part**

mm <raftOffset> [ROffset]

Raft offset from the exterior of the part outline.

How wide the raft will extend beyond the edges of your part.



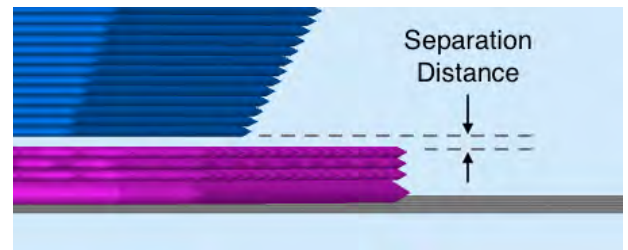
## Separation Distance

mm <raftSeparationDistance> [SepDist]

Separation distance between the last layer of the raft and the bottom layer of the part. Higher values will help the raft separate easier.

How much extra distance on the Z-axis from the raft to the bottom of your part.

This is an important setting that defines the height of the air gap between the raft and your part. Typically, having a gap of at least 0.1mm will help keep the part connected to the raft, while still allowing for easy separation once the print is complete.



## Raft Top Infill

% <raftTopInfill> [TopInfill]

Infill percentage for the top layers of the raft.

This will typically be set around 80%, set the top layer of your rafts infill. For rafts made from dissolvable filament, upping this to 100% may be helpful.

## Above Raft Speed

% <aboveRaftSpeedMultiplier> [SpAbRaft]

Modifies the print speed for the first layer of the part on top of the raft.

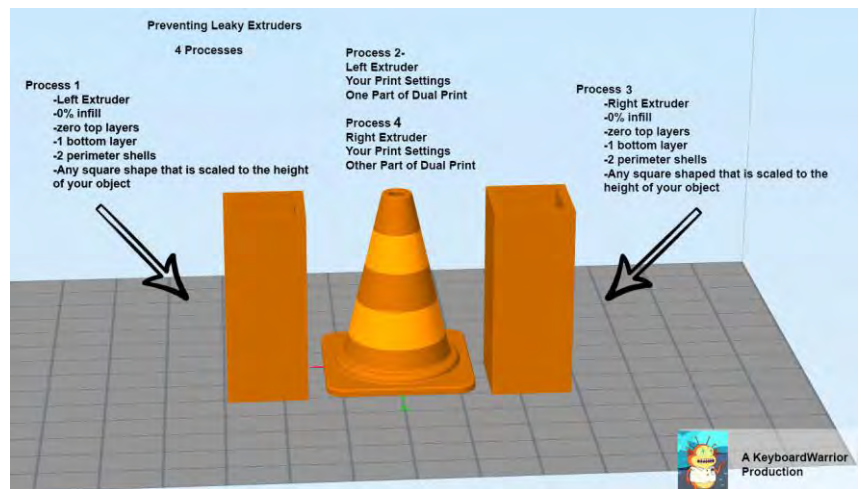
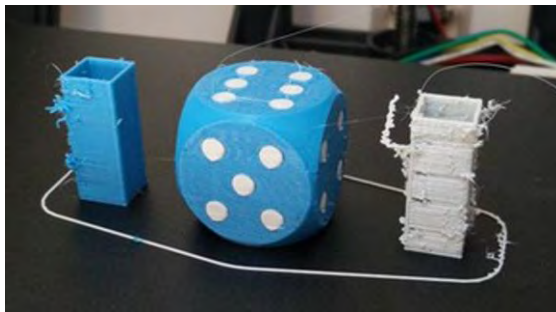
Use this setting to customize the speed of the first layer of your part that is printed on top of the raft surface. Using a slow speed for this layer will also help the part stay attached to the raft during the print.

## Prime Pillar Settings

Using a Prime Pillar can be very helpful for extruders that use multiple filaments through one nozzle. The Prime Pillar will be the first thing printed when changing nozzles. This will ensure that the nozzle is primed and ready to print before it continues the print.

See this helpful article: <https://www.simplify3d.com/support/articles/printing-with-multiple-extruders/>

However, some users have reported issues with the automatic Prime Pillar created by these S3D settings. They recommend developing your own Prime Pillars, especially if there are multiple colors per layer. Here are two example images from <https://forum.simplify3d.com/viewtopic.php?t=1266> and <https://www.thingiverse.com/thing:1553962>:



## Use Prime Pillar

On/Off `<usePrimePillar>` [Pillar]

Creates a pillar that is used to prime the extruder after a tool change command.

## Prime Pillar Extruder

Choice `<raftExtruder>` [P-Extr]

The extruder that will be used for the prime pillar. If “All Extruders” is selected (default), the pillar will be printed by multiple extruders to efficient prime each extruder after a tool change command.

## Pillar Width

mm `<primePillarWidth>` [PPWidth]

The width of the prime pillar.

## Pillar Location

Choice <primePillarLocation> [PPLoc]

The location of the prime pillar relative to the part.

The values in the .fff file for this XML entry are numbered [0 ... 7] to represent [North, North-East, East, South-East, South, South-West, West, North, North-West].

## Speed Multiplier

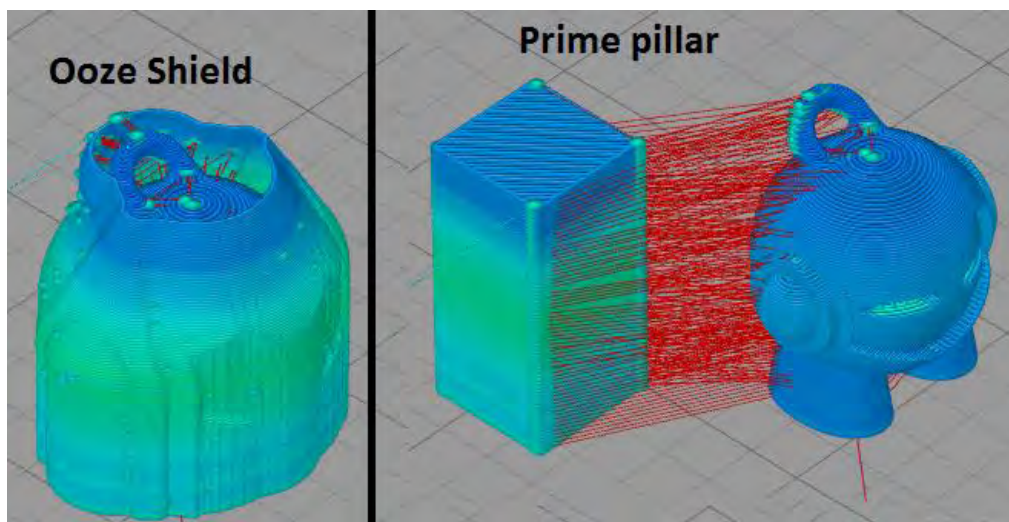
% <primePillarSpeedMultiplier> [PPSpMult]

Modifies the print speed of the prime pillar.

## Ooze Shield Settings

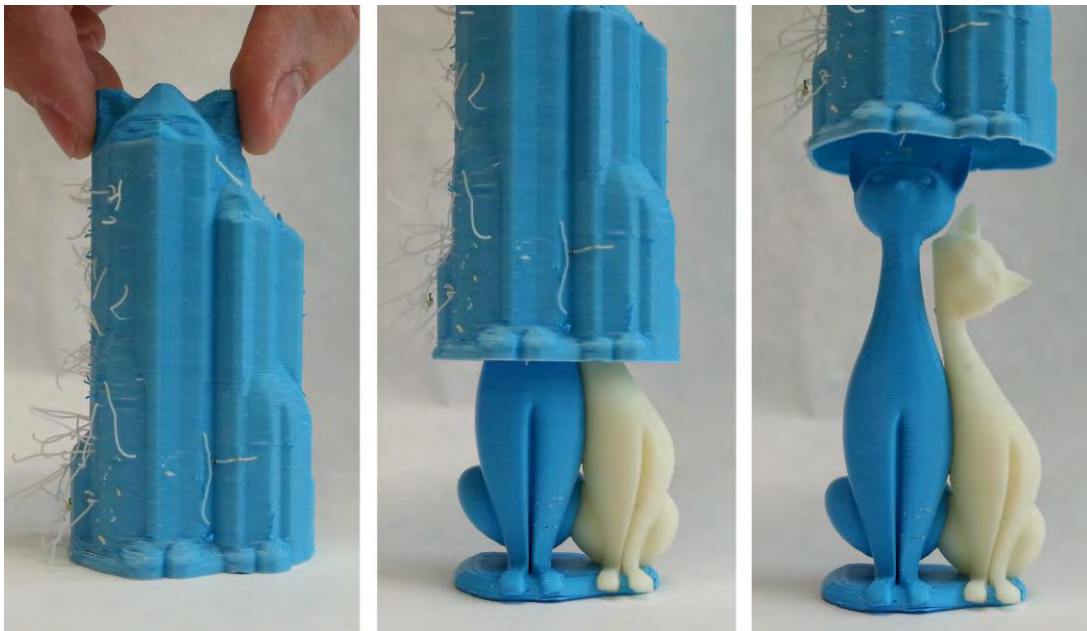
When printing with multiple extruders, using the option for an Ooze Shield is highly recommended. By default, profiles set up for dual extrusion will have the Ooze Shield feature enabled. This will help ensure that leaking and oozing during the print will wipe off on the shield rather than your models.

Here is an excellent image that demonstrates the difference between an Ooze shield and a Prime Pillar. It is from a Russian-language page on Simplify3D: [http://gadget3d.ru/blog/Simplify3d\\_nachalo/](http://gadget3d.ru/blog/Simplify3d_nachalo/)



... and here is a demonstration of an ooze shield in action. However, it is actually a photo of an ooze shield created by another slicer, IceSL. This photo was retrieved from <http://shapeforge.loria.fr/icesl/>:





## Use Ooze Shield

On/Off `<useOozeShield>` [Ooze]

Creates a shell around your model that will help prime the extruder after a tool change and can also catch extra oozing from idle extruders.

## Ooze Shield Extruder

Choice `<oozeShieldExtruder>` [O-Extr]

The extruder that will be used for the ooze shield. If “All Extruders” is selected (default), the pillar will be printed by multiple extruders to efficient prime each extruder after a tool change command.

Which extruder will be used for the Ooze Shield. Selecting All Extruders is recommended.

## Offset from Part

mm `<oozeShieldOffset>` [OOffset]

The distance between the part and the inner-most ooze shield outline.

The distance between the part and the Ooze Shield. A setting of 0.2 mm is recommended.

## Ooze Shield Outlines

Count `<oozeShieldOutlines>` [OPerims]

The number of ooze shield outlines to print on each layer.

If you have two extruders and two outlines set for this setting, two outlines will print.

## Sidewall Shape

Choice <oozeShieldSidewallShape> [OShape]

The shape of the ooze shield walls. The “**Waterfall**” and “**Contoured**” options allow the shield to be closer to the part. However, the final ooze shield can be more difficult to remove.

Vertical will go directly vertical from the base of the Ooze Shield. **Waterfall** will flow similar to how a waterfall would flow off the model as if pouring water directly down on the model. It will branch outward but will not come back inwards. However, **Contoured** sidewalls follow the shape of your print, even if that means they are smaller at the bottom than at the top.

The values in the .fff file for this XML entry are numbered [0 ... 2] to represent [**Vertical**, **Waterfall**, **Contoured**].

## Sidewall Angle Change

Degrees <oozeShieldSidewallAngle> [OAngle]

The maximum allowed change for the ooze shield walls. Only valid for “**Waterfall**” and “**Contoured**” sidewall shapes.

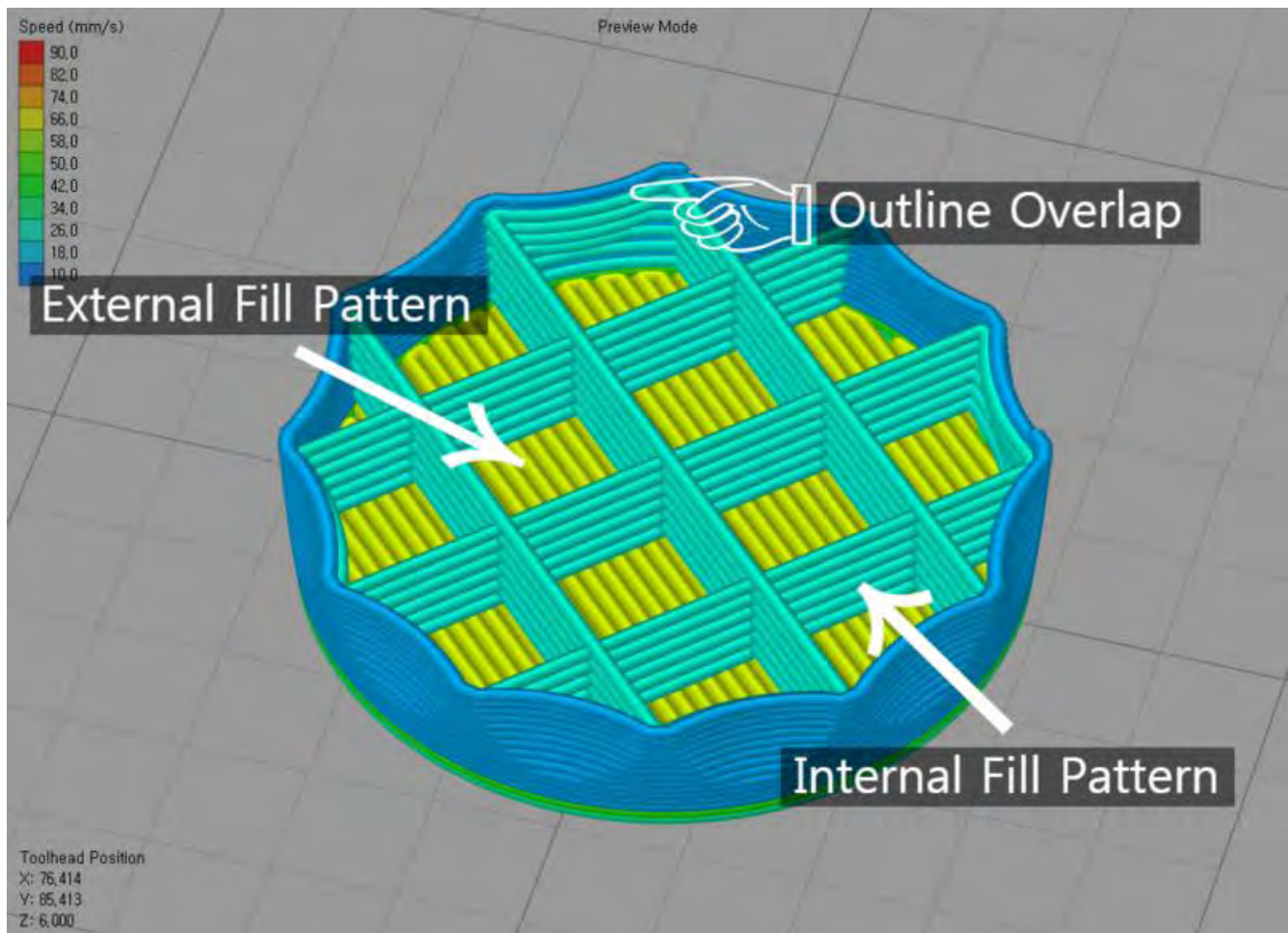
## Speed Multiplier

% <oozeShieldSpeedMultiplier> [OSpMult]

Modifies the print speed of the ooze shield.

## Infill Tab

This image from the Naver blog posts is a nice display of the basic Infill settings:



### General

#### Infill Extruder

Choice <infillExtruder> [I-Extr]

Choose extruder for the sparse interior regions of your part.

Pick which extruder you'd like to extrude the infill of your part with.

#### Internal Fill Pattern

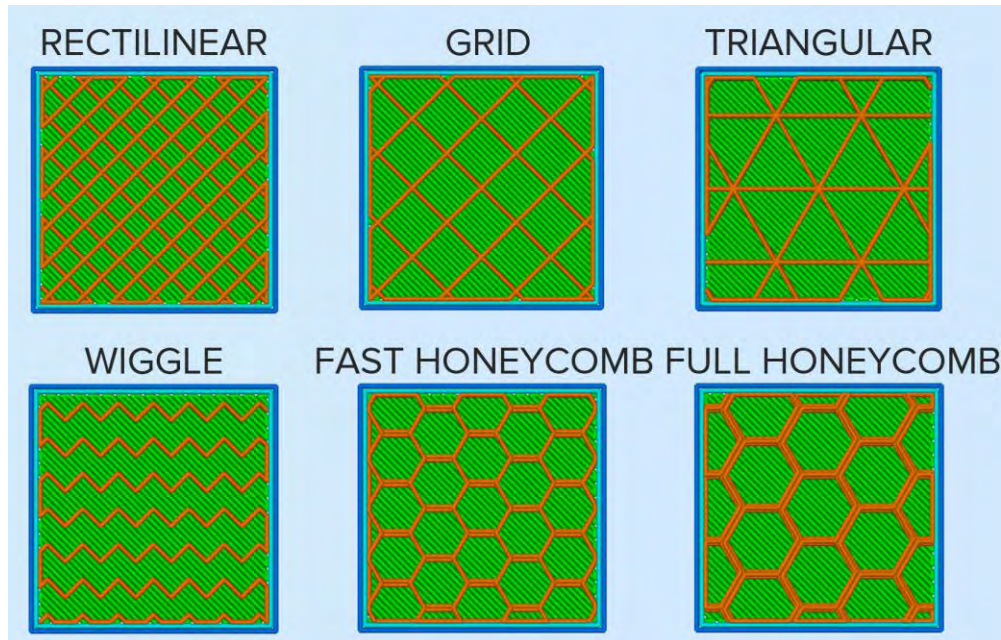
Choice <internalInfillPattern> [IntPat]

Determines the infill pattern used for the interior of the part.

The value of the XML parameter <internalInfillPattern> in the .fff file will be **Rectilinear**, **Grid**, **Triangular**, **Wiggle**, **Fast Honeycomb**, or **Full Honeycomb**. The value for the *IntPat* parameter in a Brief Profile Display is **Rec**, **Grid**, **Tri**, **Wig**, **FastH**, or **FullH**, respectively.

This parameter is critical for the strength of the printed part. See the **Strength, Speed, Cost, and Quality** section. I have often heard that **Full Honeycomb** provides the strongest infill pattern. However, testing

suggests that fused filament is strongest along the axis that it was extruded / deposited. That means that setting **Internal Fill Pattern** to **Rectilinear** is typically the best choice for strength.



### External Fill Pattern

Choice <externalInfillPattern> [ExtPat]

Determines the infill pattern used for the external surfaces of the part.

The top and bottom solid layers will be printed with this choice. Either **Rectilinear** or **Concentric**. **Concentric** infill will mimic your perimeter outlines, coming inwards. For a cylinder, it would make smaller and smaller circles.

