TRAK® 3ntr 3D Printers
SSI Supplemental Manual

Document: 32242
Version: 092420

SOUTHWESTERN INDUSTRIES, INC.
2615 Homestead Place
Rancho Dominguez, CA 90220-5610 USA
T | 310.608.4422 | F | 310.764.2668
Service Department: 800.367.3165
e-mail: sales@trakmt.com | service@trakmt.com | web: www.trakmt.com

TRAK Machine Tools
Southwestern Industries, Inc.
2615 Homestead Place
Rancho Dominguez, CA 90220-5610
Phone 310.608.4422 ▪ Fax 310.764.2668
Service Department
Phone 800.367.3165 ▪ Fax 310.886.8029
Web: www.trakmt.com

Table of Contents

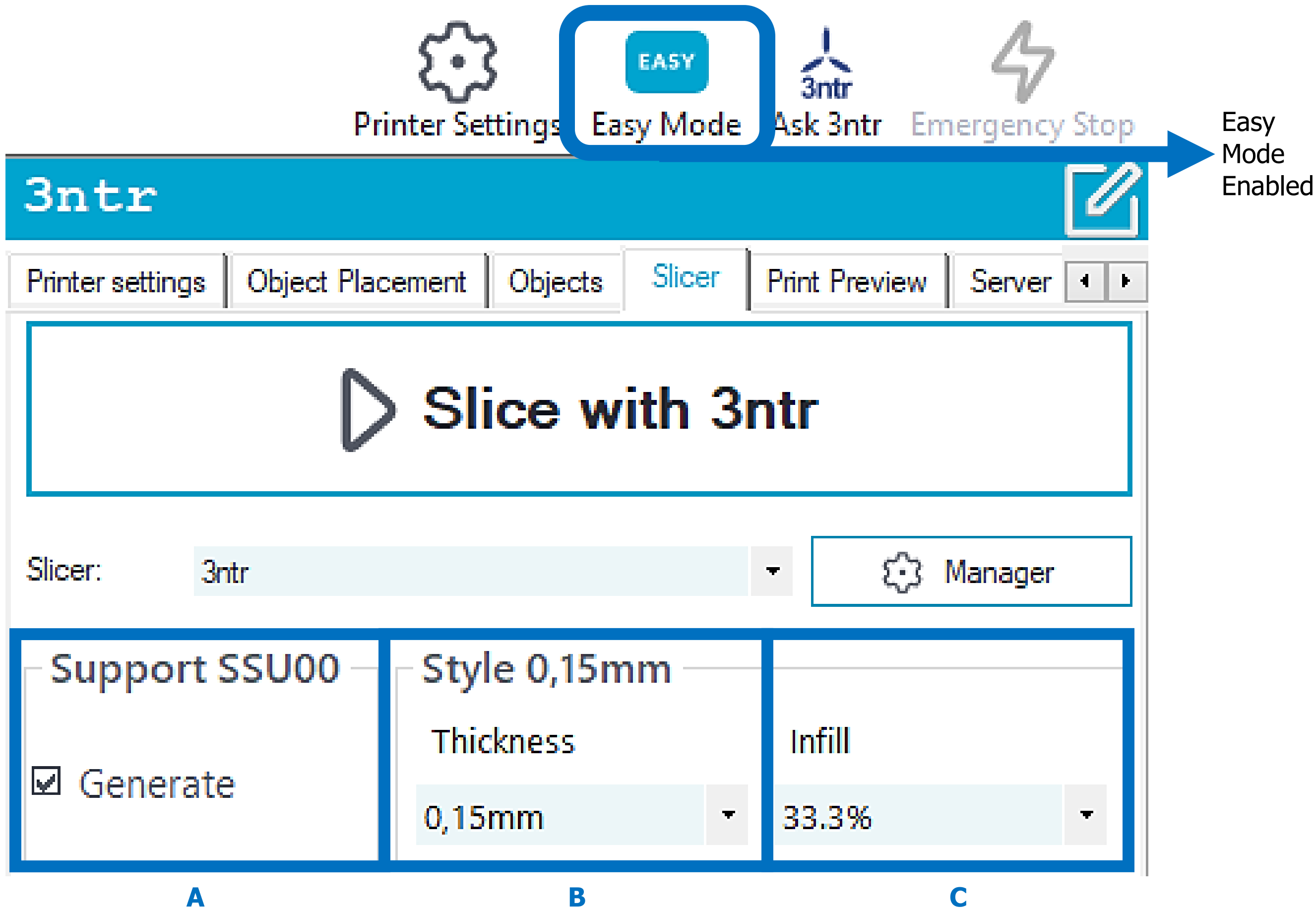
1.0 Quick Overview

When using SSI, you have two different modes to choose from: Easy Mode & Advanced Mode. Switching to Advanced mode is best when you need larger, stronger, more accurate parts, or they require thinner walls or text.