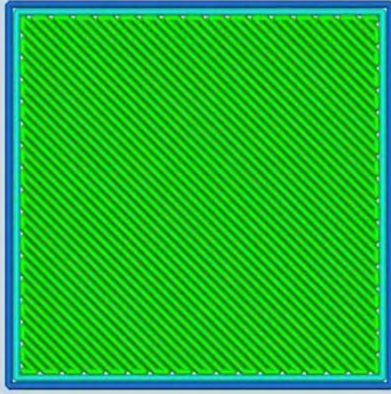
**Rectilinear** is the more common choice, it will fill in using straight lines.

**Note:** Apparently, **Concentric** disables all bridging! (according to forum user cssmythe3 – Wed Mar 16, 2016 11:50 am).

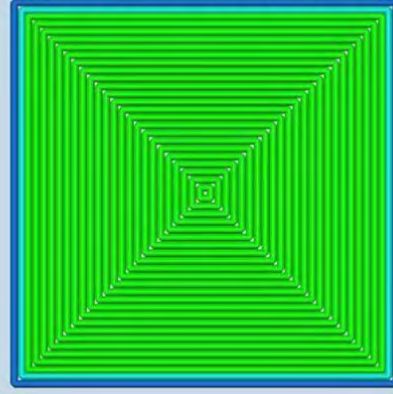
The value of the XML parameter <externalInfillPattern> in the .fff file will be **Rectilinear** or **Concentric**. The value for the **ExtPat** parameter in a Brief Profile Display is **Rect** for **Rectilinear** and **Conc** for **Concentric**.



RECTILINEAR



CONCENTRIC





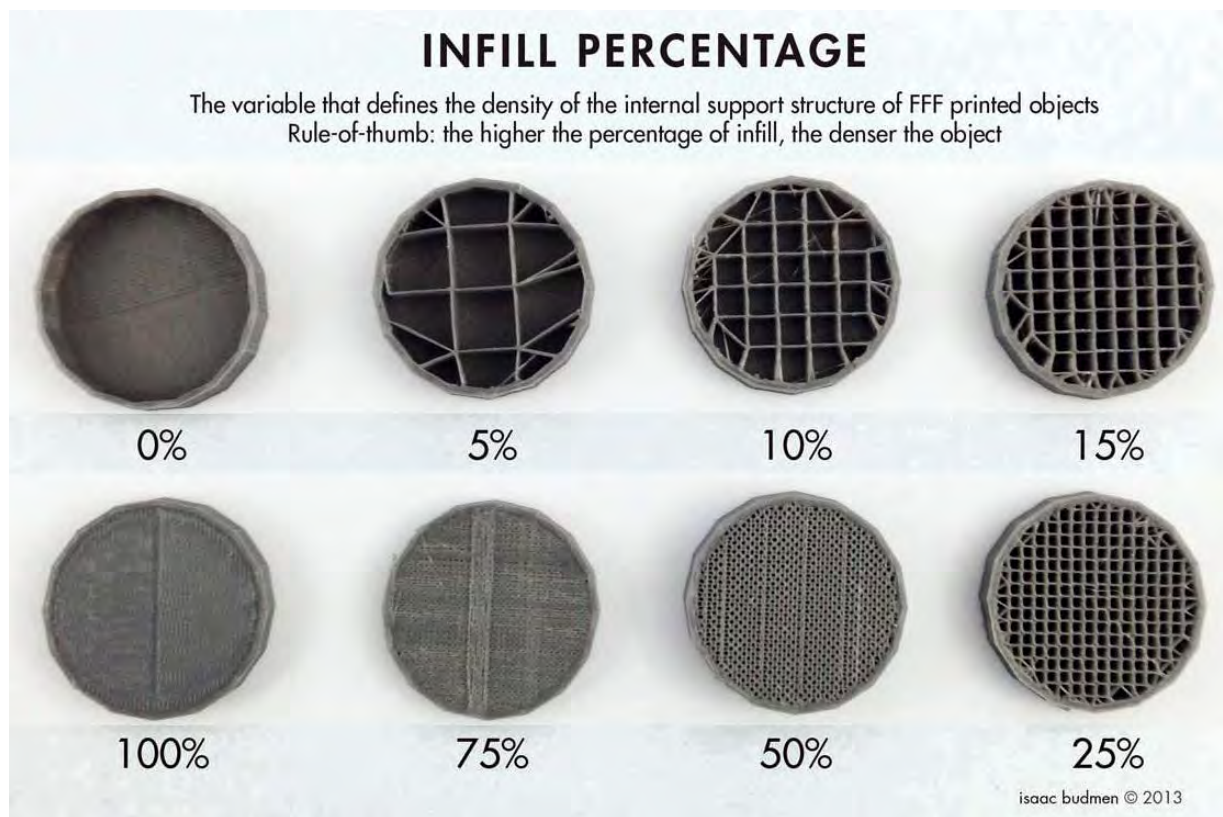
## Infill Fill Percentage

% <infillPercentage> [Infill]

Determines the interior solidity of the model.

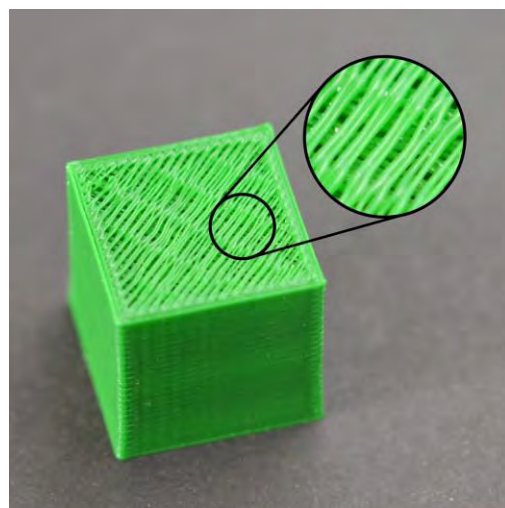
What % your part you'd like to be filled in.

Here's a nice demonstration from <http://www.diyaudio.com/forums/multi-way/273053-3d-printing-6.html> (although I have no idea what slicer was used for these samples):



This parameter is sometimes implicated in situations where there are holes or gaps in the top layer. See the **Holes and Gaps in the Top Layers** section of the *Simplify3D Print Quality Troubleshooting Guide* at <https://www.simplify3d.com/support/print-quality-troubleshooting/>.

This parameter is critical for the strength of the printed part. See the **Strength, Speed, Cost, and Quality** section. The higher the **Infill Fill Percentage**, the stronger the part ... with one exception: Parts with 100% **Infill Fill Percentage** will take the most stress before they break, but they will “Yield” (permanently deform) at a lower strain than a 90% **Infill Fill Percentage**. Additionally, 100% infill can reduce the quality of the



surface of the part. So, an **Infill Fill Percentage** setting of 90% may be the best all-around choice when you want a strong part.

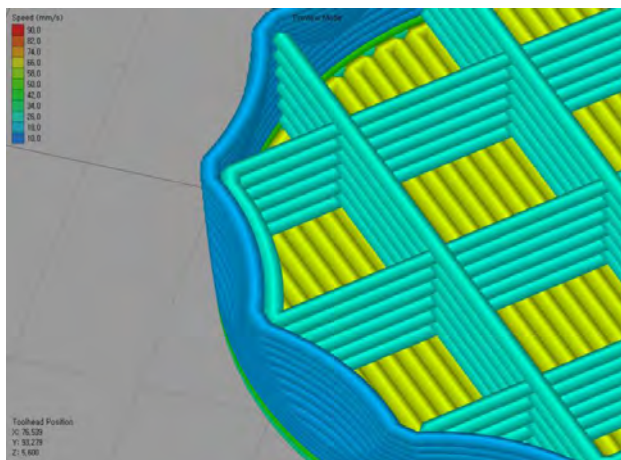
## Outline Overlap

% <outlineOverlapPercentage> [OutOvr]

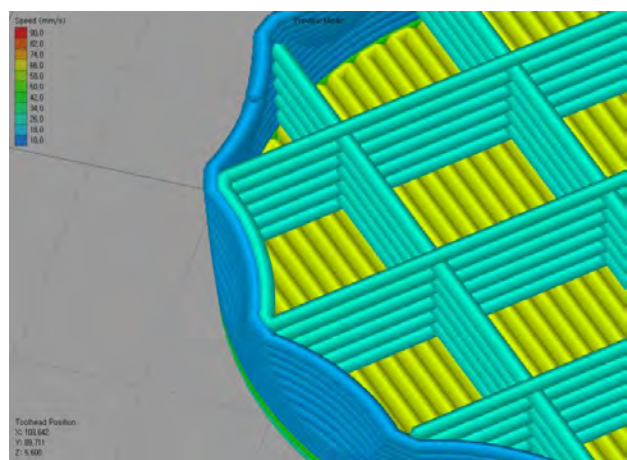
Percentage of **Extrusion Width** that will overlap with outline perimeters (ensures infill bonds to outline).

This value will dictate how far into the perimeter the infill goes. If this value was 0, the infill would start of the perimeter and not overlap at all. At 100%, you would have complete overlapping. Calibrating this value, I would say the average value I find that works well is 30% for the printers I've used, but it does vary.

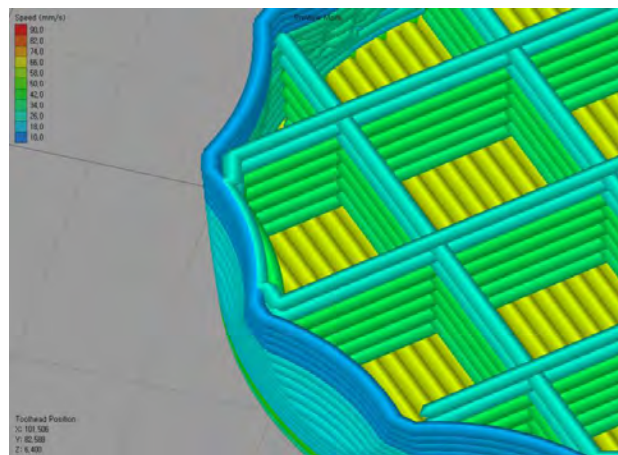
I think these three images from the Naver blog posts highlight the effect of this parameter:



Outline overlap 0%



Outline overlap 50%

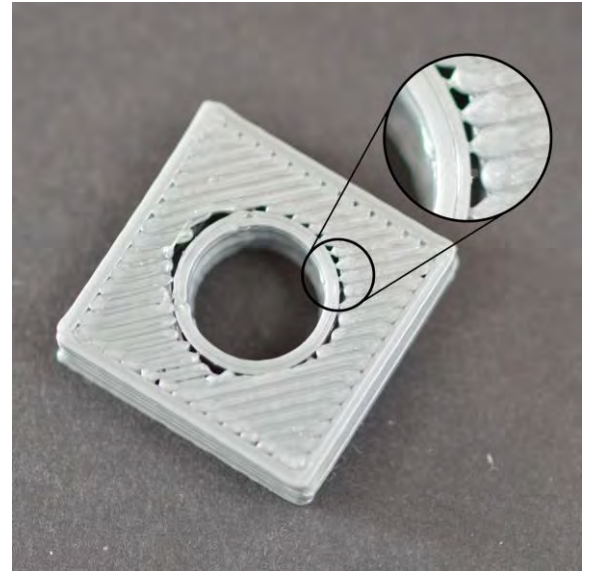


Outline overlap 100%

Outline Overlap determines the amount of overlap between the inner fill and the outer wall. This is usually set to 20%. For writing cups or airtight instruments, use a setting over 50%.

From the *Simplify3D Print Quality Troubleshooting Guide* at <https://www.simplify3d.com/support/print-quality-troubleshooting/#gaps-between-infill-and-outline>:

Each layer of your 3D printed part is created using a combination of outline perimeters and infill. The perimeters trace the outline of your part creating a strong and accurate exterior. The infill is printed inside of these perimeters to make up the remainder of the layer. The infill typically uses a fast back-and-forth pattern to allow for quick printing speeds. Because the infill uses a different pattern than the outline of your part, it is important that these two sections merge together to form a solid bond. If you notice small gaps between the edges of your infill, then there are several settings you may want to check.



Simplify3D includes a setting that allows you to adjust the strength of the bond between the perimeter outlines and the infill. This setting is called the **Outline overlap** and determines how much of the infill will overlap with the outline to join the two sections together. This setting can be found by going to “Edit Process Settings” and selecting the Infill tab. The setting is based on a percentage of your **Extrusion Width**, so that it easily scales and adjusts for different nozzle sizes. For example, if you are using a 20% outline overlap, it means that the software will instruct the printer so that the infill overlaps with 20% of the inner-most perimeter. This overlap helps to ensure a strong bond between the two sections. As an example, if you were previously using an outline overlap of 20%, try increasing that value to 30% to see if the gaps between your perimeters and infill disappear.

### **Infill Extrusion Width**

% <infillExtrusionWidthPercentage> [InWid]

The width of the infill extrusion versus the outline perimeters.

This % will modify your extrusion width under the extruder tab for the Infill. If you have a 0.5 mm **Extrusion Width** under the Extruder tab and set a 200% **Infill Extrusion Width**, your Infill will be 1 mm thick.

### **Minimum Infill Length**

mm <minInfillLength> [MinLen]

Infill segments with a total length below this value will not be printed (helps save time for unnecessary segments).

This is default set to 5 mm, if there are small sections in the G-Code previewer that aren't filling in, you may want to set this to a lower value.



## Combine Infill Every \_\_\_ layers

Count <infillLayerInterval> [Combine]

Print multiple infill layers at once for faster printing times. For example, if your primary layer height was 0.1 mm and you had chosen to combine the infill every 3 layers, this would print a single 0.3mm thick infill pattern every 3<sup>rd</sup> layer. Set to 1 to disable.

If you'd like to only print Infill once every 2 or every 3 layers, then you'd place that number in the input. My experience has been that most printers can print infill at 0.3 mm just fine. Therefore, if I'm at 0.1 mm **Primary Layer Height**, I'll use 3 sparse layers. If I'm at **Primary Layer Height** of 0.15 mm I'll use 2 sparse layers.

Earlier versions of S3D called this setting **Print Sparse Infill Every \_\_\_ layers**.

## Include solid diaphragm (Switch)

Yes / No <useDiaphragm> [IncSolid]

Creates solid horizontal layers to improve structural integrity.

## Include solid diaphragm every \_\_\_ layers

Count <diaphragmLayerInterval> [DiaphEvery]

Layer interval for solid horizontal diaphragm layers.

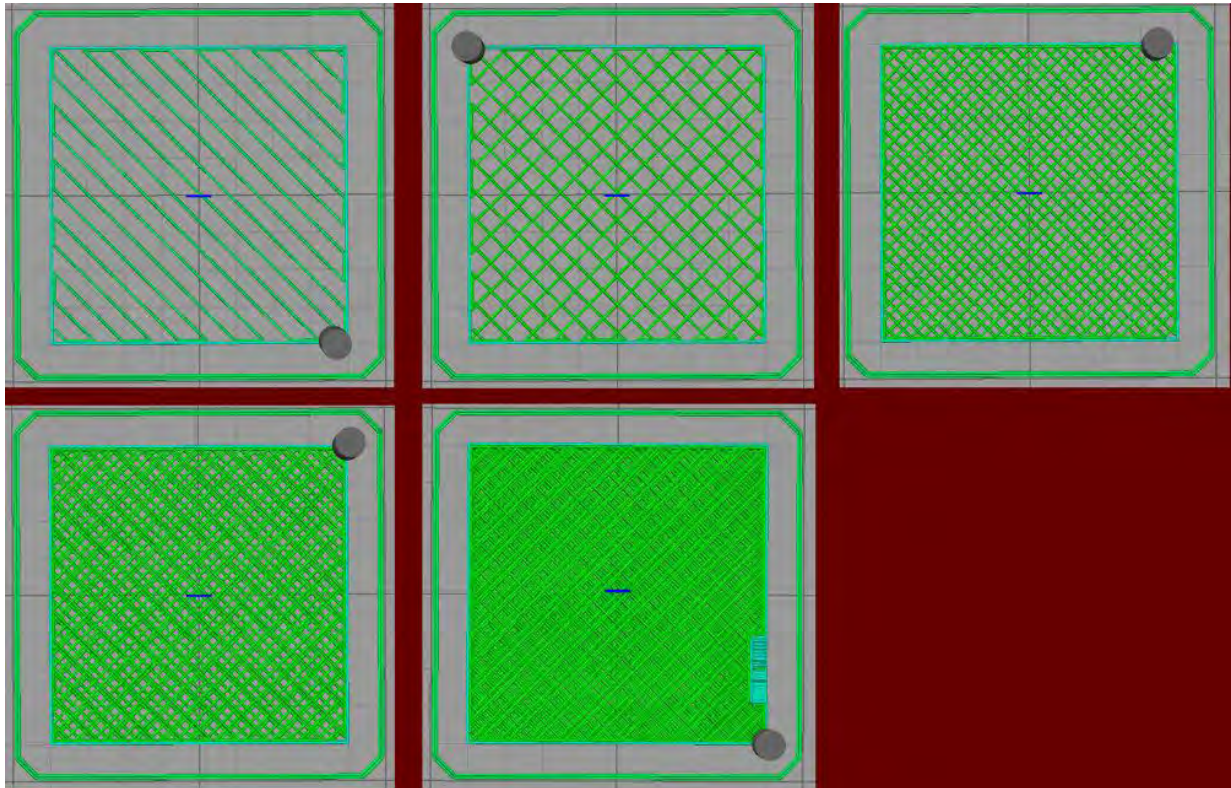
A solid diaphragm is a layer of infill at 100%. For certain parts, this can be helpful, if you want to keep the infill % low, but also add the extra structural rigidity a 100% layer of infill can provide.

### *Use random Infill placement for each layer*

This appears to have been an option prior to S3D v3.0, but is not in the current version of S3D.

When this option is enabled, each layer of infill will start at a random spot. This means that the lines of infill won't line up. I personally am a fan of keeping this option off, stacking the layers of infill on-top of each other seems to provide stronger parts.

The demise of this setting may have been unfortunate in some circles, since folks in the acoustic design community were apparently using this setting to create “foam-like” structures in speaker cabinets. See <http://www.diyaudio.com/forums/multi-way/273053-3d-printing-6.html>, which shows the progression of the build after layers 1, 2, 4, 8, and 20 layers:





## Internal Infill Angle Offsets

### Infill Angles

Degrees <internalInfillAngles> [IntAng]

Internal angles will be used one after another for each successive layer.

Standard angles are 45 / -45. That means that layer infill angles and external infill will print at alternating 45-degree angles. These values traditionally don't need to be adjusted, but I'm also a fan of using 0 / 45 / -45 / 90 as the angle orientations.

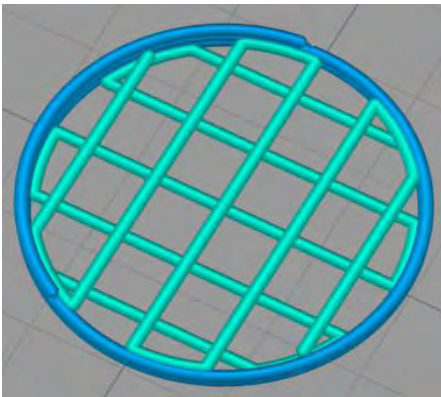
This parameter affects the strength of the printed part. See the **Strength, Speed, Cost, and Quality** section. Bonds between neighboring filament extrusions on the X-Y plane are weaker, and those between layers along the Z axis are weaker still. If you want a strong part, set the **Infill Angles** to be parallel to the axis along which you expect the greatest loads.

### Print every infill angle on each layer

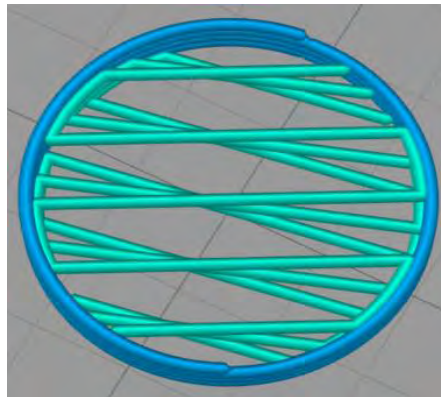
Yes / No <overlapInternalInfillAngles> [EvAngle]

This will print every single infill angle for each layer. Can create a stronger pattern, although it may cause overlap from crossing extrusions.

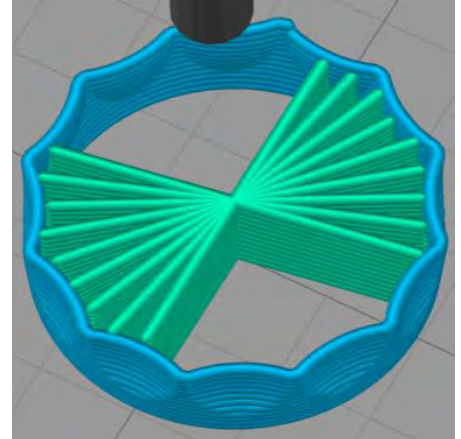
Interesting effects can be obtained with creative use of infill angles. The Naver blog posts show some examples of geometry that would be difficult to obtain during the design phase, but are straightforward as slicer settings in S3D:



The normal output,  
based on angles of  
-45 / 45 degrees



Changing angles to  
10 / 20 / 30 degrees



Setting angles of 60 / 70 / 80 / 90,  
and selecting **Print every infill  
angle on each layer**

## External Infill Angle Offsets

### External Angles

Degrees <externalInfillAngles> [ExtAng]

The external fill is the finish on the outside of the part.<sup>2</sup> This parameter sets the angle of those external fill lines. For example, if you set the **External Fill Pattern** to **Rectilinear**, you can see that you then get infill angles of +45° and -45° by default. If you try changing those to -90° and +90° and you will see the difference.

## Discussion

**KeyboardWarrior – Tue Jan 20, 2015 12:08 pm**

*montressor wrote:* Thanks for these, again, hugely useful. One question about the "Print Sparse Infill...": I assume using this to skip layers \*must\* be used in conjunction with increasing the "Infill Fill Percentage" to ensure the proper height of fill is being used, correct? I changed the number of layers I printed without an according increase in the fill percentage and wound up with a bit of a mess.

Subsequent question: how big a percentage can you do here? Up to the nozzle width? e.g.: with a 0.4mm nozzle printing 0.1mm layers, I could set "Infill Fill Percentage" to 400% and set my "Print Sparse Infill" to every 4 layers?

The software will calculate and scale the extrusion amount for the sparse feature, so you don't have to worry about that. You'll still get your infill %, since the sparse infills will have modified extrusion values. That is a good thought though

**JFettig – Tue Jan 20, 2015 12:49 pm**

Infill extrusion width doesn't increase the width of infill except for where it crosses previous layers, it increases the depth – if you use only 2 orientations (45 / -45) and 200%, it will make a relatively solid infill and not just be a pile of strings, it strengthens the part significantly, adding another angle to it negates that and the infill isn't stacked. Using random infill placement has no use, it only has negative effects.

Using sparse infill is also kind of pointless IMO – it extrudes more material per infill pass but only makes infill every so many layers specified – set to 2, every 2 layers of border, it'll infill once, but twice as much material, it'll be crude.

In my opinion, both angles the infill is laid, should be laid on the same layer to make solid infill and have to use the extrusion width at 100% like every other slicer does it, its stronger and more reliable, it only takes a little longer

**TenKOhms – Tue Jan 20, 2015 1:03 pm**

JFettig wrote: Infill extrusion width doesn't increase the width of infill except for where it crosses previous layers, it increases the depth – if you use only 2 orientations (45 / -45) and 200%, it will make a relatively solid infill and not just be a pile of strings, it strengthens the part significantly, adding another angle to it

---

<sup>2</sup> This information was provided by S3D forum users rrdavis and aircsapes on 4/23/2018 in response to my query on this parameter. Thanks!

negates that and the infill isn't stacked. Using random infill placement has no use, it only has negative effects.

Using sparse infill is also kind of pointless IMO – it extrudes more material per infill pass but only makes infill every so many layers specified – set to 2, every 2 layers of border, it'll infill once, but twice as much material, it'll be crude.

In my opinion, both angles the infill is laid, should be laid on the same layer to make solid infill and have to use the extrusion width at 100% like every other slicer does it, its stronger and more reliable, it only takes a little longer

Increasing infill extrusion width definitely helps increase the extrusion rate of infill. I've seen a couple pictures of a "whispy" infill vs one with increased extrusion width, and the width, solidarity, and rigidity were noticeably different. IMO, this is one of the most useful features I've seen added.

Sparse infill is not useless, and in most cases (at least when I last checked), many of lulzbot's company profiles (for a different slicer) use sparse infill. It helps save time by not having to do infill every layer. This way you can have a fine outer perimeter, yet have a coarse infill.

With an increase in my infill extrusion width, I've noticed much more strength in my parts.

***Jimc – Tue Jan 20, 2015 5:23 pm***

x2 gotta agree with 10k. the infill width setting is great. jfettig is right though that in normal instances it does not actually increase the width because there is a 1 layer gap below it so the extrusion wont squish to the proper width. it just pumps out more material to make the infill solid and fill that 1 layer gap. the strength is night and day. sparse infill works great for time savings as well. it doesnt however work good for petg. abs it always worked great for.

***Dsegel – Tue Jan 20, 2015 9:01 pm***

You may want to add something about the density that is required for a good surface finish – I find that you need at least 20% infill to get a smooth surface finish on top layers. Below that and the surface threads can fall between the infill layers, leading to a bumpy or less-than-solid surface.

***cssmythe3 – Wed Mar 16, 2016 11:50 am***

External Infill Pattern: The top and bottom solid layers will be printed with this choice. Either Rectilinear or Concentric. Concentric infill will mimic your perimeter outlines, coming inwards. For a cylinder, it would make smaller and smaller circles.

Rectilinear is the more common choice, it will fill in using straight lines.

Turning on Concentric disables all bridging – I'm not sure why.

Also, all solid layers, not just the bottom and top, get set to this method.

***jmunkki – Wed Mar 15, 2017 3:03 am***

My Wanhao i3 with a Flexion Extruder seems to have issues extruding PETG at high rates. I have had my infill width set to 125% for some reason and on large large infills, the extruder started skipping (motor

stalling). The parts still come out OK, although the infill obviously has a few gaps here and there. Outer surfaces are fortunately perfect. It took me quite a while to figure out what was going on.

It would be nice if S3D had a flow rate limit somewhere. What I mean is that any extrude G-Code would be limited by filament extrusion rate in addition to other speed limits. Whichever would be lower would set the limit.

Mechanical accuracy of course affects quality, but in my case the limiting factor is the extruder. For example, increasing the layer height can increase the flow rate beyond what the extruder can handle. I think upgrading to the HT version of the Flexion might improve my print speeds. I already upgraded the stepper a few months ago (the old one was dying), but I'll probably check the reference voltage in the near future to see if I can get a bit more torque out of the one I have now.

P.S. After the print, I tested a 0.3mm layer PETG print that worked two days ago and found that it was also making the stepper stall, so I took the whole extruder assembly apart, cleaned everything and replaced the PTFE tube (and ordered a Flexion HT upgrade). It's possible running some cleaning filament would have done the same trick, but at least now the 0.3mm print is working fine again. What I said above still applies though: if your printer is thermally limited rather than mechanically, it would be nice if there was a filament mm/minute speed limit option. Also, the infill width setting can cause issues.

## Support Tab

One of the best features in Simplify3D is the support material. The option of where to put it, in addition to how easily it breaks off makes it a key feature. For more info, see:

<https://www.simplify3d.com/support/tutorials/adding-and-modifying-support-structures/>

### Support Material Generation

#### Generate Support Material

Yes / No <generateSupport> [GenSupp]

Determines if support material will be used for this model.

#### Support Extruder

Choice <supportExtruder> [SuppExtr]

Choose extruder that will be used for support material.

Which extruder your support material will be printed with.

#### Support Infill Percentage

% <supportInfillPercentage> [SuppInfill]

Adjust the spacing between support material webbing.

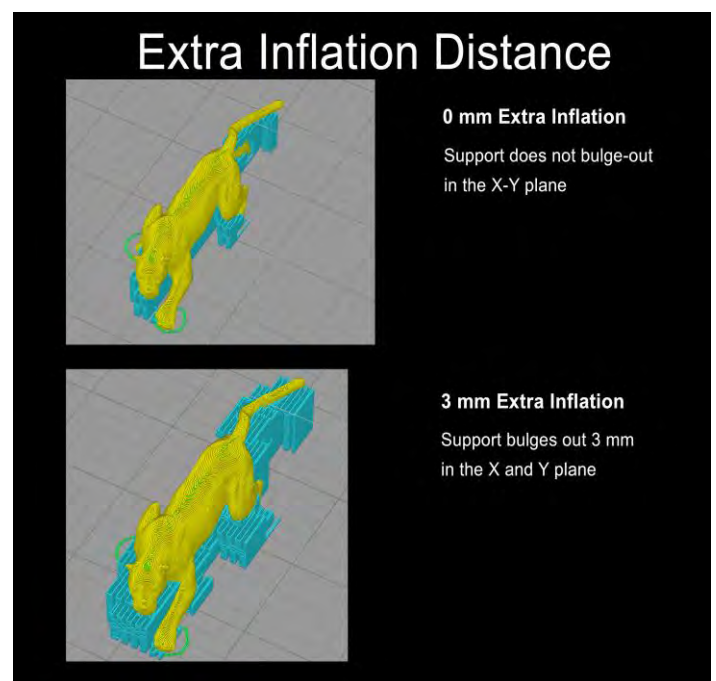
The Infill % your support will be printed with. For PLA/ABS anywhere from 20 to 40% usually works well. I've typically left this on the default value.

#### Extra Inflation Distance

mm <supportExtraInflation> [ExInflDist]

Use this setting to expand the support structure foundation beyond the exterior of your part.

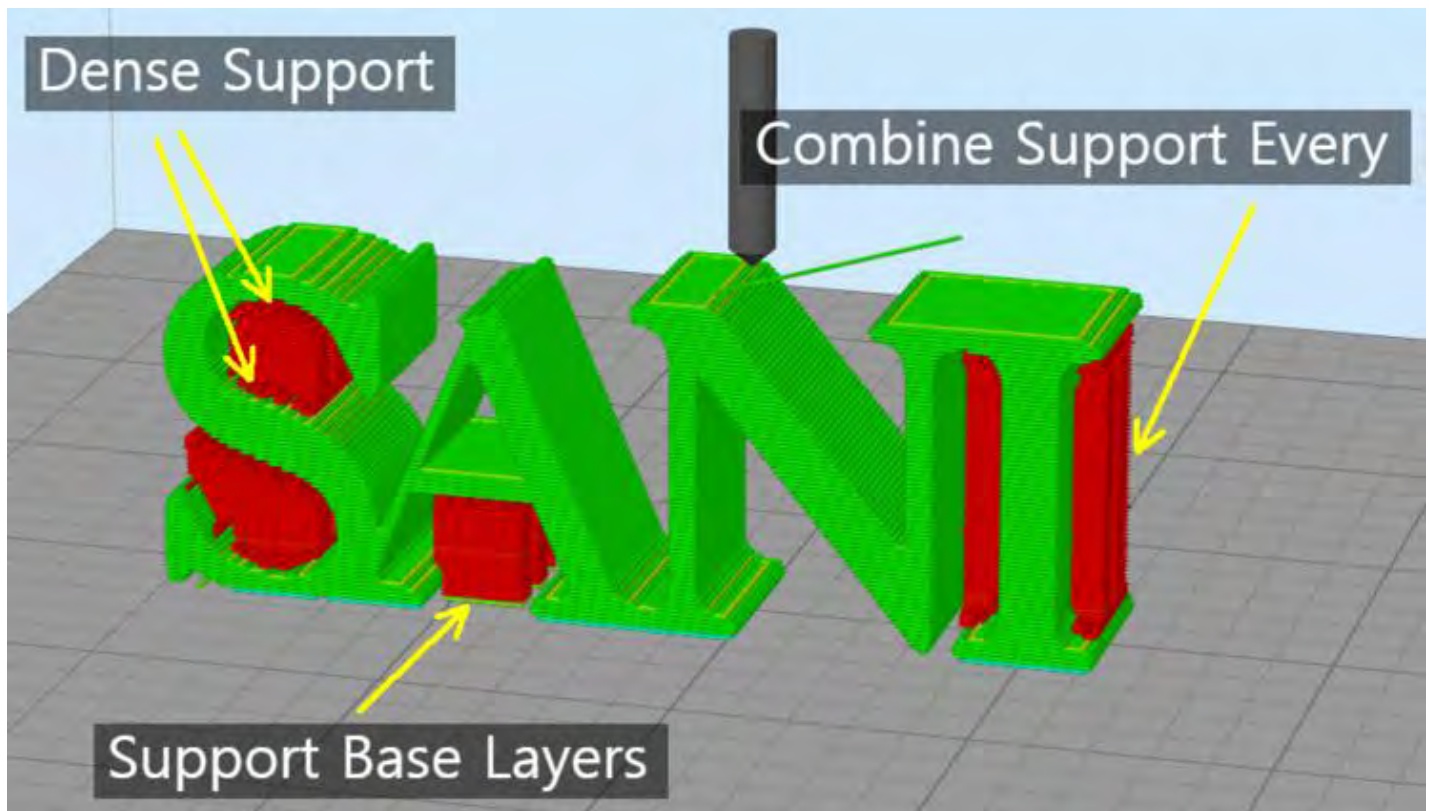
This will increase the amount of support generated in the X-Y plane.





## Support Settings

I think this image from the Naver blog posts captures the important aspects of the next three parameters:



### *Support Base Layers*

Count `<supportBaseLayers>` [BaseLayers]

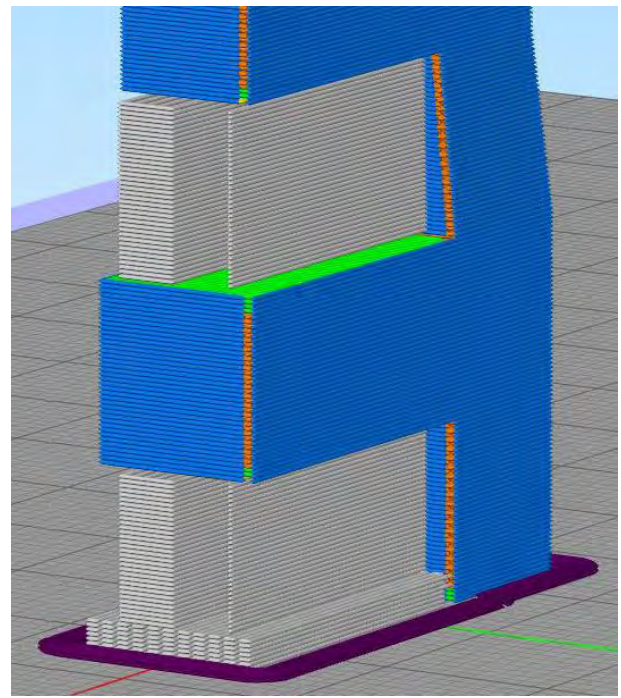
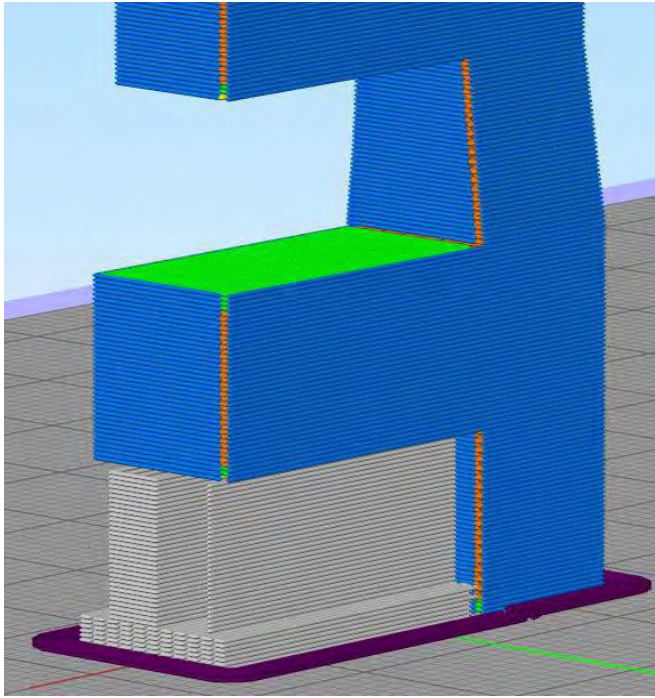
Number of solid support layers to include at the base of the print to help with adhesion. Set to zero to disable.

### *Rafts Exclusively for Support Structures*

This technique is from a S3D forum post by user Sylvian on 4/24/18 that was answered by Doug Kightley. Thanks to user Airscapes for pointing out this useful technique:

I'm looking for a way to add raft only under support material touching the printbed. I don't want a raft under the printed model, but I need more stability under support pillars.

**Support Base Layers** will provide a good base to supports ... but it won't look like a raft! The image on the left shows supports only from the bed (**Support Type** is set to **From Build Platform Only**) and not above the model to an underside above it. In this case, there are seven support base layers on the bed:



The image on the right shows supports above the model as well. These supports do not have base layers. To do this, use the wizard to create two processes and set just the lower process to have **Support Base Layers**.

### Combine Support Every \_\_\_\_ layers

Count <supportLayerInterval> [Combine]

Print multiple support layers at once for faster printing times. For example, if your **Primary Layer Height** was 0.1 mm and you had chosen to combine the supports every 3 layers, this would print a single 0.3mm thick support pattern every 3<sup>rd</sup> layer. Set to 1 to disable.

Earlier versions of S3D called this setting **Print Support Every \_\_\_\_ layers**.

Similar to sparse infill, sparse support means that it will extrude for your support, but only print the support every \_\_\_\_ layers. For instance, if you have a **Primary Layer Height** of 0.2 mm and **Combine Supports Every 2 Layers**, then it will print your support at a layer height of 0.4 mm.