1.1 Easy Mode

When printing while Easy Mode is enabled, the user will only be allowed to edit three parameters, all other parameter values will automatically be set to default values and cannot be changed by the user. In order to use the Easy Mode feature, you must enable it by clicking on the button. Once the button turns blue, Easy Mode is enabled, as demonstrated below:

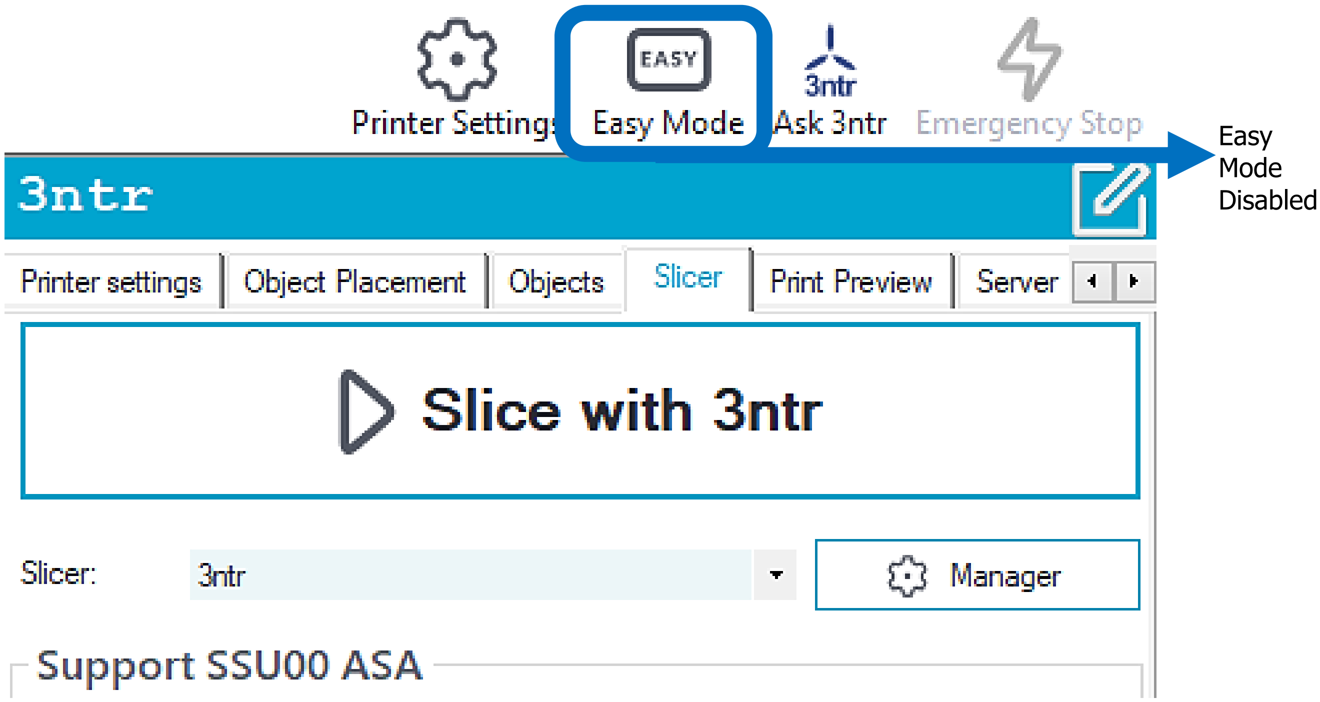
1. Generate Support (Check);
2. Thickness (mm);
3. Infill (%)



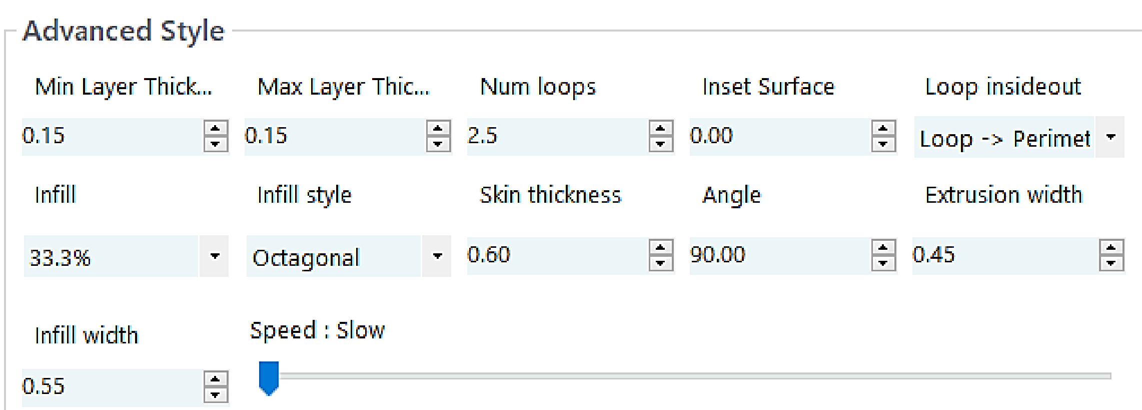
Note – The default parameter values may not be enough to give the part the more specific characteristics that it may need like, more strength or higher accuracy. Such part characteristics may be achieved by switching to Advanced Mode and changing the parameter values, manually.