## Dense Support

### Dense Support Extruder

Choice <denseSupportExtruder> [DenseSupp]

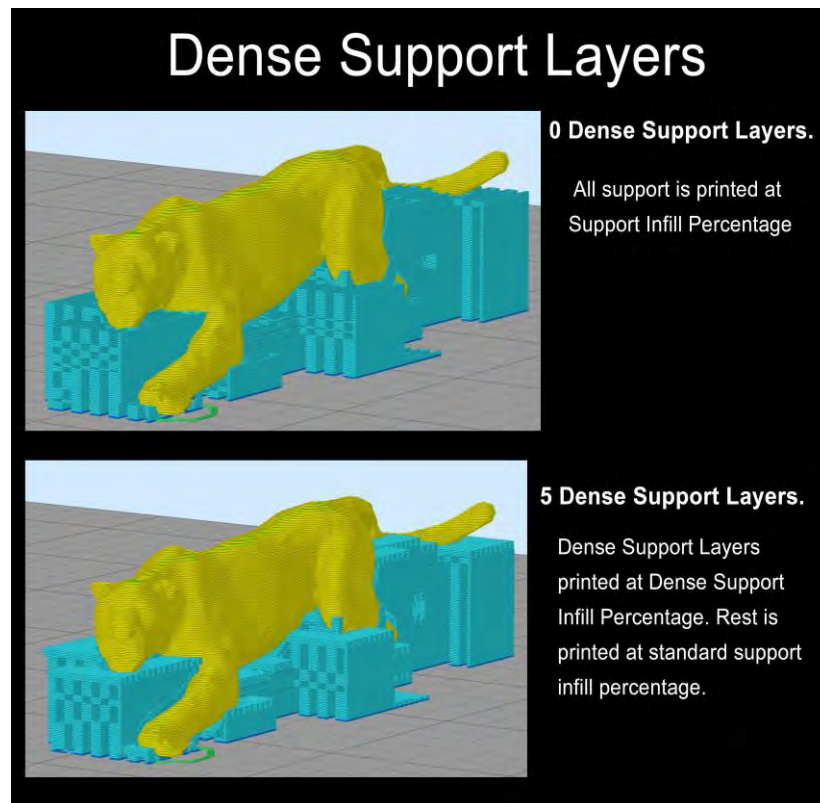
Choose extruder that will be used for dense support layers.

### Dense Support Layers

Count <denseSupportLayers> [DenseLayers]

Number of dense support layers to include at the interface between the part surface and the normal sparse support. Set to zero to disable.

How many of the layers closest to your part will be filled with the dense infill percentage.



### Dense Infill Percentage

% <denseSupportInfillPercentage> [DenseInfill]

Infill percentage for the dense support layers.

The infill percentage for the top layers closest to your part. If you use PVA/HIPS filament, this feature is definitely a must-have, so you can print the bulk of your support at 20–30% support, then bump it up to 60–100% for the layers that will have contact with your part.

## **Automatic Placement**

Only used if manual support is not defined.

### **Support Type**

Choice <supportType> [APType]

Choices are Normal and From Build Platform Only.

### **Support Pillar Resolution**

mm <supportGridSpacing> [APRes]

Determines the resolution used for support material calculations.

The size of the pillars generated. This is default at 4 mm, for smaller parts you'll most likely need to lower this to one or two mm to get the support pillars you want.

### **Max Overhang Angle**

Degrees <maxOverhangAngle> [APAngle]

If no manual support has been defined, automatic support will be added to support overhang angles greater than this angle (0 = vertical, 90 = horizontal).

The max overhang angle that will be allowed. I recommend this calibration test piece for figuring out how well your printer works with support generally: <http://www.thingiverse.com/thing:40382>

## **Separation from Part**

### **Horizontal Offset From Part**

mm <supportHorizontalPartOffset> [HSep]

Modifies the horizontal separation distance between support structure and part outline (allows for easy cleanup).

How far away on the X-Y plane the support will be generated from your part.

### **Upper Vertical Separation Layers**

Count <supportUpperSeparationLayers> [UpLay]

Modifies the number of upper separation layers from part outline (allows for easy cleanup).

How far off in the Z-plane the support will be set from your part. Generally, you can keep this set to 1, unless you're using HIPS/PVA, you may want to set it to zero.

### **Lower Vertical Separation Layers**

Count <supportLowerSeparationLayers> [LowLay]

Modifies the number of lower separation layers from part outline (typically limited to 0 or 1 to ensure support structure has solid base).

## **Support Gap in between Processes**

This section is from <http://www.forum.simplify3d.com/viewtopic.php?t=1973> ...



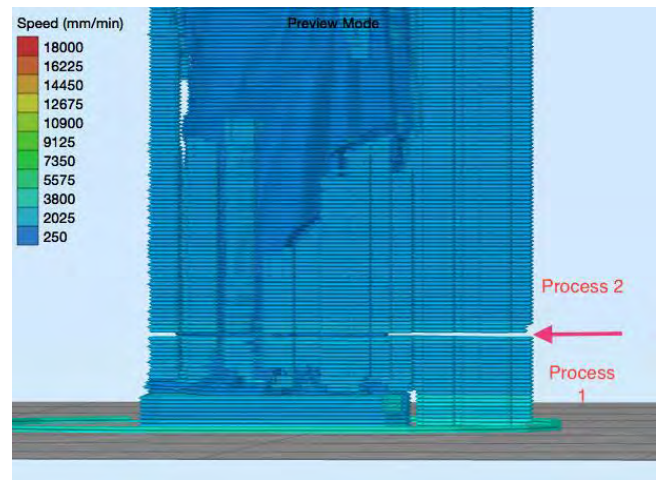
This is something will happen if you set up a part with two processes that has support. The default settings for support material configuration will set it up, so that you will have a gap of 2 layers in the middle of your support that is very-much unwanted.

The screenshot below is for a print of the Statue of Liberty, default supports and two processes:

- Process 1 – From 0 to 5 mm
- Process 2 – From 5 mm onwards

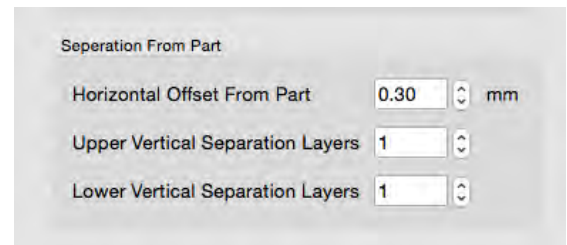
You can clearly see that there's missing support layers where the arrow points, where Process 1 ends, and Process 2 starts.

To fix this, we need to change settings in the Support Tab for each process.



In the Support tab settings, there's the option for Upper Vertical Separation Layers and Lower Vertical Separation Layers. In order to get rid of the Gap in the support in the picture above, you would need to change these settings.

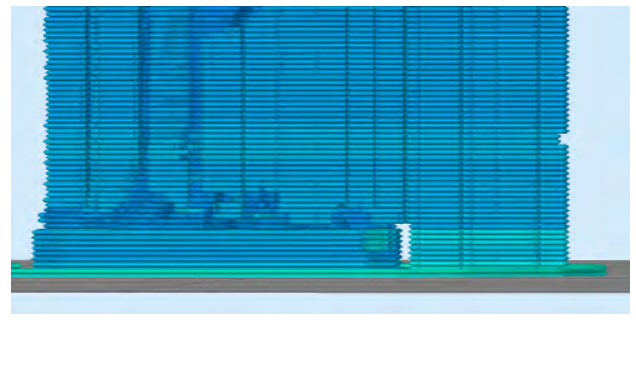
- Process 1 – From 0 to 5 mm: Upper Vertical Separation Layers set to Zero
- Process 2 – From 5 mm onwards: Lower Vertical Separation Layers set to Zero



After those changes are made, the file will slice without any gap between the two processes.

If you're new to printing one part with different settings for different regions, this tutorial is very helpful in explaining the process:

<http://www.simplify3d.com/support/tutorials/different-settings-for-different-regions-of-a-model/>

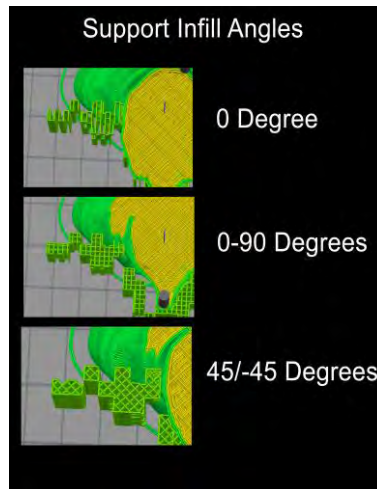




## Support Infill Angles

Degrees <SupportAngles> [SupAng]

Angles used for support structure webbing (typically want only 1 or 2 angles in the list).



### Discussion

**Rebekah\_harper – Thu Mar 12, 2015 9:06 am**

Does the support resolution also control the thickness of the extruded line?

**JoeJ – Fri Mar 13, 2015 5:38 am**

No, it does not. The support extrusion width is the same as every other extrusion width.

**KDan – Sat Jul 18, 2015 8:41 am**

How does support infill percentage differ from support pillar resolution? They would seem to be similar. How do they interact?

**KC\_703 – Sat Jul 18, 2015 9:48 am**

My understanding is the "Pillar Resolution" specifies the granularity of the analysis to determine supports – which could increase slicing time. If the overhang has many details, then it's probably best to specify a higher granularity to ensure the supports are properly generated for the varying levels.

Infill percentage is the density of the supports generated. A lower percentage creates a less dense support area with 3–5 mm gaps in the grid. The infill percentage specifies the amount of actual filament used for supports.

Best way to visualize this is through the "Support customization tool". Vary the resolution from 4.0mm to 2.0mm to see how the finer smaller pillars will fit into an overhang with varying levels (ie. concave overhang). Use in conjunction with "Cross Section" tool to see how the specified "pillars" cut into the edge of the project surface – adjust so the support pillars conform well to the project surface.

So for an overhang which is large in area and has a shallow convex profile – high resolution pillar resolution (2.0mm) and low infill percentage (15–25%) could work.

### **newtoon – Wed Sep 02, 2015 6:09 am**

I would like to know how to:

- 1- space better the supports. I often find them too close to each other when it is not necessary.
- 2- if I print a model in high quality, can I make supports with low quality ? Importing them separately for instance ?

### **KeyboardWarrior – Wed Sep 02, 2015 7:59 am**

Great questions. For #1, you would just want to lower the **Support Infill %**, this would space the supports out further.

For #2, you can print the supports with lower quality by using **Print support every X layers**, so if you're printing with a 0.1 mm layer height, and you set this to 2, you'll only print supports every other layer.

### **Paul M Smith – Tue Sep 08, 2015 3:45 am**

I have print that puts some quite small area supports as islands well away from everything else. This is correct for the support but since they will not be used for some height they are very fragile. Is there any way to make supports link together to increase stability?

### **dkightley – Tue Sep 08, 2015 8:08 am**

I have a support structure set up with Support Infill angles set at 45, -45, 45, 45, 45, 45, 45, 45, 45.

The single layer that is at 90 degrees to the rest gives a slightly more stable support that is still easy to remove.

Doug Kightley

### **dkightley – Mon Oct 12, 2015 6:13 pm**

Another useful tip ...

If you have a standard profile for printing a variety of parts where you want to only add support manually...and don't want support to be generated automatically, then switch support on and set the support resolution pillar size to something ridiculously large, ie too large for the auto-generation to take place.

Then whenever you want to apply manual support, you just need to set the resolution to what you want so you can do the manual support. If you don't want any support ... you have nothing to do. The auto-generation of support will not occur ... as the pillar size is too big.

This will stop you getting a confirmation box asking if you want support turned on after you have added manual support.

## Temperature Tab

---

### Temperature Controller List

List `<temperatureController>` [T-List]

List of temperature controllers.

This is where you'll have your extruders and heated bed listed. If you need to add an extruder/heated bed you would click the button for "Add Temperature Controller"

### Temperature Identifier

Choice `<temperatureNumber>` []

Identifier for this temperature controller. Required for some firmware temperature commands.

The firmware identifier for this toolhead. Most printers use the notation for T0 for the Right Extruder and T1 for the Left Extruder. If you aren't 100% sure, you can open the Machine Control Panel and use the temperature identifier selector in that screen to find out what toolhead is which.

### Temperature Controller Type

Choice `<isHeatedBed>` [TType]

The temperature controller is a heated nozzle (extruder) / heated build platform.

Sets whether your element is an extruder or heated bed ("HBPlat" in the brief SID coding). This makes a difference in the G-Code used to control it. For instance, the RepRap commands for an extruder are M104 and M109 whereas for a heated bed it's M140 and M190.

### Relay Temperature Between Each Layer

Yes / No `<relayBetweenLayers>` [Layer]

Reports temperature to host for monitoring after each layer.

This will set how often to relay the temperature for USB prints.

### Relay Temperature Between Each Loop

Yes / No `<relayBetweenLoops>` [Loop]

Reports temperature to host for monitoring after each loop.

### Wait for temperature controller to stabilize before beginning build

Yes / No `<stabilizeAtStartup>` [WaitStab]

Sends a command at startup to stabilize the selected temperature before proceeding.

## Per-Layer Temperature Setpoints

Degrees°C <setpoint> [SetP]

Defines the temperature at each build layer. To use the same temperature for the entire build, add a single setpoint entry for layer 1.

Allows you to set temperature on a per layer basis. The temperatures that are set are read to the left. The adjustable numbers on the right do not affect your print, only when you click [Add Setpoint] are the temperature commands entered in.

Also, if printing with multiple processes, the layer # chosen is relative to the process. For instance, if this process applied from 5 mm to 10 mm on the model. The temperature for layer 1 would take place at the first layer after 5 mm.



# Cooling Tab

## Per-Layer Fan Controls

% <setpoint> [FanSpeed]

Defines the fan speed at each build layer. To use the same fan speed for the entire build, add a single fan speed entry for layer 1.

You can edit the fan speed on a per layer basis. This is very similar to how the temperature tab works<sup>3</sup>. The fan speed will only change when there's a command to have it change. Therefore, if you set it to 60% at layer 1, the entire print will run at 60%. The most common recommendations are:

- PLA: 1:0 / 2:100 (no fan to start, then full fan after the first layer)
- ABS: 1:0 (no fan at all)

## Blip fan to full power when increasing from idle

Yes / No <blipFanToFullPower> [Blip]

This is useful for some fans that may have trouble getting up to speed at low voltages.

With small fans, there's very little torque. Meaning the fans can sometimes not generate enough force to get moving, (once they're moving they're fine, but getting started can be tough), this setting can help blip the voltage to the fan and get it running. For instance, if you have **Blip fan** on and your PLA settings are 1:0 / 2:60 (no fan to start, then 60% fan after the first layer) ... the G-Code would be written to turn the fan on at 100% first (higher torque blip), then turn the fan to 60%.

```
;layer 2
Fan on 100%
Delay .5 seconds
Fan on 60%
```

## Fan Overrides

### Increase fan speed (Switch)

Yes / No <increaseFanForCooling> [Incr]

Increases fan speed to help layers achieve adequate cooling.

### Increase fan speed for layers below \_\_\_ seconds

Seconds <minFanLayerTime> [ITime]

The fastest layer time we allow without modifying fan speed.

<sup>3</sup> When printing with multiple processes, the layer referenced is relative to where the process starts. For instance, if this process started at 5 mm (Advanced tab setting), then the layer 1 command would take place at the first layer after 5 mm.

### Maximum cooling fan speed

% <maxCoolingFanSpeed> [MxFSp]

Maximum fan speed that is allowed for layer cooling purposes.

This is similar to the speed overrides above, but you would just be controlling fan speed. I personally have not used this feature, as I typically like to print with either 0 or 100 % fan speed.

### Bridging fan speed override (Switch)

Yes / No <increaseFanForBridging> [Bridge]

Use a custom fan speed for all bridging regions.

### Bridging fan speed override

% <bridgingFanSpeed> [BrSpOvr]

The desired fan speed for all bridging regions.

For bridging it's important to get the material to solidify as fast as possible, setting a fan speed for bridging only can help accomplish this. For the rest of the bridging settings, see the [Other tab](#).

## G-Code Tab

---

### 5D firmware (include E-Dimensions)

Yes / No <use5D> [5D]

Most modern 3D printing firmwares include an explicit E-dimension to allow flowrate tweaking.

5D firmware means that your printer supports X, Y, Z, Speed, and Extrusion amounts. Almost every printer currently on the market does, so this should remain checked for 99% of printers.

### Relative extrusion distances

Yes / No <relativeEdistances> [RelDist]

Determines if relative extrusion values will be generated for each loop or if absolute values will be maintained throughout the entire print.

The options for G-Code creation are absolute or relative extrusion distances. For relative extrusion distances, each line would have an E-value that only applies for that line, whereas for absolute extrusion distances the E-values stack up.

Relative Extrusion:

```
G1 X## Y## E2.5
G1 X## Y## E2.5
G1 X## Y## E2.5
G1 X## Y## E2.5
```

Absolute Extrusion:

```
G1 X## Y## E2.5
G1 X## Y## E5
G1 X## Y## E7.5
G1 X## Y## E10
```

### Allow zeroing of extrusion distances (i.e. G92 E0)

Yes / No <allowEaxisZeroing> [AllowZ]

Typically enabled for all RepRap firmwares. May need to be disabled for some Makerbot firmwares.

When using Absolute extrusion mode, the printer keeps a running virtual position of the extruder. Using the command **G92 E0** can reset this virtual position to zero, this command is in most the starting scripts for RepRap machines prior to and after priming. Certain printers have firmware compatibility issues with this command and certain printers (relative extrusion machines) don't need **G92 E0** commands, so this command will vary from printer to printer.

## Use independent extruder axes

Yes / No <independentExtruderAxes> [Indep]

This determines if multiple extruders each have their own coordinate system. This options should be disabled for Marlin, Sprinter, and Repetier firmwares.

Certain firmwares (MakerBot/Sailfish) will track the extrusion rates of printers individually, whereas RepRap printers will use a single running tally (not independent) for extrusion rates.

## Include M101/M102/M103 Commands

Yes / No <includeM10123> [M101]

Legacy commands that are not typically used by most modern firmwares.

The tool-tip states that these are legacy commands and not used by modern printers. In looking up these commands this was what I found they used to do (in case you are interested):

```
M101 Extruder on, fwd
M102 Extruder on, reverse
M103 Extruder off
```

## Firmware supports “sticky” parameters

Yes / No <stickySupport> [Sticky]

Most G-Code firmware interpreters support what is known as “sticky” parameters, mean that the previous value for that parameter will be retained, even if it is not included in the next command.

Setting **sticky commands** is a great way of preventing much larger G-Code files. The way to think of sticky commands is something won't change unless it's told to change.

If sticky commands were not supported, changing the X value and extruding

```
G1 X10 Y10 E1 F100;
G1 X20 Y10 E2 F100;
G1 X30 Y10 E3 F100;
G1 X40 Y10 E4 F100;
```

With Sticky Commands:

```
G1 X10 Y10 E1 F100;
G1 X20 E2;
G1 X30 E3;
G1 X40 E4;
```

## Apply toolhead offsets to G-Code coordinates

Yes / No <applyGOffsets> [Offsets]

This option will shift all G-Code coordinates to account for toolhead offsets. This setting should only be enabled if the machine’s firmware does not support setting toolhead offsets.



## Global G-Code Offsets

If your prints are off-centered or too high off your build-plate the G-Code offsets are a great way to fix that. For instance, if your prints are 2 mm too high off your build plate, apply -2 mm in the Z-axis. If your prints are 10 mm to the right, place -10 mm in the X-axis. You will see these shifts in the G-Code previewer, but since these offsets are what works for your machine then that shouldn't be an issue!

### **Offset X-Axis**

mm <gcodeXoffset> [GOffsets X:]

X-axis offset applied to all coordinates in final G-Code file. If your printer homes at an X-coordinate of 100 mm for example, then set this offset to -100 mm.

### **Offset Y-Axis**

mm <gcodeYoffset> [GOffsets Y:]

Y-axis offset applied to all coordinates in final G-Code file. If your printer homes at a Y-coordinate of 100 mm for example, then set this offset to -100 mm.

### **Offset Z-Axis**

mm <gcodeZoffset> [GOffsets Z:]

Vertical (Z-axis) offset that is used to account for a slightly misaligned endstop positioning. A negative value will move the nozzle closer to the bed.

## Update Machine Definition

There are two ways to set the Machine Definition in Simplify3D:

- Under Tools → Options (Windows) or Simplify3D → Preferences (Mac) and
- under the FFF Settings window.

If you set the Machine Definition under the Options window and then find that it's being overwritten every time you click on your process, that would mean that this option is enabled and overwriting your build volume with different values.

### **Update Machine Definition**

Yes / No <overrideMachineDefinition> [UpMDef]

Overrides the current Machine Definition when loading this FFF profile. Allows easy switching between multiple printers with different machine settings.

### **Machine type**

Choice <machineTypeOverride> [MType]

The choices are Cartesian robot (rectangular volume) and Delta Robot (cylindrical build volume).

## Build volume

mm <strokeX/Y/Zoverride> [Build]

Override for X / Y / Z-axis max printing dimensions.

This is pretty straight forward ... the only time that it gets a bit tricky is with Delta style printers. For a delta printer, take your diameter and multiply it by 0.707 (Since Simplify3D measures the largest square that will fit in your build-plate), you will still get to use your entire build volume when printing, it just uses a different number when setting the build volume.

## Origin Offset

mm <originOffsetX/Y/Zoverride> [Orig]

The X / Y / Z position of the coordinate system origin. If the origin is in the center of your build platform, This value should be half the X / Y / Z-axis build dimension.

If your origin is at the bottom left of your printer (RepRap), it would be [0, 0, 0]. If your origin is in the center of your machine (MakerBot/Sailfish and Delta printers) you take your build volume # and divide by two in order to place the origin in the center of your build plate.

## Homing Dir

Min / Center / Max <homeX/Y/ZdirOverride> [Home]

Overrides the current Machine Dimensions when loading this FFF profile. Allows easy switching between multiple printers with different machine settings.

This setting doesn't change the print instructions, but just changes the G-Code previewer. You can set where your homing endstops are relative to your axis that way when Simplify3D has a G28 command (home all axis) it will know where the extruder gantry is.

## Flip Build Axis

Yes / No <flipX/Y/Zoverride> [Flip]

Flip the X/Y/Z-axis direction in virtual preview.

This is for the build area in Simplify3D. Generally, you can leave this at its default setting. If you need to flip one of the axis relative to your origin for your printer's configurations you can do so from here.

Note that the XML value 1 indicates No (not checked) and -1 indicated Yes (checked).

## Toolhead offsets

ToolID x XY offset <toolheadOffsets> [THeadOffsets]

A list of up to 6 toolheads (Tool 0 ... Tool 5), together with X and Y offsets for each. The format of this string in the .fff file is, for example:

```
<toolheadOffsets>0,0|0,0|0,0|0,0|0,0|0,0</toolheadOffsets>
```

## Update Firmware Configuration

Yes / No <overrideFirmwareConfiguration> [UpdFirm]

Overrides the firmware configuration when loading this FFF profile. Allows easy switching between multiple printers with different machine settings.

### Firmware Type

Choice <firmwareTypeOverride> [FType]

Select your firmware configuration.

The choices are:

- RepRap (Marlin/Repetier/Sprinter)
- MakerBot/Sailfish Firmware
- FlashForge Dreamer Firmware
- Bits from Bytes (BFB) Firmware

### GPX profile

Choice <GPXconfigOverride> [GPX]

Select your GPX machine profile (only applicable to MakerBot/Sailfish Firmware types).

If **Firmware type** is **MakerBot/Sailfish Firmware**, then the user may choose among:

- Cupcake Gen3 XYZ, Mk5/6 +Gen4 extruder
- Cupcake Gen4 XYZ, Mk5/6 +Gen4 extruder
- Cupcake Pololu XYZ, Mk5/6 +Gen4 extruder
- Cupcake Pololu XYZ, Mk5/6 +Polulu extruder
- TOM Mk6 – single extruder
- TOM Mk7 – single extruder
- TOM Mk7 – dual extruder
- Replicator 1 – single extruder
- Replicator 1 – dual extruder
- Replicator 2 (default config)
- Replicator 2 with HBP
- Replicator 2X
- Malyan M180
- 3Dison Multi
- 3Dison Pro and H600

### Baud rate

Bits/sec <baudRateOverride> [Baud]

## Discussion

***pinthenet – Wed Jan 11, 2017 4:17 am***

I'm puzzled how negative G-Code Offsets can be applied. I have a Vertex K8400 printer, which has end stops on each axis. When I home any axis, it moves the print head (X, Y) and/or print base (Z) until it hits the end stops. If that is so, how is it possible to apply a negative offset, e.g. for Z – the end stop prevents moving the build plate any closer to the nozzle.

Positive offsets are clear. Maybe I misunderstand the concept or the mechanics or FW, or is it just for certain printer types?

***brian442 – Wed Jan 11, 2017 4:37 am***

Your first layer is likely printed at  $Z = 0.2\text{mm}$  or something like that, so it's pretty easy to apply a  $-0.05\text{mm}$  Z-offset. That just shifts the Z position to  $0.15\text{mm}$  for the first layer.

If you try to apply larger offsets to where the Z-axis would actually go negative, the printer's endstops will prevent you from moving there, so you're already protected

***pinthenet – Wed Jan 11, 2017 8:04 am***

OK I think I understand, but I still think it won't work on the K8400.

To calibrate Z, I home the Z 'platform' then I raise/lower the the print plate relative to the platform so that it's  $0.2\text{mm}$  from the tip on all corners, but the Z platform is still at the end stop so I don't see how I can move it any closer using. Or am I barking up the wrong extruder?

# Scripts Tab

---

## Introduction

The Scripts tab is a powerful tool that allows you to set Starting Scripts, set G-Code for every Tool Change, set G-Code for every Layer Change and set G-Code for the end of your print.

If you're intimidated by G-Code, it's actually a lot easier than it looks, a great way of learning and testing is to use the Communications tab over USB connection and send your printer a line of G-Code at a time to see the responses. One thing to keep in mind is that different printers will accept different commands to accomplish the same task. This is just something you should keep in mind if your wondering why a certain G-Code command does not work with your printer.

## Basics of G-Code

G-Code is read line by line, the printer will accomplish one line of G-Code, then move on to the next. Always capitalize the letters in G-Code, spacing is important, and anything that comes after a semi-colon will not be read on that line. The two lines below are for home-all, and then move to 100, 100, 100 at a certain speed. The printer will home all 3 axis, then once that is done, move to the next line of G-Code.

```
G28; home all axis
G1 X100 Y100 Z100 F3600; Move to X100, Y100, Z100 at 3600 mm/min
(G-Code is natively in mm/min)
```

The link below is extremely useful, a glossary of G-Code commands for RepRap style printers

<http://reprap.org/wiki/G-code>

## Starting Script

Text <startingGcode> [StartingScript]

Custom G-Code that is included after temperature settings have been initialized. Note that all speeds must be entered in mm/min.

This is the G-Code that will be run after heating up but before your print. Traditionally, this will consist of homing the printer, potentially purging the nozzle, wiping the nozzle and running an auto-leveling function if the printer supports it. In addition, you can add PID values, or set toolhead offsets here if you'd like to.

If you're not a fan of the fact that Simplify3D heats up the bed/extruder, then runs your starting-script, you can customize this, by using the commands below. If you have the commands for [bed#\_temperature] or [extruder#\_temperature] in your starting G-Code script, the software won't add in heating commands, it will just run off of the commands in your starting script.

```
[fan_speed_percentage]
[fan_speed_pwm] – for example, M106 S[fan_speed_pwm]
[extruder0_temperature] – for example, M104 S[extruder0_temperature] T0; this
will take the layer 1 temperature for extruder 0
[extruder1_temperature]
```



```
...  
[bed0_temperature] – for example, M140 S[bed0_temperature] T0; this will take  
the layer 1 temperature for bed 0  
[bed1_temperature]  
...
```

## Layer Change Script

Text <layerChangeGcode> [LayerChangeScript]

Custom G-Code that is included between each layer changer after initial retraction has taken place. Note that all speeds must be entered in mm/min.

I personally haven't had to use this, but I'm sure that there are some excellent reasons/ideas to use for this. If you'd like for a G-Code script to be inserted in-between each layer, then you can simply place it in this tab. One interesting use of this, is for the FlashForge Dreamer, to have the lights blink in between each layer, however that can be a bit too much at times! The placeholders that are available for this tab are below:

```
[previous_Z_position]  
[current_Z_position]
```

## Retraction Script

Text <retractionGcode> [RetractionScript]

Custom G-Code that is included right before a retraction takes place. Note that all speeds must be entered in mm/min.

## Tool Change Script

Text <toolChangeGcode> [ToolChangeScript]

Custom G-Code that is included right before each tool change command takes place. Note that [old\_tool] and [new\_tool] placeholder variables can be used. Note that all speeds must be entered in mm/min.

This tab will insert G-Code on tool change commands. These commands can help do a ton of things for multiple extruder prints, especially if using the IF command functionality. For instance, if you wanted to change the temperature of your extruders (T0, and T1) for each tool change switch:

```
G28 X0 Y0; homes X-Y axis  
{IF NEWT00L=0}M104 S165 T1; set T1, inactive extruder to 165 C  
{IF NEWT00L=0}M109 S220 T0; Set T0, new active extruder to 220 and wait for it  
to reach temperature before proceeding.  
{IF NEWT00L=1}M109 S220 T1; set T1, Heat T1 to 220  
{IF NEWT00L=1}M104 S165 T0; Cool T0 to 165
```

If you were using one process (printed part with one extruder, support with the other), you would only need this script in your one process. However, if using multiple processes, you would need to copy+paste this into each process.

```
{IF OLDTOOL=0}G1 E-10 F1800 – this will only include the line that follows the  
{IF} brackets if the old tool (the one that was active prior to the tool  
change) is tool 0. You could use scripts like this to have different tool  
change retract distances for different tools  
{IF NEWTOOL=0}G1 E10 F1800 – similar to the above command, however it checks  
the new tool (the one that will be active after the tool change)  
[old_tool]  
[new_tool]  
[current_Z_position]
```

## Ending Script

Text <endingGcode> [EndingScript]

Custom G-Code that is included at the end of the build. Note that all speeds must be entered in mm/min.

Usually the Ending G-Code's purpose will be to get the nozzle off of the part and then turn off the heaters/motors. The Ending G-Code below is taken from the RoBo 3D profile:

```
M104 S0 ; turn off extruder  
M140 S0 ; turn off bed  
G28 X0 ; home X axis  
M84 ; disable motors
```

The placeholders available for the Ending G-Code: [current\_Z\_position]

## Post Processing

### Export file format

Choice <exportFileFormat> [ExpFmt]

The choices are:

- Standard G-Code (.gcode)
- Binary X3G File (.x3g)
- MakerBot 5<sup>th</sup> Generation (.makerbot)
- XYZprinting Da Vinci Type 1 (.3w)
- XYZprinting Da Vinci Type 2 (.3w)
- XYZprinting Da Vinci Type 3 (.3w)
- Dremel (.g2drem)
- Bits from Bytes (.bfb)

This was formerly controlled by two now-defunct checkboxes:

### ~~Create X3G file for MakerBot printers using GPX plugin~~

This is enabled for all of the MakerBot (Not 5th Gen MakerBot) / Sailfish style printers. The conversion will take your Machine Profile from the X3G tab of the Firmware Configuration and use that to convert your G-Code file to X3G. When you click Save Toolpaths to disk after slicing, the X3G file is automatically created when you save a .Gcode file.

### ~~Create .MakerBot file for 5th generation printers~~

Converts the G-Code file into .MakerBot file. Similar to the X3G, when you click Save Toolpaths to disk after slicing, the .MakerBot file is automatically created when you save a .Gcode file.

### Add celebration (Switch)

Checkbox <celebration> [Celeb]

Add song to end of build. Using Random always keeps things new when printing.

### Add celebration at end of build (for .x3g files only)

Choice <celebrationSong> [Song]

If Export file format is set to Binary X3G File (.x3g), the choices are:

- Random Song
- Indiana
- Take On Me
- Entertainer
- Looney Tunes
- Star Wars
- Funky Town

### Additional Terminal Commands for Post Processing

If you are using a custom GPX ini file, this is a very important section, as you will need to reference GPX in this section rather than use the built-in functionality. There's also a ton of post-processing commands below that help for those who are working with custom firmwares, or doing projects with their printers that require more functionality.

Using the Post Processing box would be if using a MakerBot/Sailfish style printer that doesn't have traditional steps-per-mm values, you would need to do the following:

- 1) Use the Machine ini example to set a text document ini with your machine information:  
<https://github.com/whpthomas/GPX/blob/master/examples/example-machine.ini>
- 2) Save the file as GPX.ini, place it in the Simplify3D installation folder
- 3) Place the following in the post-processing box: GPX [output\_filepath]

Below are the post-processing tools that also work in the post-processing box:

{REPLACE "E" "A"} – search and replace for the text within quotes, in this example every "E" character would be replaced with an "A" character

{PREPEND "G92 E0\n"} – prepends the specified text at the very beginning of the G-Code file, note that the \n is converted into a true newline character, not two separate "\" and "n" characters

{APPEND "G28 X0 Y0\n"} – appends the specified text to the very end of the G-Code file

{DELETE "M82\n"} – deletes every occurrence of the specified text from the G-Code file, note that it will not automatically delete a line if it is suddenly empty after the deletion, so that is why you might want to include the \n at the end (so that the empty line is also removed)

{STRIP ";"} – completely deletes every line in the G-Code file that begins with the specified text

{TOOL0REPLACE "E" "A"} and {TOOL1REPLACE "E" "B"} – these special TOOL#REPLACE commands will do a search and replace, very similar to the {REPLACE} command, however, the replace only occurs if the specified tool is active. For example, when using TOOL1REPLACE, the replacement will only occur if tool 1 was currently active at that line of the .gcode file.

## Discussion

*ghiom – Fri May 15, 2015 8:54 am*

Hello, my goal is to mix color.

My idea is to change the tool at every layer.

By this way, when your are not to close of the printerd object , it's seems colors mixed from the 2 colors.

Layer change G-Code tab is only able to add a line between previous and current Z position.

So I managed to do my effect like this :

In the layer change G-Code tab I put this code:

[previous\_Z\_position]

T1

[current\_Z\_position]

then, in a text editor ( windows notpad ) i'm searching all the T1 ( at layer change ) and replace T1 by T0 one time in 2.

It's a big job to do it by hand in notpad when you have a lot of layers.

did you know a text editor which is able to automate a replace one time in two?

Because it works! One layer on two are different colors!

*blaknite7 – Mon May 25, 2015 1:25 pm*

This is an awesome post! I did have a couple questions though:

I've been working a lot with dual head prints and I am trying the tweak in the settings ...With regard to the tool change G-Code and reducing temperatures of the inactive extruder, is it not possible to reference the extruder temperature variable and use it in the tool change G-Code?

For example, I mean this:

From:

```
G1 X0 Y40 F4000 ; move to wait for temperatures
{IF NEWTOOL=0}M104 S165 T1; set T1, inactive extruder to 165 C
{IF NEWTOOL=0}M109 S205 T0; Set T0, new active extruder to T0 Temperature and
wait for it to reach temperature before proceeding.
{IF NEWTOOL=1}M104 S165 T0; Cool T0 to 165
{IF NEWTOOL=1}M109 S205 T1; set T1, Heat T1 to T1 Temperature
```

To:

```
G1 X0 Y40 F4000 ; move to wait for temperatures
{IF NEWTOOL=0}M104 S165 T1; set T1, inactive extruder to 165 C
{IF NEWTOOL=0}M109 S[extruder0_temperature] T0; Set T0, new active extruder to
T0 Temperature and wait for it to reach temperature before proceeding.
{IF NEWTOOL=1}M104 S165 T0; Cool T0 to 165
{IF NEWTOOL=1}M109 S[extruder1_temperature] T1; set T1, Heat T1 to T1
Temperature
```

The reason for doing this is to have different temperatures at different layers that are controlled by the temperature tab in the settings ... It didn't seem to work for me when I tried it so I was curious if there is another way to reference the temperature/layer tables defined?

Also, there seems to be a slight pause when switching tools prior to the move "G1" code in the tool change script. is there a way to minimize this/eliminate this? Normally it's not so bad but if you have a small feature you are trying to print the dwell tends to screw up the finish.

Thanks for the input. Hopefully someone figured this out already.

Advice for those that want to use this functionality, you may want to change the order from the original post of what tools are activated and when. if you command it to wait for temperature (M109) first and then set the inactive extruder to a lower temperature it will operate in that order. Its more often best to always command the inactive extruder to the lower temp first and then the active extruder to the target temp/stabilize. This prevents extra ooze from the soon to be inactive extruder while the new tool head is warming up.

**Kyuubi – Tue Jun 16, 2015 11:39 am**

I want post too the LCD "(current layer) of (layers)" on every layer change, is that possible?

**KeyboardWarrior – Tue Jun 16, 2015 12:28 pm**

Kyuubi wrote: I want post too the LCD "(current layer) of (layers)" on every layer change, is that possible?

Since there isn't a current layer or # of layers variable, this isn't currently doable. If you're looking to do this for time-measurement reasons, I don't think that # of layers is a great indicator of speed by the way, since some layers will take much longer than others.

For instance, a pyramids base layers will print much slower than the layers towards the tip.



### **ShaqFoo – Tue Aug 04, 2015 12:13 am**

KeyboardWarrior wrote:

Kyuubi wrote:I want post too the LCD "(current layer) of (layers)" on every layer change, is that possible?

Since there isn't a current layer or # of layers variable, this isn't currently doable. If you're looking to do this for time-measurement reasons, I don't think that # of layers is a great indicator of speed by the way, since some layers will take much longer than others.

For instance, a pyramids base layers will print much slower than the layers towards the tip.

Great post but he never asked about time measurement. The single biggest reason to display the layer height on the LCD screen is in the case of failed print so you know where to resume by LAYER #. Yes, you could do it by z height too, but visually seeing the layer number is easier to remember and find in the G-Code file that S3D creates. S3D inserts the layer# and z height in the G-Code for every layer printed but it puts inserts them as comments.