1.2 Advanced Mode

When slicing in Advanced Mode, the user will be allowed to edit all parameters under Advanced Style. In order to use Advanced Mode, you must disable Easy Mode by clicking on the Easy Mode button. Once the button turns white, Easy Mode is disabled, as demonstrated below:



Advanced mode will unlock the parameters listed below which you are free to change, but should do so with caution.

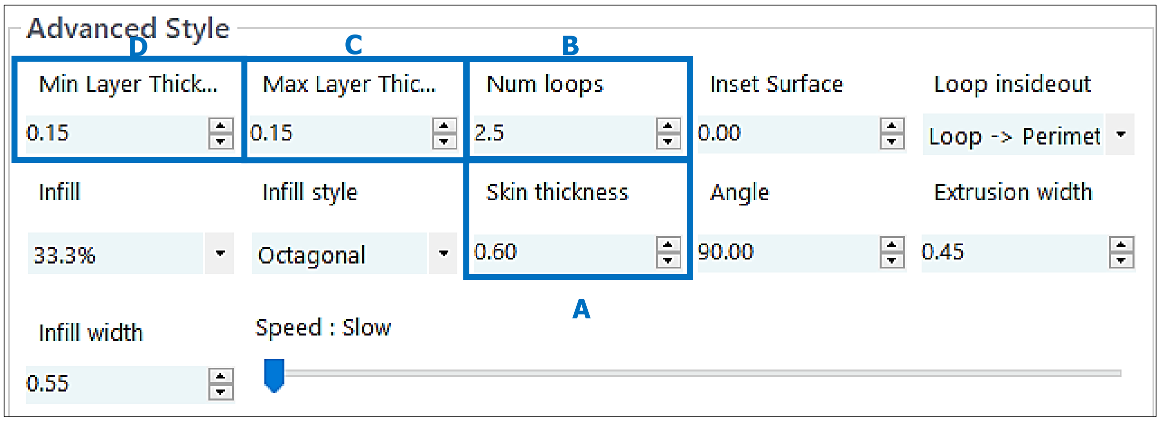


2.0 Recommended Values

The following sections will provide you with some basic recommended parameter values and some descriptions to get you started. Any additional explanations, information, or secondary contributing factors may be found in the "Additional Considerations" section.

Note – The values recommended in the upcoming sections may be used in *most cases*, as they do not account for all the variables associated with printing a part, like material type and nozzle sizes.

2.1 Strong Parts

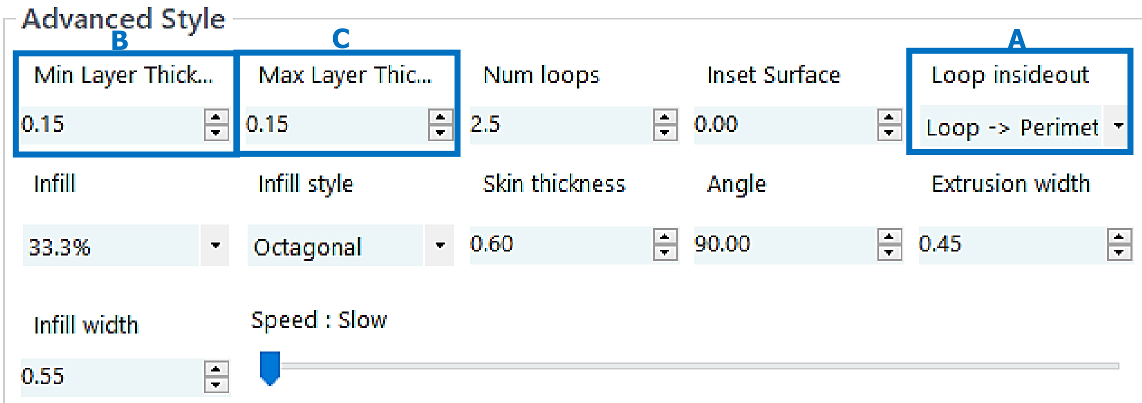


Screenshot of SSI Advanced Style Section

Table of Recommended Values

Parameters	Description	Recommended Values
A. Skin Thickness	Generally, higher skin thickness values, result in enhanced part strength. **NOTE** - Skin printing takes considerable time, particularly when the part has larger, flat surfaces.	• Recommended: 1.5 – 2 mm **More may increase warp*
B. Num Loops	Generally, higher loop counts lead to stronger parts, and higher resistance to deformation. **NOTE** – Always add the ".5" at the end, as that tells the slicer to interlock the layers, not stack them on top of each other. *Ex: For three loops, enter "3.5", for four loops, enter "4.5".*	• Recommended: 3.5 or more **More may increase warp*
C. Min Layer Thickness	With some material, a thicker layer makes for stronger parts, but it can also compromise z-axis tensile strength. When layer adhesion is compromised, a thinner layer will improve layer adhesion, which in turn also gives it more strength.	• Recommended: ○ 50 – 60% of the nozzle size for thicker layers ○ 30 – 50% of the nozzle size for thinner layers • 60% or more may result in weaker parts overall
D. Max Layer Thickness		

2.2 Accurate Parts

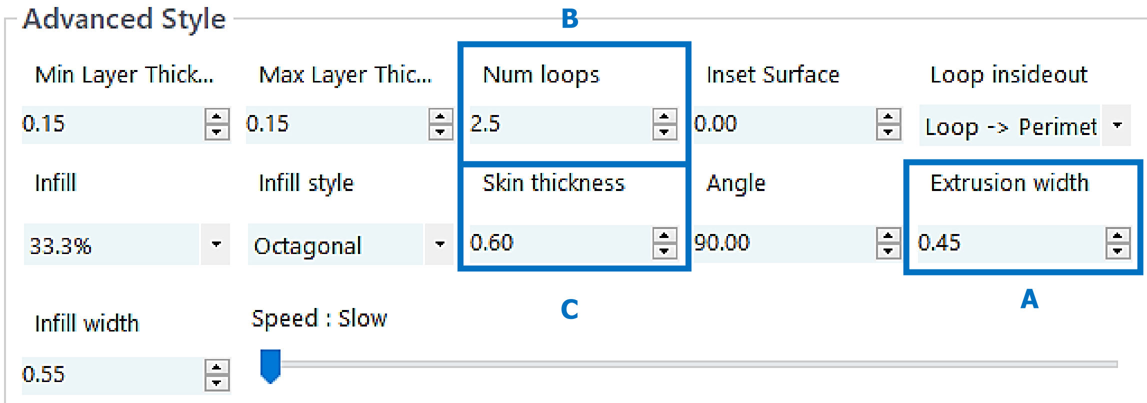


Screenshot of SSI Advanced Style Section

Table of Recommended Values

Parameters	Description	Recommended Values
A. Loop Insideout	The most accurate parts are printed outer Perimeter first. This outermost loop defines the part dimensions in the X and Y direction. It solidifies and contains the remaining polymer for best accuracy.	• Recommended: Perimeter First
B. Min Layer Thickness	• **Adaptive Layers:** Automatically adjusted by SSI when Min & Max Layer Thickness are different. In terms of part accuracy, it does two things: • where it sees a surface at a given Z elevation, it will ensure that surface is printed at that Z elevation, not rounded to the nearest layer • it will also improve curved surfaces by alternating layer thicknesses to build more accurately.	• N/A
C. Max Layer Thickness		
Part Shrink Compensation	This is automatically calculated by SSI once material type is selected. If you wish to change it manually, please see section 3.0.	• N/A

2.3 Thin Wall Parts



Screenshot of SSI Advanced Style Section

Table of Recommended Values

These recommendations do not apply to all materials, see Additional Considerations for more information.

Parameters	Description	Recommended Values
A. Extrusion Width	Generally, higher loop counts lead to stronger parts, and higher resistance to deformation, therefore lower Extrusion Width would leave space for more loops.	• Recommended: 0.35 – 0.45 *(For a 0.4mm nozzle)*
B. Num Loops	The target is a minimum of three loops in the wall thickness. The number of loops allowed, depend on the extrusion width. Loops impact the wall thickness of the part as seen from the top view. **NOTE** – Always add the ".5" at the end, as that tells the slicer to interlock the layers, not stack them on top of each other.	• Recommended: 3.5 or more **More may increase part warp*
C. Skin Thickness	Generally, higher skin thickness values, result in enhanced part strength. **NOTE** - Skin printing takes considerable time, particularly when the part has larger, flat surfaces.	• Recommended: 1.5 – 2 mm **More may increase part warp*

2.4 Text

For best results, take the information you find in this section and size your font, choose cut out or protrude and consider the nozzle diameter you are working with. Then, print the lettering out on a simple test chip to validate everything. Once you have a good test chip, you can apply the same parameters to most parts and expect legible prints.

For Testing Fonts:

Note – Some smaller strokes may print but need testing.