After reading the post, I was really intrigued by the terminal post processing window and was able to display both the layer # and the Z height on the LCD screen during the print for every layer printed. See attached pictures.

Place the following code in the post processing window and you will get layer number printed on your LCD screen along with the z height. The replace command simply removes the comment and replaces it with the M117 command. The second line is for formatting only it is not needed but tightens things up.

```
{REPLACE "; layer" "M117 Layer"}  
{REPLACE " Z = " " Z="}
```

### **nka – Wed Aug 05, 2015 7:16 am**

Can I do calculation, like something like this to raise the nozzle 10mm at the end of a print?

```
G0 X0 Y0 Z{[current_Z_layer]+10}
```

### **CompoundCarl – Wed Aug 05, 2015 7:42 am**

nka wrote:Can I do calculation, like something like this to raise the nozzle 10mm at the end of a print?

Just use relative mode. See below.

```
G90 ; change to relative mode  
G1 Z10 F3000 ; raise nozzle 10mm in Z-axis  
G91 ; switch back to absolute mode
```

It's the same thing that S3D uses with their jog controls.

## Speeds Tab

(CG Note that there was no initial documentation for this in the ToD posts, probably because the Speeds Tab did not exist at that time ... I'm pulling documentation from other tabs and other sources, as appropriate.)

### Speeds Section

#### Default Printing Speed

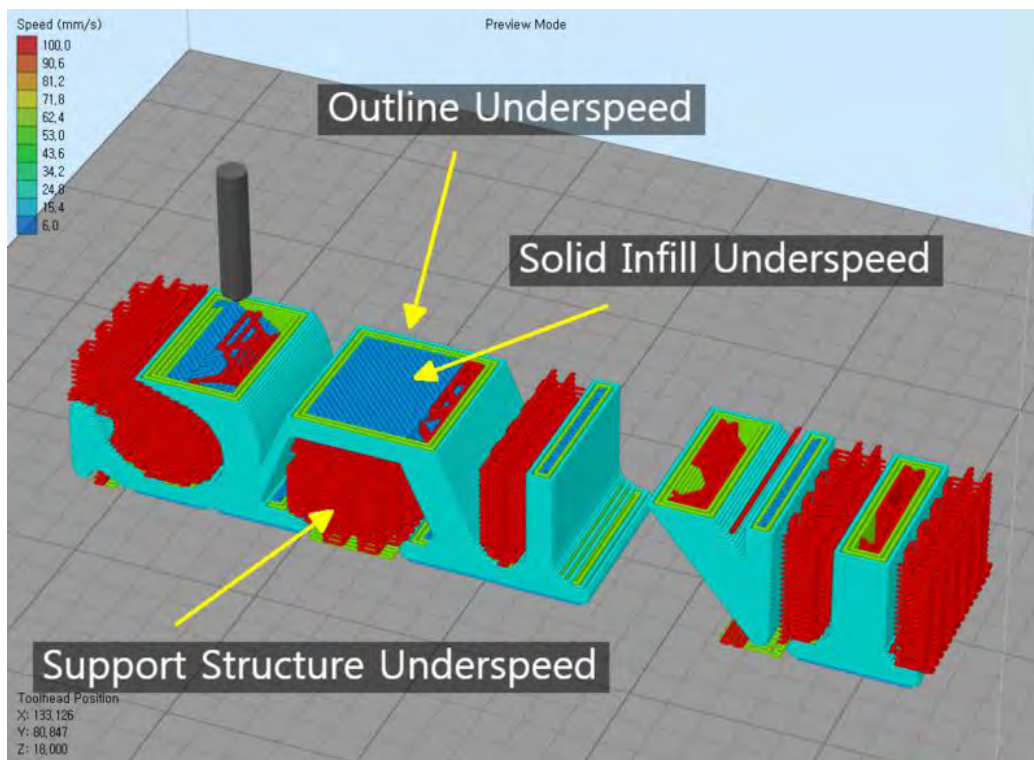
mm/min <defaultSpeed> [SpDefault]

Initial speed used for all printing movements (modifications may be added for cooling or outline underspeed).

This is the speed at which the Infill of your part will be printed, assuming the Cooling Overrides don't adjust it and that it's not a 100% infill layer.

### Underspeed Settings

I think this image from the Naver blog posts captures the important aspects of the next three parameters:



Simplify 3D can adjust the speed for each output area, giving you optimum speed and quality.

The usual recommendation is that:

- \* the outer wall is set to  $\frac{1}{2}$  of the default speed,
- \* solid infill is set to  $\frac{2}{3}$  of the default speed, and
- \* support structures are set to  $\frac{1}{3}$  of the default speed.
- \* The movement speed should be the same as the default speed and
- \* the Z-axis speed should match the retraction speed.

## Outline Underspeed

% <outlineUnderspeed> [SpPerim]

Modifies printing speed for outer-most outline segment (allows for higher-quality extrusions).

For your perimeters, you print at a % of your **Default Printing Speed**, this is the %. For instance, if you want to print your Perimeters at 75% of your **Default Print Speed**.

## Solid Infill Underspeed

% <solidInfillUnderspeed> [SpSolidIn]

Modifies printing speed for the top and bottom solid layers (used to improve exterior surface finish).

For any layer that uses 100% infill, including **Bottom Solid Layers**, **Top Solid Layers** and if you have 100% infill set in your part, you print at this % of your **Default Printing Speed**.

## Support Structure Underspeed

% <supportUnderspeed> [SpSupp]

The support material will be printed at a percentage of your **Default Printing Speed**.

## X/Y Axis Movement Speed

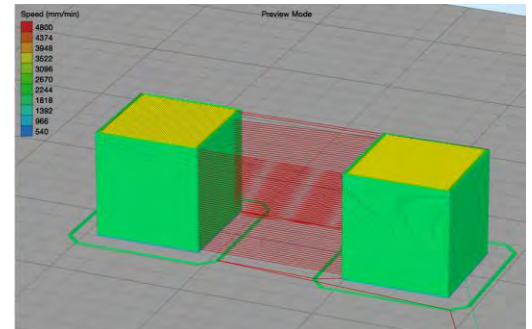
mm/min <rapidXYspeed> [SpXY]

Rapid movement speed for X/Y axes when machine is not printing.

This is the speed your printer will move when your printer is not printing in the X–Y axis. When looking at the G-Code previewer, you can see these lines as the Travel Moves.

Tomas Sanladerer believes that:

*“Travel moves should always be as fast as your printer can handle, because it’s going to give the hotend as little of a chance of oozing and going out of the intended plastic flow situation as possible if that makes sense. Your printer’s firmware should know how fast it can reliably go, so you can actually set the travel speed on your slicer to some obscene value and let your printer handle the rest.”*



<https://toms3d.org/2018/02/05/things-know-petg/> and  
[https://www.youtube.com/watch?v=8\\_adY2K-YIc](https://www.youtube.com/watch?v=8_adY2K-YIc)

Also, higher values will reduce any tendencies of stringing.

## Z Axis Movement Speed

mm/min <rapidZspeed> [SpZ]

Rapid movement speed for Z-axis when machine is not printing. Should match actual Z-axis movement speed between layers for accurate print times.

The speed at which your Z-axis will move.

## Speed Overrides

### Adjust printing speed (Switch)

Yes / No <adjustSpeedForCooling> [AdjBelow]

Slows printing speed to give layers adequate printing time.

### Adjust printing speed for layers below \_\_\_\_ seconds

Seconds <minSpeedLayerTime> [AdjBelowSec]

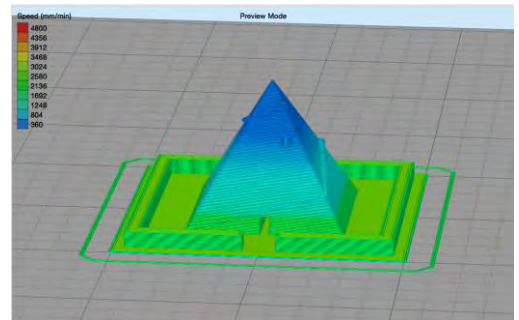
The fastest layer time we allow without modifying speed.

### Allow speed reductions down to \_\_\_\_ %

% <minCoolingSpeedSlowdown> [AdjDown]

Minimum speed reduction that is allowed for cooling purposes.

This setting takes effect for layers that would otherwise print too quickly. For instance, if you printing a pyramid as seen below, you can see that the print speed lowers as the layers get smaller. You can see from the legend on the side, that the dark blue is the slowest speed, it's because the **Cooling Speed Overrides** are slowing this layer down dramatically because it would take far less than 15 seconds per layer.



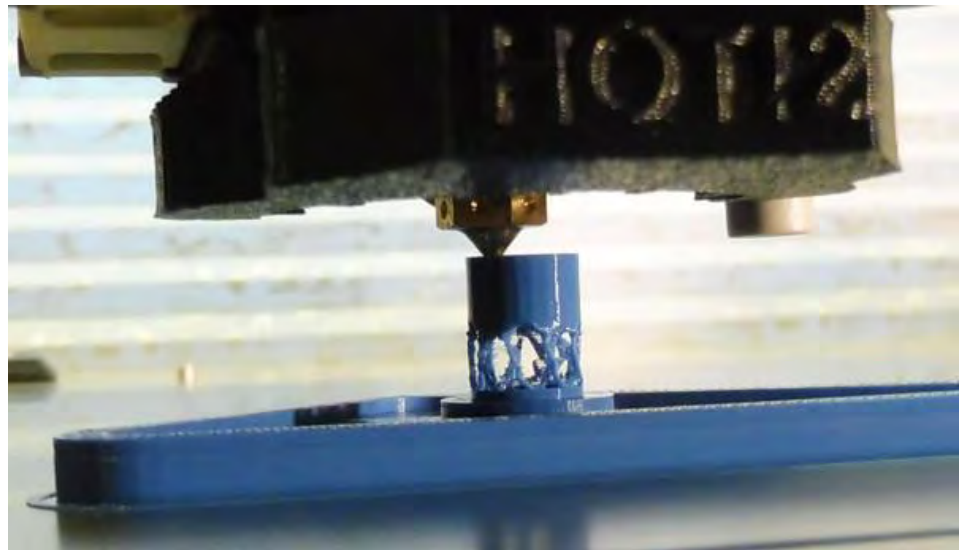
## Bulge Jam

While printing the Filament Spool components, I encountered this problem 4 times in a row on the thin vertical element of the print. Symptoms included severe under-extrusion and a loud “clicking” sound from the extruder. The filament was almost completely jammed, and the gears were slipping on the filament.

The jams were cleared each time with the acupuncture needle that came with Pi3Mk3 and opening the gear compartment and cutting out the bulging filament.

This post on the Prusa forum on 4/15/2018 explains my findings on this issue:

<https://shop.prusa3d.com/forum/others-archive--f66/clicking-printhead-filament-stuck--t14916-s40.html#p77927>



**Bottom line: I was printing very slowly.** The filament softened further up the PTFE tube than usual and, on a retraction, that soft material was above the tube. On the next forward extrusion, the filament fattened enough to prevent passage back down the tube. I'll dub this a *"Bulge Jam"*.

How I got to this slow print mode was a combination of the CSG model design ("Simple Smooth Spool Holder ! Updated !" by John Pfeiffer – <https://www.thingiverse.com/thing:989957>) and the cascading slowdown parameters of Simplify3D 4.0.1 (S3D).

The model design has a tall, hollow tube that slices as 2 perimeters with no infill. The speed of printing this hollow tube seems to be controlled by the S3D parameters:

- A. Speeds → Default Printing Speed (speed)
- B. Speeds → Outline Underspeed (percentage)
- C. Speeds → Speed Overrides → Adjust Printing Speed (checkbox)
- D. Speeds → Speed Overrides → ... for layers below (time)
- E. Speeds → Speed Overrides → Allow speed reductions down to (percentage)

If C (Adjust Printing Speed) is checked, then the actual printing speed for the hollow tube (which takes longer to print than D) is  $A \times B \times E$ .

*S3D scales the Default Printing Speed by both the Outline Underspeed and the Allow Speed Reductions Down To settings.*

The default settings in the Prusa i3 Mk3 profile currently provided by Simplify are (A) 4,800 mm/min (80 mm/sec), (B) 50%, and (E) 20%. I lowered the default print speed for better rendition of the lower part of the model to 2,400 mm/min, so  $A \times B \times E$  worked out to 240 mm/min or 4 mm/sec. This triggered the Bulge Jam.

**Changing E (Allow speed reductions down to) from 20% to 40% cured the issue.**

Personally, I think that users of S3D would benefit from a GUI that allowed (provided the option of) setting of speeds directly, rather than in percentages ... but that's another issue. We wind up back-figuring the speeds by trial and error, rather than by having direct control.



## Other Tab

### Bridging Section

#### Unsupported area threshold

mm<sup>2</sup> <minBridgingArea> [Area]

Bridging calculations will only be applied to unsupported areas greater than this amount.

Any area larger than this that is unsupported will have the bridging multipliers applied to the infill in the bridging.

#### Extra Inflation Distance

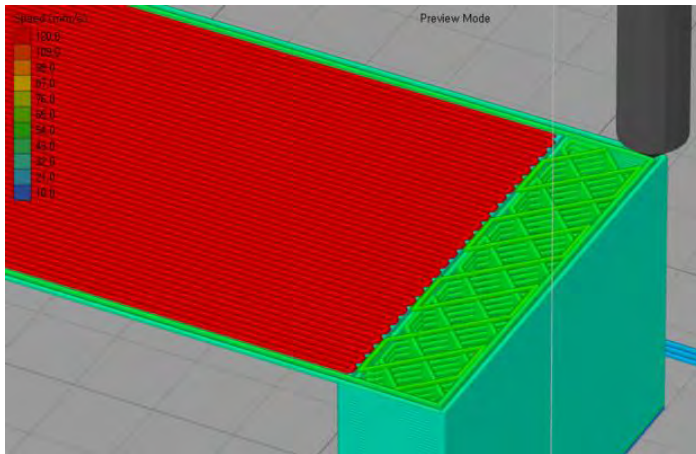
mm <bridgingExtraInflation> [ExInflat]

Expands the bridging region so that it has a larger overlap with the layer window. Set to zero to disable.

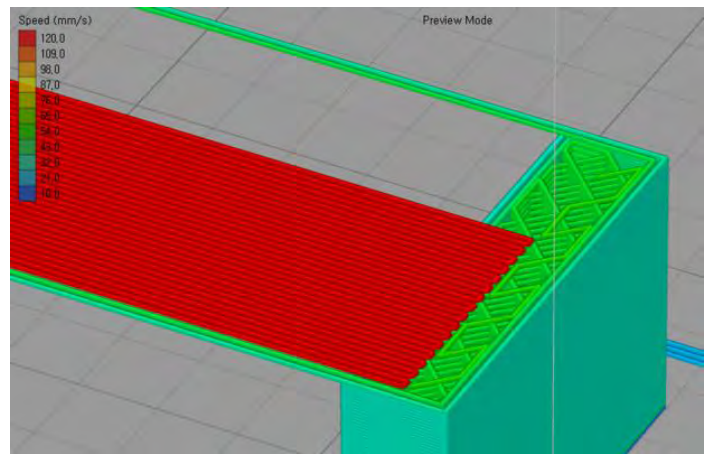
I believe this was formerly called *Bridging Inflation Distance*

This setting is used to extend the bridge further over the previously printed material. This allows for better adhesion. Example: Say you want to print <http://www.thingiverse.com/thing:179261> in an upright position. The bridging inflation distance can be used to add some more adhesion when the large bridged section occurs.

I think these edited images from the Naver blog posts highlight the effect of this parameter:



Extra inflation distance: 0 mm



Extra inflation distance: 3 mm

#### Bridging extrusion multiplier

% <bridgingExtrusionMultiplier> [BrExMult]

Lowering the extrusion multiplier for bridges can help stretch filament and avoid drooping.

If you find that extruding a lot more/less material helps with bridging, then you would set that here.

## Bridging speed multiplier

% <bridgingSpeedMultiplier> [BrSpMult]

Modify the speed multiplier for bridging sections.

If you find that printing faster/slower helps with bridging, then you would set that here. One trick I use occasionally, is I'll temporarily set the bridging speed multiplier to 999, then slice my file, that way the areas in which the bridging multiplier is taking place are bright red and stand out in the G-Code previewer, that way I have a better idea of where the bridging settings are occurring.

## Use fixed bridging infill angle (Switch)

Yes / No <useFixedBridgingAngle> [FixedAngle]

Enable this option to force all bridging infill to be printed in the same direction.

## Use fixed bridging infill angle (Value)

Degrees <fixedBridgingAngle> [FixedAngle]

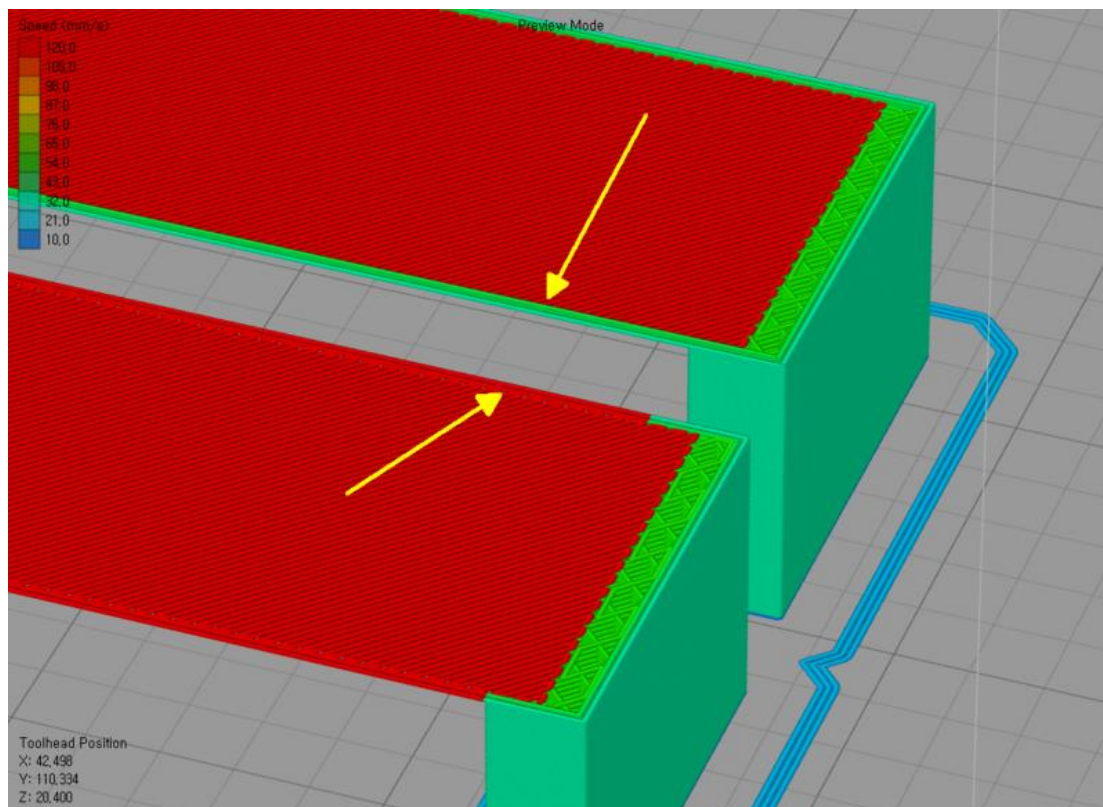
Enter the angle to use for all bridging infill.