1. Perform protrude test print
2. Perform cut test print
3. Definitely test on intended glyphs
4. Recommend simple, rectangle, with both tests present
5. Slice first, examine paths
6. Consider removing small crown paths
7. Print test chip

When 3D printing text, there are limitations, so generally, you want to avoid fonts that are:

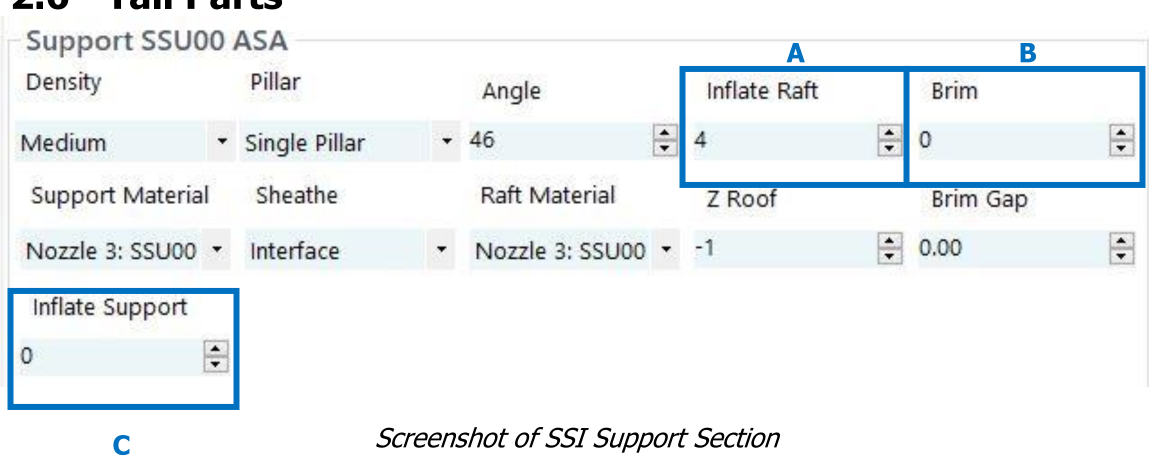
Characteristic	Recommendation / Note
Small in size relative to nozzle diameter	Thinnest font strokes need to be at least 2x the nozzle diameter to guarantee printability. Some smaller strokes may print but need testing.
Serif Fonts	The serifs tend to create geometry that will slice poorly unless the font is large relative to the nozzle diameter.
Bitmap Fonts of any kind	Will not print.
Very complex, highly stylized, or script type fonts	Make sure all strokes are large relative to nozzle diameter, and font is large, with more square aspect ratio.
Extreme aspect ratios, such as font is much taller than it is wide	May print poorly.
Fonts with minimal kerning between glyphs	Spacing between characters should be nozzle diameter or better.
Engraving or machining "single line path" type fonts. Pen plotter type fonts apply here	Will print poorly.

2.5 Larger Parts

SSI, in general, is a very powerful and useful tool when it comes to assisting users with automating a lot of the manual work and taking away a lot of the guesswork. But unfortunately, when it comes to printing larger parts. A large part is one where any of its overall dimensions exceed 75% of any of its printer axis. SSI does not have the capacity to assist users with this, so we recommend the user switch to KISSlicer (KS).

KISSlicer is the engine powering SSI, and has the advanced controls needed to print more challenging parts successfully. Because SSI is a fairly new application, features are still being added, and experts are still testing new materials to add, in order to make 3D printing so much easier.

2.6 Tall Parts



Screenshot of SSI Support Section

Table of Recommended Values

Parameters	Description	Recommended Values
A. Inflate Raft	This increases the size of the raft underneath the part to a size greater than the minimum automatically generated size needed to produce the part. **NOTE** - Be sure and check your part and raft remain within the X and Y boundaries of the build area.	• Recommended: 5 - 10mm
B. Brim	A brim on tall and narrow parts can improve adhesion to the raft as well as improve flatness on the lower build surface of the part.	• Recommended: 5 - 10mm
C. Inflate Support	Narrow and tall support features can fail. Expanding them makes a larger, more robust support feature less prone to failure.	• Recommended: 5 – 10mm

2.7 Flat Parts

**Currently under development*

3.0 Additional Considerations

In this section you will find additional value recommendations, or more detailed explanations if needed.

3.1 Nozzle Size Considerations

The relationship between nozzle size and part strength, level of detail and print speed.

3.1.1 Nozzle Size Strength

The nozzle diameter directly affects the 3D printer extrusion width of each line in your print. Hence with an increase in nozzle size, the strength and toughness of the part is increased provided correct print parameters are selected for layer thickness, extrusion width and infill width.

3.1.2 Nozzle Size and Part Details

A smaller diameter of the nozzle allows for a thinner bead of plastic to extrude. This thinner line of plastic allows the printer to create finer details, and overall tighter control over XY accuracy. A smaller nozzle size, you will be able to print a more detailed text – assuming it's placed on the top side of the printed object.

The nozzle diameter affects the overall level of detail almost exclusively in the horizontal plane parallel to the print surface. Contrary to that, the layer height affects the level of detail on vertical and slanted sides of an object. You have a much greater chance of seeing individual layers on organic-looking objects. The lower you set the layer height, the better the overall result.

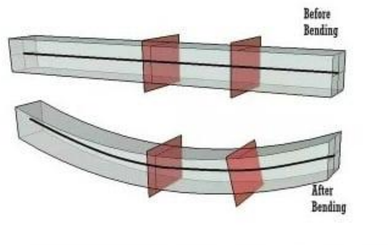
3.1.3 Nozzle Size vs Print Speed

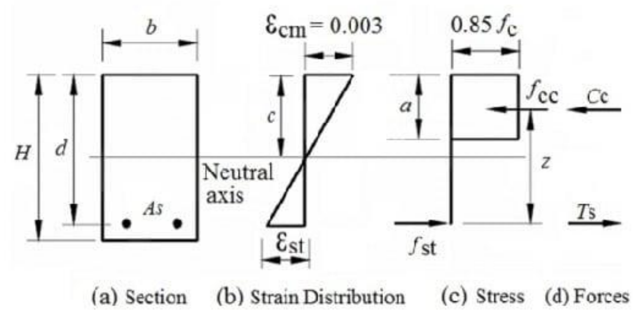
A larger-diameter nozzle lays down wider perimeters, which means that it uses fewer travel moves than a smaller-diameter nozzle to print a wall of the same thickness. A nozzle with a larger diameter also allows for printing with increased layer height. Combining these two effects, leads to a noticeably shorter print time.

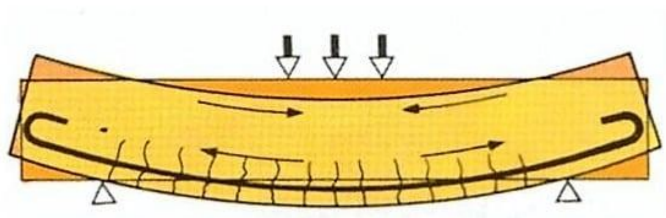
3.2 Strong Parts

How strong a part is, material properties aside, boils down to the outer surfaces and wall thicknesses. Fractures from stress and strain begin on the part exterior and represent the primary failure mode. Some materials, such as elastomers, may not experience failure as they can tolerate very significant expansion and contraction. Other materials, such as PETG, are stiff, and can tolerate very little expansion and contraction. These suggestions are for stiffer materials.

In the diagram below, there is a neutral axis, where the material does not really compress or expand much. In the bent condition, one face will see expansion, the other compression, and there are some other effects on the side faces we can ignore for this simple discussion. Fractures begin at where the strain and stress concentrates, and the faces farthest away from the neutral axis, for a given body, see the greatest change and with that, the greatest chance of failure.







- **Skin Thickness** – Skin thickness is a decimal number, and may be rounded to the nearest layer Z, based on layer height when adaptive is not used. In general, 1.5 - 2mm skins make much stronger parts. Skin printing on many parts takes considerable time, particularly when they have large, flat surfaces. Excessive skin can increase warp, depending on material.

- **Num Loops** – Loops impact the wall thickness of the part as seen from the top view. Loops are integer numbers, and in general, they represent **(# of Loops) x (Nozzle Diameter)**. For example, for a 0.4mm nozzle, 2.5 loops represent a mere ~0.8mm part wall.

- **Min Layer/Max Layer Thickness** – In general, larger layer heights make stronger parts, but the nozzle size needs to be considered or the outcome may actually be weaker parts. There is a tradeoff between the continuous strands of polymer and overall layer adhesion in play. Some materials may benefit more from layer adhesion too, or they may require thinner layers for adhesion to happen. Where this may be true, it is good to try 30 to 50% of the nozzle diameter as layer thickness and compare with parts printed with larger layer thicknesses of 50 to 60%. For example, for a 0.4mm nozzle, 0.2 to 0.25 are good layer sizes for strong parts, for example.

3.2.1 Secondary Contributing Factors

Other contributing factors include infill, extrusion width, speed and use of chamfers.

Infill	Dimensional inaccuracy may increase with increase in infill density. Some materials may warp at higher infill density. Usually lower infill values lead to higher part accuracy.	• Recommended: 20% - 50%
Infill Style	Some infill styles are beneficial for stronger parts.	• Recommended: Rounded/Octagonal
Infill Width	Wide infill contributes to the structural stability of the printed part.	• Recommended 0.45 – 0.55 *(For a 0.4mm nozzle)*
Extrusion Width	The number of loops allowed, depend on the extrusion width. Generally, higher loop counts lead to stronger parts, and higher resistance to deformation, therefore lower Extrusion Width would leave space for more loops.	• Recommended: 0.45 – 0.50 *(For a 0.4mm nozzle)*
Chamfers	Any part features, with dimensions of size, that are less than roughly 2.5 times the nozzle diameter will be weak due to the lack of polymer allowed by the printing process. Where possible, thicken or support these features with chamfers. Avoid sharp edges where surfaces intersect at close to right angles. Use a chamfer or bridge surface to make the transition. Replace radii with chamfers where possible and practical.	• N/A – This has to do with your part design.

3.3 Accurate Parts

- **Loop Order** – Printing outermost first does decrease support overhang performance. For many materials, this is minor, but something to be aware of should you encounter support performance issues.