## Apply bridging settings to perimeters

Yes / No <applyBridgingToPerimeters> [BrPerim]

If this option is enabled, bridging settings will be applied to any perimeter sections that cross a bridging region.

I think this image from the Naver blog posts highlights the effect of this parameter:



When **Apply bridging settings to perimeters** is checked, the first exterior wall is changed to bridge speed.



## Dimensional Adjustments

### Horizontal Size Compensation

mm <horizontalSizeCompensation> [HComp]

A negative value will inset (shrink) your model outline in the X-Y plane. Useful to account for small dimensional differences in final print quality. Set to zero to disable.

Information from <https://forum.simplify3d.com/viewtopic.php?t=2042>.

The horizontal size compensation is a new feature in version 2.2.2, if you don't see this option under the Other tab, this may mean you are using version 2.2.1 or earlier.

There are many, many usage case scenarios for which you could use this feature. Taken from the blog post, the gear bearings by Emmet (<http://www.thingiverse.com/thing:53451>).

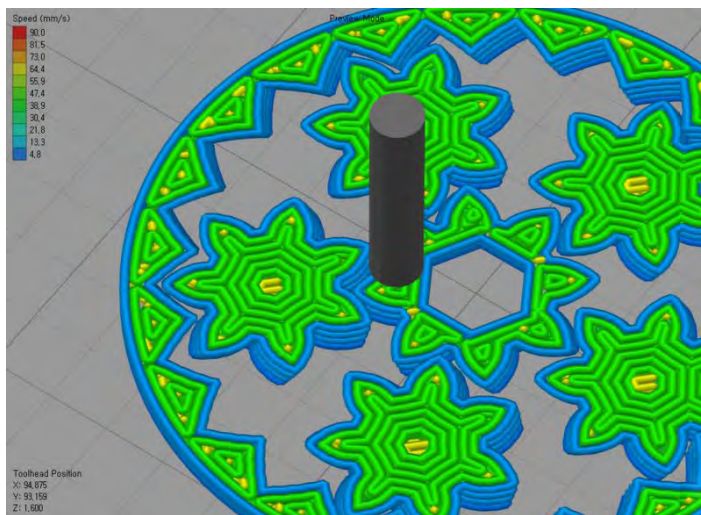
If you print these out, and the plastic has molded together, you could place a value of  $-0.05$  to increase the gap by  $0.1$  mm, or  $-0.1$  mm gap to increase the gap by  $0.2$  mm (half the average nozzle diameter for perspective). The gap is doubled, since both perimeters are compensating away from one another, so the gap is twice the value of the input.

#### Modify tolerances without changing the 3D model

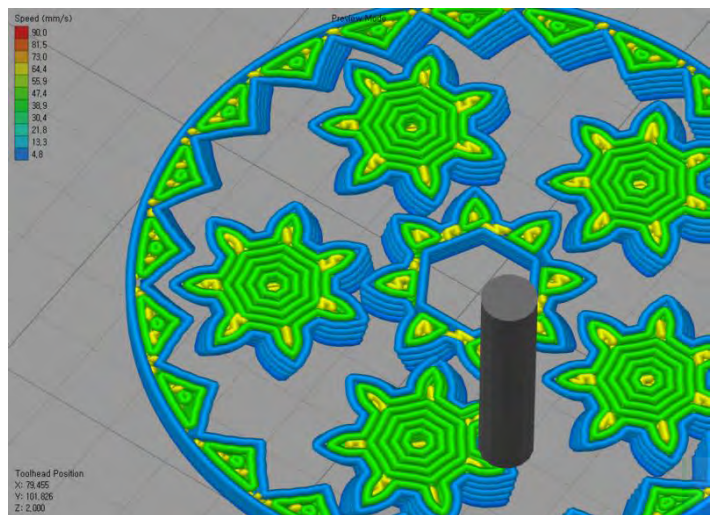


That's one use-case scenario, but there's also the use case for a standard print. For instance, if you print a 25 mm cube and notice that it's 24.8 mm in the X-Y axis, then you print a 50 mm cube and it's 49.8 mm, you could add  $0.1$  mm to the size compensation to quickly remedy the size of your prints.

I think these images from the Naver blog posts highlight the effect of this parameter:



Horizontal size compensation: 0 mm



Horizontal size compensation:  $-0.1$  mm

## Filament Properties

### Filament Toolhead Index

Index <filament...> [F-List]

Select the toolhead index for the filament properties below.

### Filament Diameter

mm <filamentDiameters> [FilDiam]

Measured raw filament diameter.

The diameter of filament your printer uses. This is very important for extrusion values. For instance, if you use 3 mm filament but have 1.75 mm filament in your setting, the software will be turning your extruder motor *far* too much since it will think you are using only 1.75 mm diameter filament. Small tweaks to this would be the same as small tweaks to your extrusion multiplier.

### Filament Price

Price / kg <filamentPricesPerKg> [FilPrice]

Allows you to customize the purchase price of your filament for accurate cost calculations.

Price that you paid for your filament per kg, so that when you slice your file in the G-Code preview you can see the estimated cost.

### Filament Density

Grams / cm<sup>3</sup> <filamentDensities> [FilDen]

Allows you to customize the density of your filament for accurate weight calculations.

## Tool Change Retraction

### Tool change retraction distance

mm <toolChangeRetractionDistance> [ChgRetDist]

How much plastic to pull back into the nozzle after a tool change (in terms of raw filament).

When changing tools, a retract of \_\_ mm will occur. The idea is that instead of a standard small retract, you can retract the filament far up the extruder to prevent any oozing while the other tool is printing.

### Tool change extra restart distance

mm <toolChangeExtraRestartDistance> [ChResDist]

Extra extrusion distance on top of initial tool change retraction amount. Negative values are allowed (in terms of raw filament).

When priming the extruder that was previously retracted, this extra restart distance will be added to that value. For instance, if you have a tool change retraction of 12 mm, and an extra restart of -0.5 mm, the following will occur:

- Print with Right extruder
- Tool Change Retract Right Extruder (12 mm)

- Prime Left Extruder (12 mm + (-.5 mm)) = 11.5 mm prime
- Print with Left Extruder

### *Tool change retraction speed*

Mm/min <toolChangeRetractionSpeed> [ChRetSp]

Extruder speed for the tool change retraction movements. May need to be significantly lower than standard retraction speed if using higher retraction distance.

The speed at which the retraction and primes will occur for the tool change retract/primes. I've found since you are moving much more filament, moving slower is very important. Trying to retract 12 mm of filament at F4800 may cause a lot of skipping which leads to other issues.



## Advanced Tab

### Layer Modifications Section



The next four options (two switches and two values) are used primarily if you want to print with different settings in one part. For instance, if printing the Statue of Liberty below, you may want to print the base of the statue at a much thicker layer height than the statue itself.

#### Two Process Statue of Liberty Example:

The first process would stop at 50 mm (assuming 50 mm is where the base ends) and the second process would have a starting point of 50 mm so that the second process print settings would be applied to everything after 50 mm.

See the tutorial on printing different settings for different regions at <http://www.simplify3d.com/support/tutorials/different-settings-for-different-regions-of-a-model/>

#### Start printing at height (Switch)

Yes / No <useMinPrintHeight> [Start]

Process will not print below the specified height.

#### Start printing at height (Value)

mm <minPrintHeight> [Start]

Process will begin printing at this height.

All models in this process will start printing at \_\_\_\_ Height (mm)

#### Stop printing at height (Switch)

Yes / No <useMaxPrintHeight> [Stop]

Process will not print above the specified height.

#### Stop printing at height (Value)

mm <maxPrintHeight> [Stop]

Process will stop printing at this height.

All models in this process will stop printing at \_\_\_\_ Height (mm)

## Thin Wall Behavior

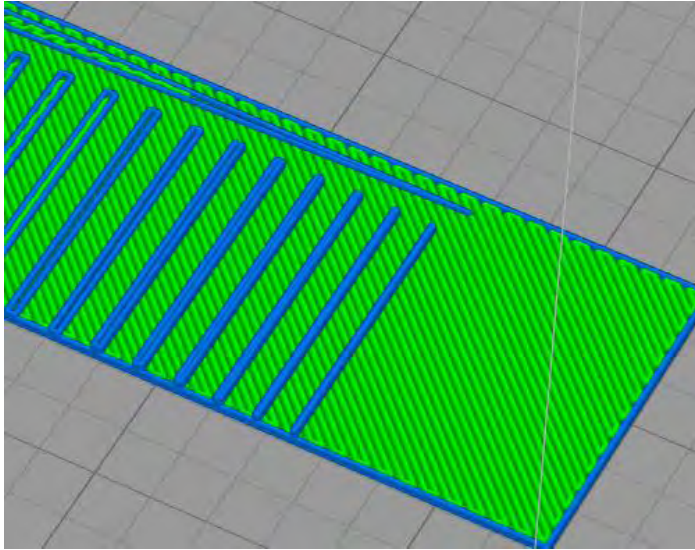
### External Thin Wall Type

Choice <externalThinWallType> [ExThinType]

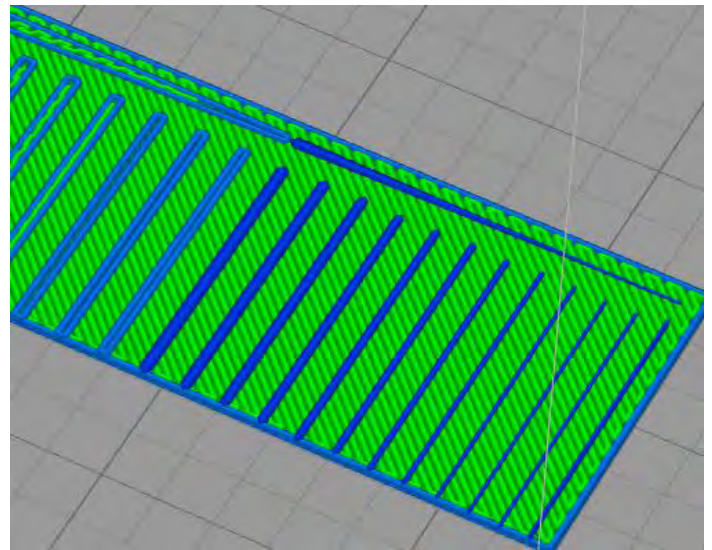
Choose how external thin walls on the outside of the part should be printed.

The choices are **Perimeters only** and **Allow single extrusion walls**.

I think these images from the Naver blog posts highlight the effect of this parameter:



Perimeters only



Allow single extrusion walls

S3D forum user jhaupt wrote on 1/2/2018:

I just discovered the **Allow Single Extrusion Fill** option under **Thin Wall Behavior**, which has reduced build time significantly on some large prints. However, the quality of this kind of infill is such that it can look quite bad on top (external) layers. It would be very nice to be able to turn this on for all the internal layers, but revert to the standard rectilinear/raster pattern infill for any top surfaces.

For anyone who hasn't tried this option, it's for cases where the walls are thin enough that the infill could be made with just a single extrusion along the wall's length instead of many minute raster movements, which can really kill the build time.

### ***Only use perimeters for thin walls***

This info from the original ToD documentation probably relates to a checkbox option that appeared in earlier version of S3D.

If printing a very thin wall, this means that the software will not add in Infill into very small crevices. If there's room to fit multiple perimeters, it will print in multiple perimeters, but otherwise you'd be left with a small gap.

Choose how internal thin wall gaps should be filled within the model.

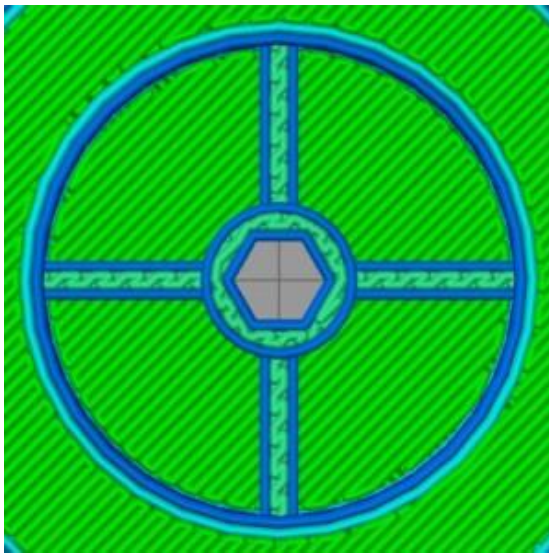
The choices are **Perimeters only**, **Allow single extrusion fill**, and **Allow gap fill**.

Some of the documentation in this section comes from the *Printing Thin Walls and Small Features* article at <https://www.simplify3d.com/support/articles/printing-thin-walls-and-small-features/>.

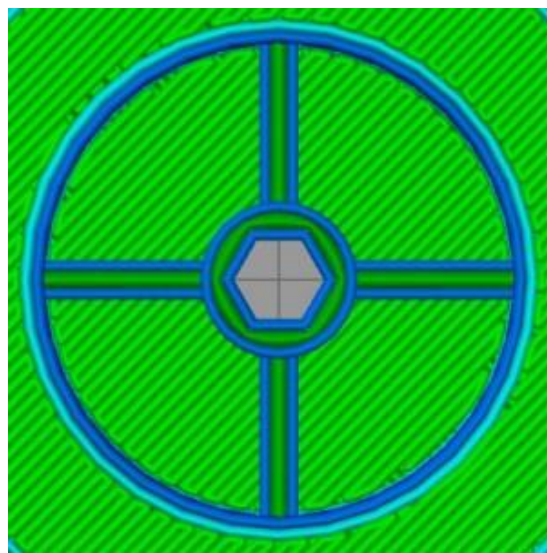
The two images below show an important use of this parameter. The key areas are the four spokes that connect the central hub to the rim. These spokes are about 1.4 mm wide. When they are printed with a 0.4 mm **Extrusion Width**, you will have one 0.4 mm perimeter along each side of the spoke and a 0.6 mm wide gap between those perimeters.

If **Internal Thin Wall Type** is set to **Allow gap fill** (the default), S3D fills the space between the perimeters with an infill similar to the normal infill on the interior of your model. The extruder moves in a back-and-forth pattern, extruding lines of plastic that connect both perimeters together. This can be useful since it creates many structural connections between these perimeters. However, it may be preferable to fill those gaps in a single movement.

If **Internal Thin Wall Type** is changed to **Allow single extrusion fill**, the gap between these spokes is now filled with a single extrusion – the darker green extrusions in the spokes and around the central hub. The extrusion is adjusted by S3D to fill the 0.6 mm gap:



Allow gap fill



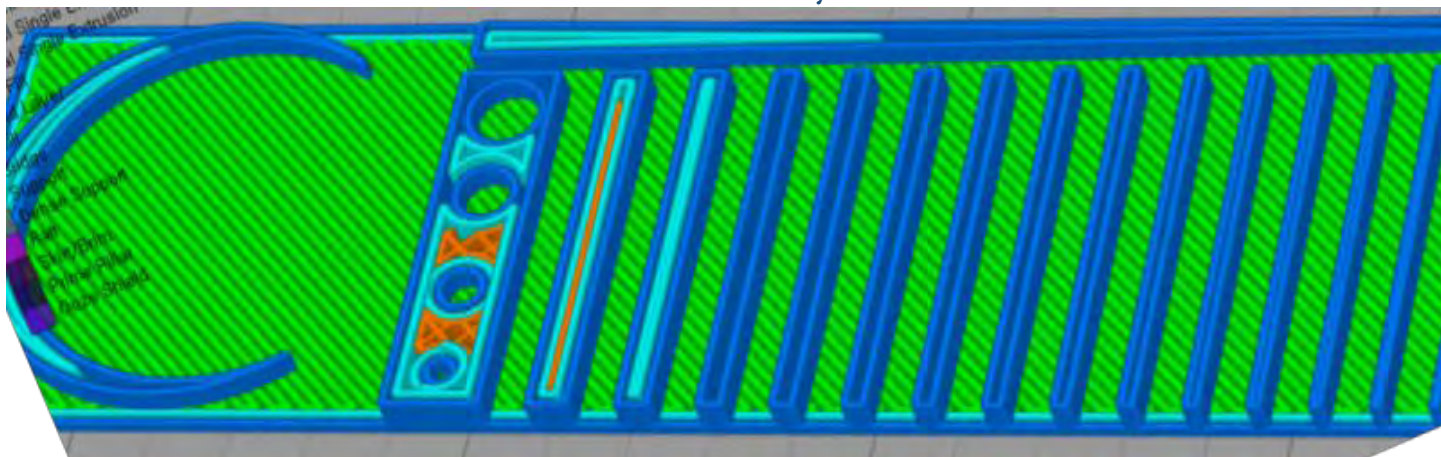
Allow single extrusion fill

Using single extrusions allows the printer to fill these gaps in a single pass instead of a back-and-forth pattern. This can improve your printing time and produce a better surface finish. The single extrusion around the central hub is printed in a single continuous loop, with the thickness of the extrusion dynamically adjusted along the loop to fill the varying gap between the perimeters.