- **Adaptive Layers** – This is automatically adjusted by SSI when the min & max layer thicknesses are different.

3.3.1 Secondary Contributing Factors

Other contributing factors to accurate parts include infill, speed, brim, part shrink compensation, and toolpath fidelity, like coarse STL files.

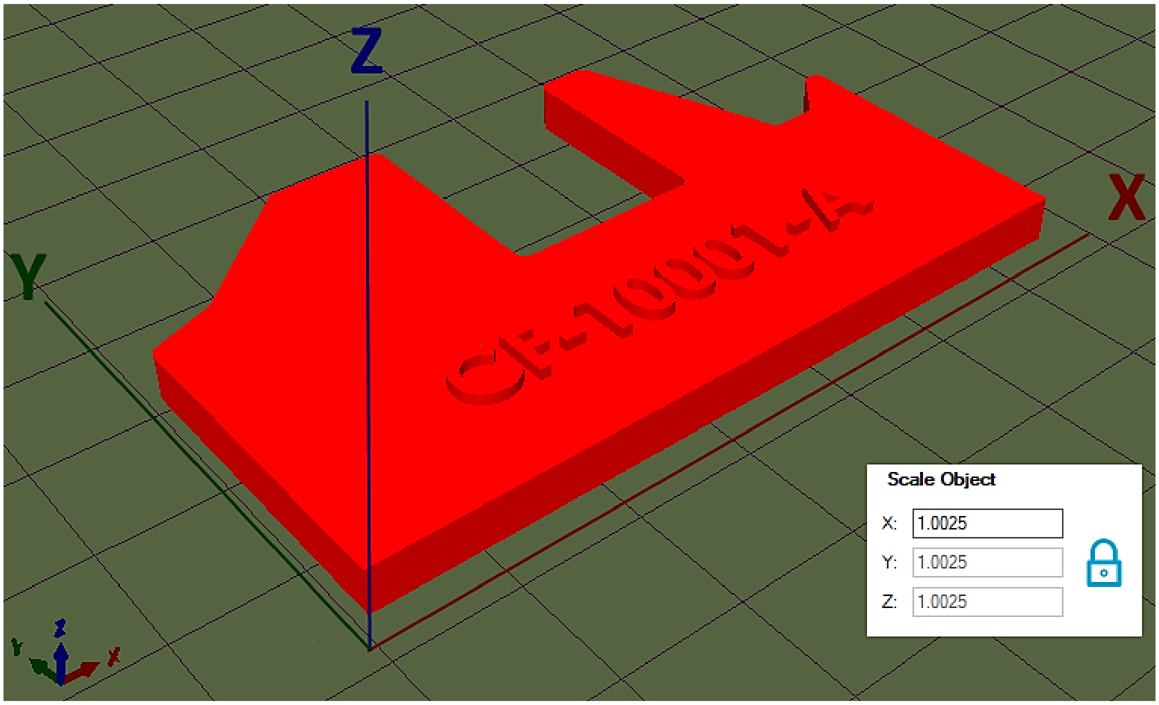
Infill	Dimensional inaccuracy may increase with increase in infill density. Some materials may warp at higher infill density. Usually lower infill values lead to higher part accuracy.	• Recommended: 20 - 50%
Brim	Adding brim on the part helps in reducing the warp and get straighter edges. In SSI, this makes for a wider foot area to increase stability and tray adhesion. **For parts with flat surfaces, please see section on Flat Surfaces (under development)*	• Recommended: Under Evaluation

Materials can vary extremely widely in how they perform when printed. Where tolerances are tight, consider low shrink materials where possible. At this time, the STL file and slicing process may render small holes and other features undersized.

3.3.2 Part Shrink

Use this functionality with caution!

In SSI the default scale for a given object includes the 3ntr, TRAK and Plural known and tested shrink compensations. In general, these are sufficient for most scenarios and represent nominal machine settings for that material and tolerance expectations. There are times when making an overall dimension adjustment may be necessary or desirable. In that case, you can combine your desired dimension change, expressed as a percentage of overall part size, with the default shrink compensation scale shown to arrive at a custom part scale.



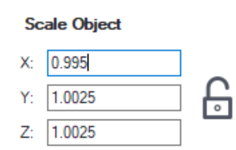
The illustration above shows the default shrink compensation for ASA. By default, SSI computes 1.0025:
(intended part size * known shrink factor)

The scale object icon can be found on the Object Placement tab in SSI. The scale can be uniform, shown as the lock icon in closed position, or non-uniform shown as lock in open position.

For example, say the part overall dimension in the X direction is 5.010" as printed, and a different value, say 4.995", would perform better. The part is fine in the Y and Z directions.