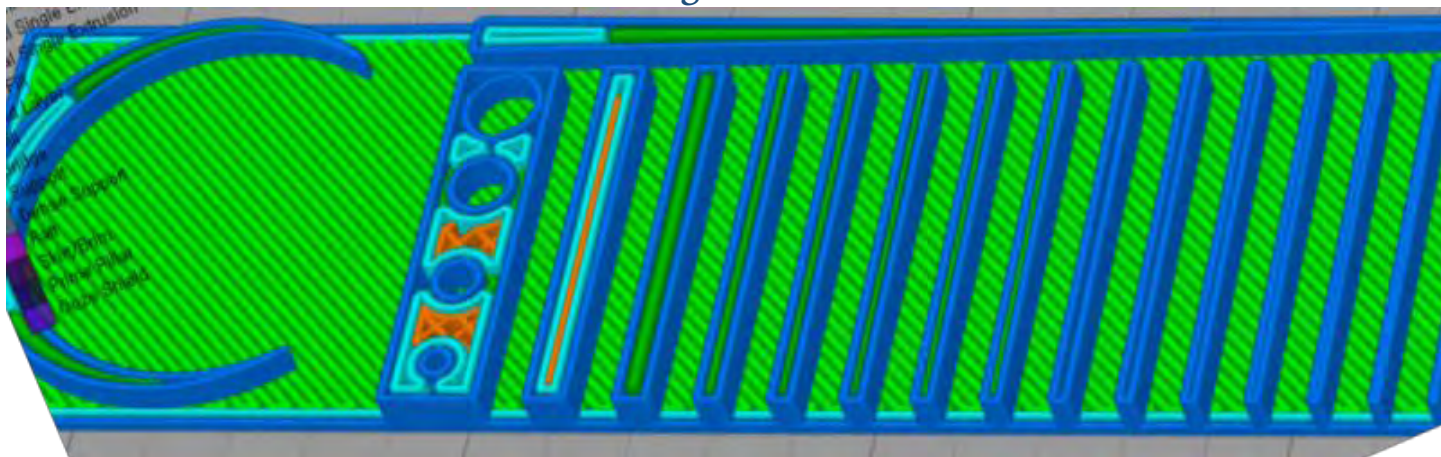


I think these three edited images from the Naver blog posts highlight the effect of this parameter:

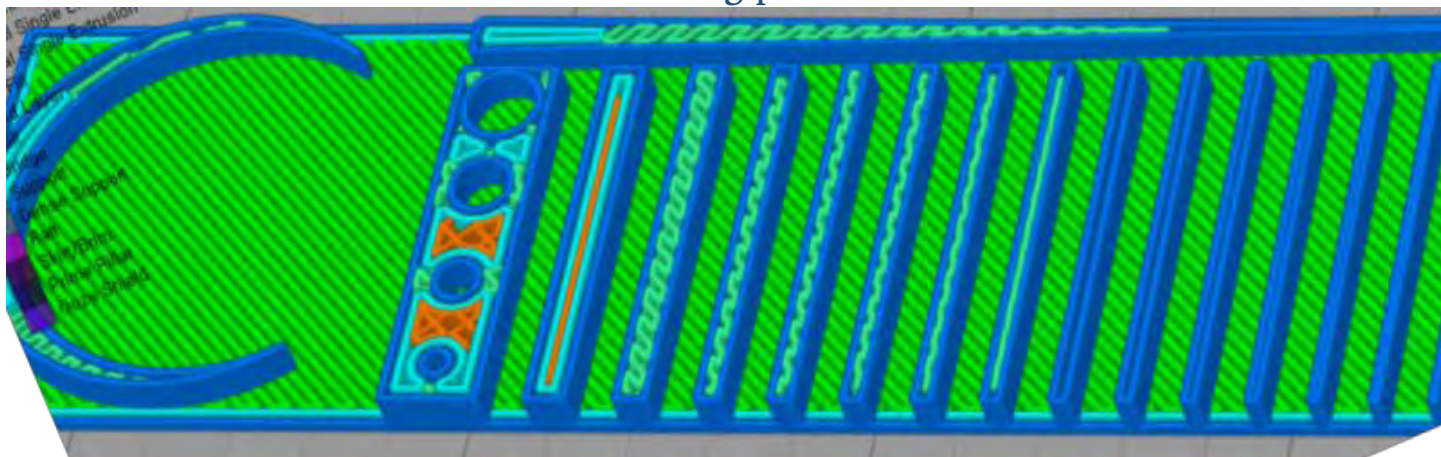
Perimeters only



Allow single extrusion fill



Allow gap fill



### ~~Allow gap fill when necessary~~

This info from the original ToD documentation probably relates to a checkbox option that appeared in earlier version of S3D.

This setting will work to add in another perimeter by allowing as long as it's within the overlap setting. Otherwise, it will fill the area with Infill.

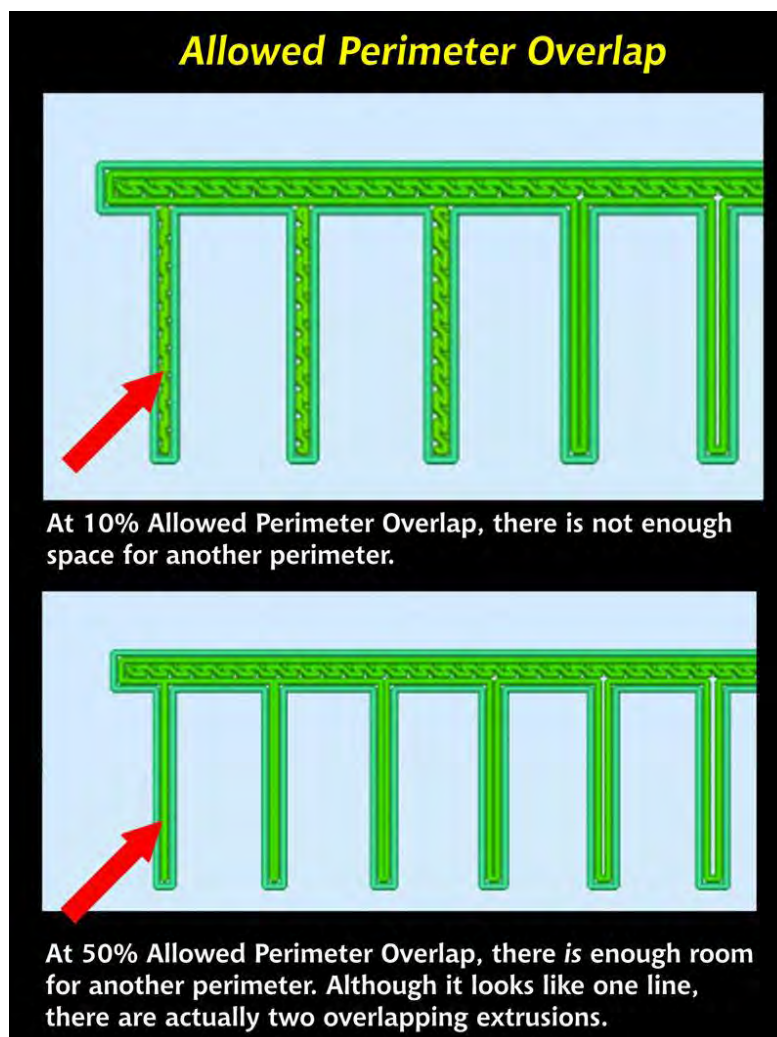
### Allowed perimeter overlap

% <thinWallAllowedOverlapPercentage> [Overlap]

The allowed overlap for perimeters inside a thin wall. Larger values will define a higher preference for using perimeters inside a thin wall versus gap fill or single extrusion fill.

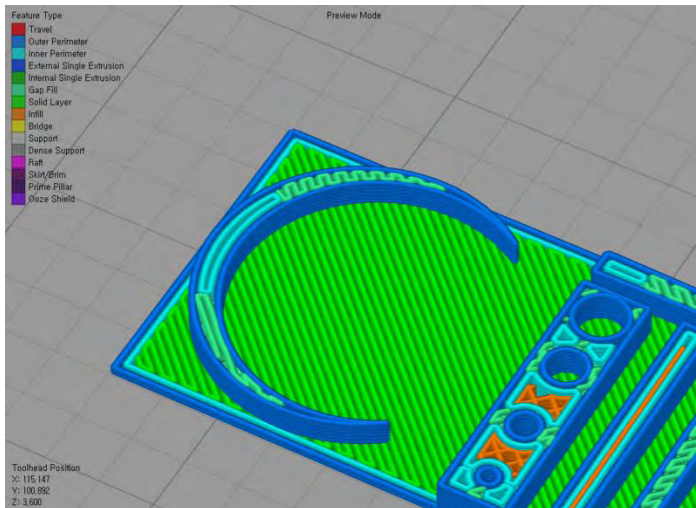
This setting determines the preference between perimeters and single extrusions. If the walls become narrow enough that the perimeters overlap by more than this amount, single extrusions will be used instead. Use higher values if you want to create more perimeters, or smaller values if you want to create more single extrusions.

Here is an image by KeyboardWarrior that describes this setting. It's from a 7/27/2015 post on the S3D user forum that I have updated to use the current parameter names:

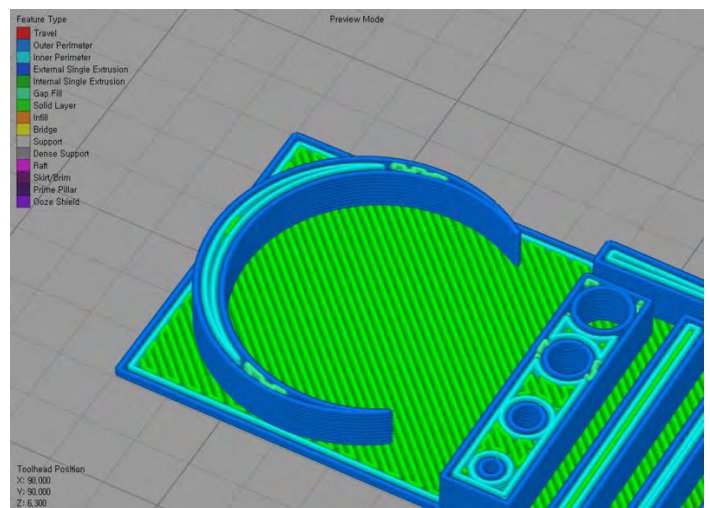




I think these images from the Naver blog posts further highlight the effect of *Allowed perimeter overlap*:



Allowed perimeter overlap: 1%



Allowed perimeter overlap: 99%

## Single Extrusions

Some of the documentation in this section comes from the *Printing Thin Walls and Small Features* article at <https://www.simplify3d.com/support/articles/printing-thin-walls-and-small-features/>.

### *Minimum Extrusion Length*

mm <singleExtrusionMinLength> [MinExLen]

Single extrusion segments with a total length below this value will not be printed (helps save time for unnecessary segments).

If a single extrusion is shorter than this distance, it will not be printed. This is useful, as it allows you to filter out very small movements saving valuable print time.

### *Minimum Printing Width*

% <singleExtrusionMinPrintingWidthPercentage>  
[MinPWid]

The minimum width to allow for any single extrusion segments.

You may find that your printer has trouble printing extremely small or very thick extrusions. The *Minimum Printing Width* and *Maximum Printing Width* settings allow you to limit the width that the single extrusions will be printed with. For example, if you were using a 0.4mm nozzle with a 50% minimum extrusion width, this means that all single extrusions that are printed will be at least 0.2mm wide. The same principle applies for *Maximum Printing Width*.

## Maximum Printing Width

% <singleExtrusionMaxPrintingWidthPercentage>  
[MaxPWid]

The maximum width to allow for any single extrusion segments.

(See the description of the *Minimum Printing Width* parameter above.)

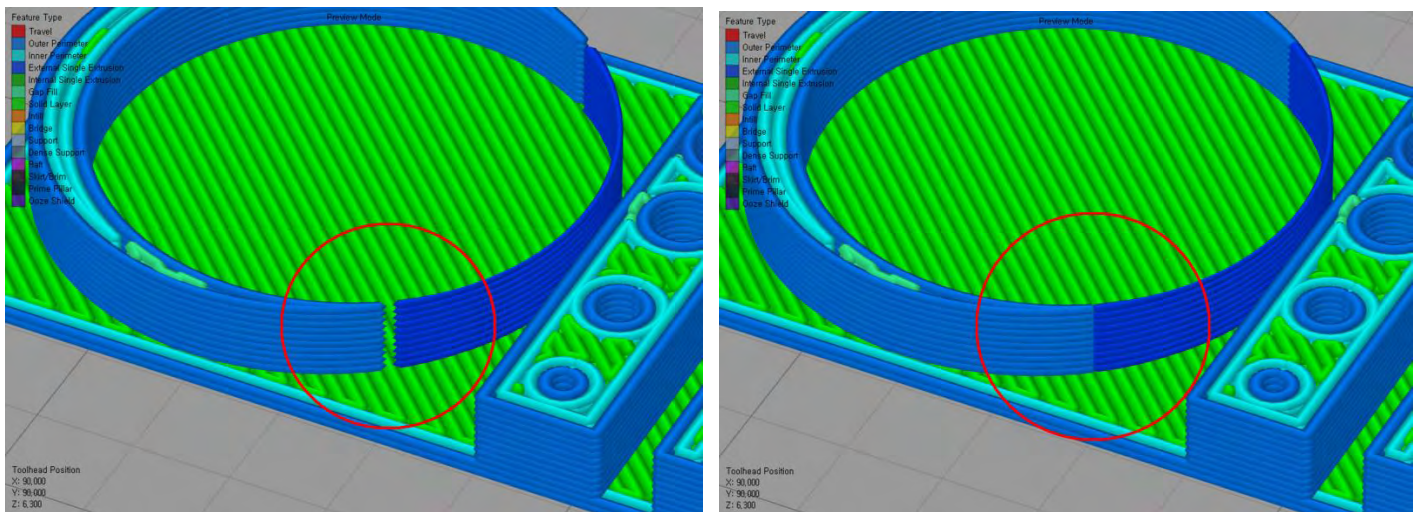
## Endpoint Extension Distance

mm <singleExtrusionMinLength> [EndExtDist]

Single extrusion segments with a total length below this value will not be printed (helps save time for unnecessary segments).

When printing single extrusions, it is important to ensure that the start-points and end-points of these extrusions are securely connected to the base model. For example, the inner end of our turbine blades may have difficulty sticking to the rim, as the toolpaths only have single point of contact. This setting gives the ability to extend the length of the single extrusion by the set value on both sides, which can increase the overlap with the base model to create a stronger connection.

I think these images from the Naver blog posts highlight the effect of this parameter:



Increases the length to the point where the wall ends. *Usually set to the size of the nozzle.*

## **Ooze Control Behavior**

### ***Only retract when crossing open spaces***

Yes / No <onlyRetractWhenCrossingOutline> [Open]

This will limit retraction movements so that they only take place if the nozzle is moving over open spaces.

This is enabled by default for most profiles, since you really only need to do a retraction if you are crossing over an open space, since you would be crossing perimeters and be at risk for stringing.

### ***Force retraction between layers***

Yes / No <retractBetweenLayers> [ForceRet]

Force retraction to occur between every single layer.

Forces a retraction from layer to layer. This is recommended.

### ***Minimum travel for retraction (Switch)***

Yes / No <useRetractionMinTravel> [MinTRetr]

Only retract if the travel movement is longer than this length.

### ***Minimum travel for retraction (Value)***

mm <retractionMinTravel> [MinTRetr]

Retractions will only take place if the travel movement is greater than this value.

If a movement is below this threshold, it won't retract for that movement. This would be if your printer has a lot of problems with retractions, you may need to tweak this, to limit the number of retractions your print has.

### ***Perform retraction during wipe movements***

Yes / No <retractWhileWiping> [RetWipe]

This will perform a moving retraction that takes place over the wipe movement. When disabled, a stationary retraction is used, which may create blobs.

## Only wipe extruder for outer-most perimeters

Yes / No <onlyWipeOutlines> [WipeOuter]

This will limit extruder wipe movements to the outer-most perimeter outline only.

This will ensure that you only use the Wipe feature (Extruder tab) when needed, when it would affect the appearance of your model. Therefore, to save time and be efficient it will only wipe the extruder for outer-most perimeters. It may not be helpful to wipe the extruder after doing Infill, however if you wanted the extruder to wipe after Infill you could do that by unchecking this box.

### ~~Extruder ooze rate~~

An experimental parameter of an earlier version of S3D ...

The Tool-Tip for this parameter states that this is an experimental option, so I personally have not experimented much with it. However, the idea would be to heat up your extruder, prime it, then wipe the filament away and measure how many millimeters of filament will slowly extrude out of the nozzle over a minute, then using that data you would set that in this *Extruder ooze rate* setting.



## Movement Behavior

### *Avoid crossing outline for travel movements*

Yes / No <avoidCrossingOutline> [AvoidCross]

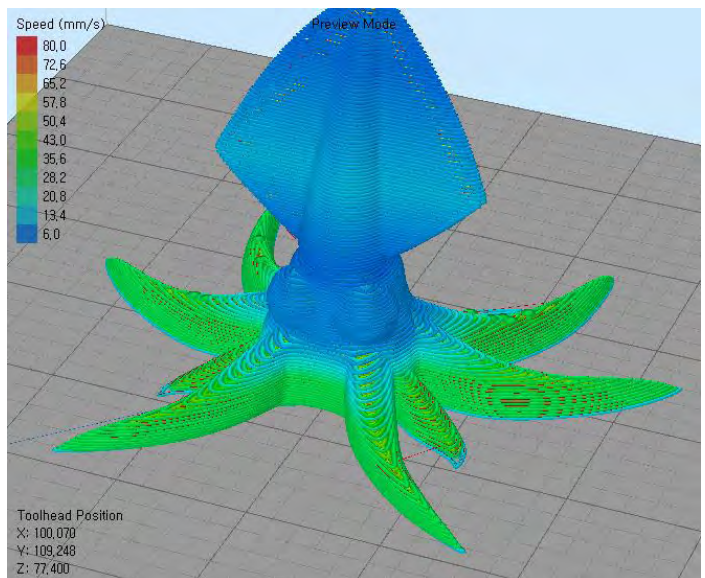
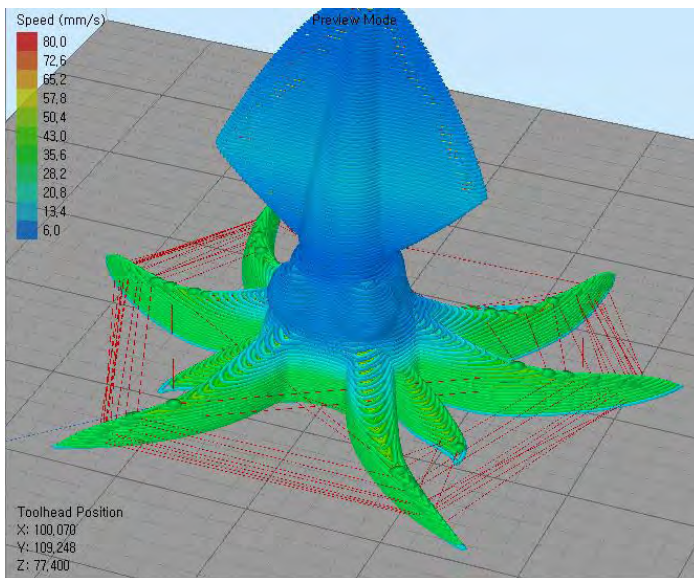
This option will divert the travel path to attempt to avoid crossing the part outline.

### *Maximum allowed detour factor*

Factor <maxMovementDetourFactor> [MaxDetour]

This determines how far the travel movement is allowed to deviate from the original straight-line path. For example, a value of 2.0 means that the detour is allowed to be up to twice as long as the original move.

I think these images from the Naver blog posts highlight the effect of this parameter:



The squid legs are made up of several strands. When you use these parameters, the nozzle moves inside the part and you can eliminate unnecessary spider webs (filaments in the air) that are shown by the red lines.



## Slicing Behavior

### Non-manifold segments

Choice <robustSlicing> [NonManSeg]

Discard: Non-manifold, open loop segments will be completely discarded.

Heal: Non-manifold, open loop segments will be automatically healed, if possible.

Non-Manifold segments are errors in the mesh. These are relatively common. The best behavior traditionally is to use **Heal** to attempt to patch up these missing areas. I personally have not needed to use the **Discard** option.

### Merge all outlines into a single solid model

Yes / No <mergeAllIntoSolid> [Merge]

All surfaces in the model will be fused together into a single solid object.

This feature in the software will slice the model taking all of the outer most perimeter outlines and slice with those assuming a solid model as the interior. For instance, if you had a model that was a block with screw-holes in it, but had **Merge all outlines into a single solid model** checked, the holes would not show up. Using **Merge all outlines** is really useful for models that have errors in the mesh or if using **Vase Mode**.