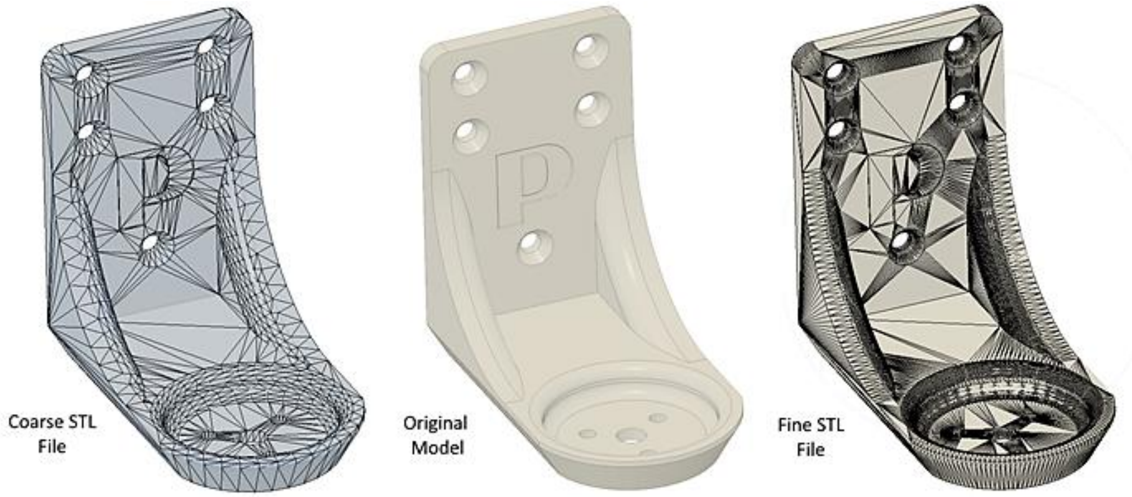
The necessary scale in X would be: (default SSI scale) x (as desired / as printed)
(1.0025) x (4.995/5.010) = **0.995**

To input this compensation, click the lock to decouple all the axis and input the new factor into the X scale dialog as shown below. Then slice the part as usual and when printed, the part will be changed in the X direction only.

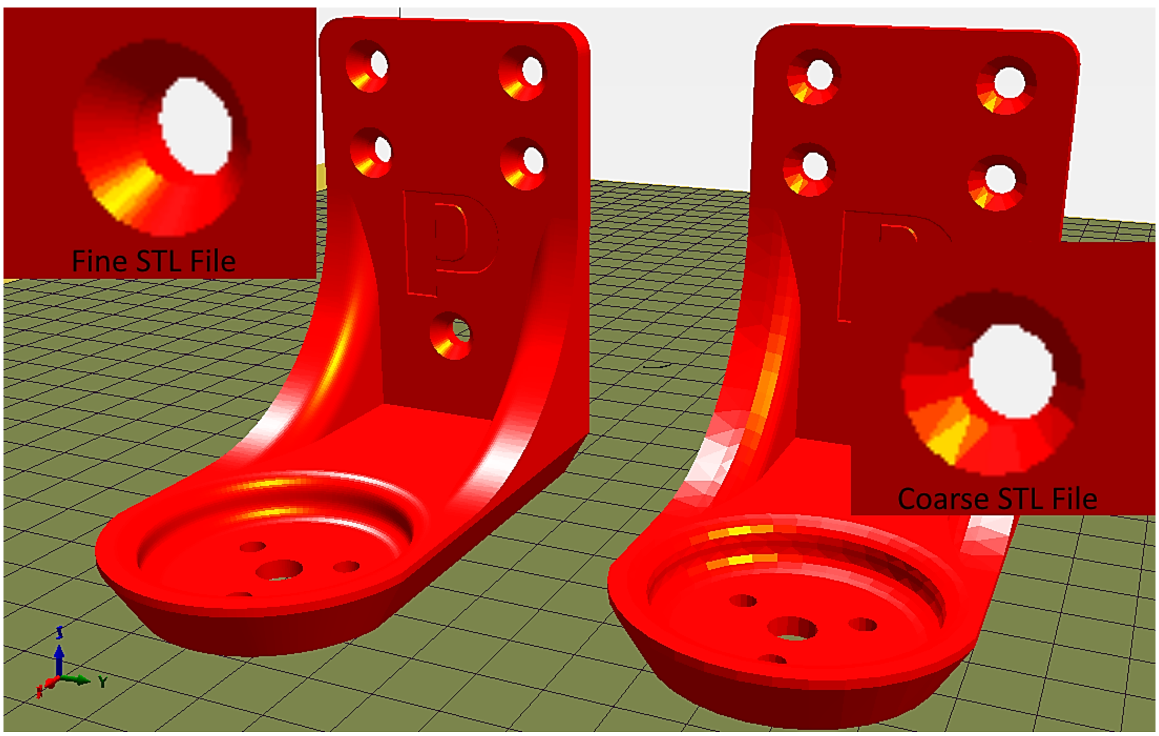


3.3.3 Coarse STL Files

It is beyond the scope of this document to detail STL file settings out of the various CAD system. However, it is very useful to gauge overall file precision visually. Consider the model and two STL file outputs shown below:



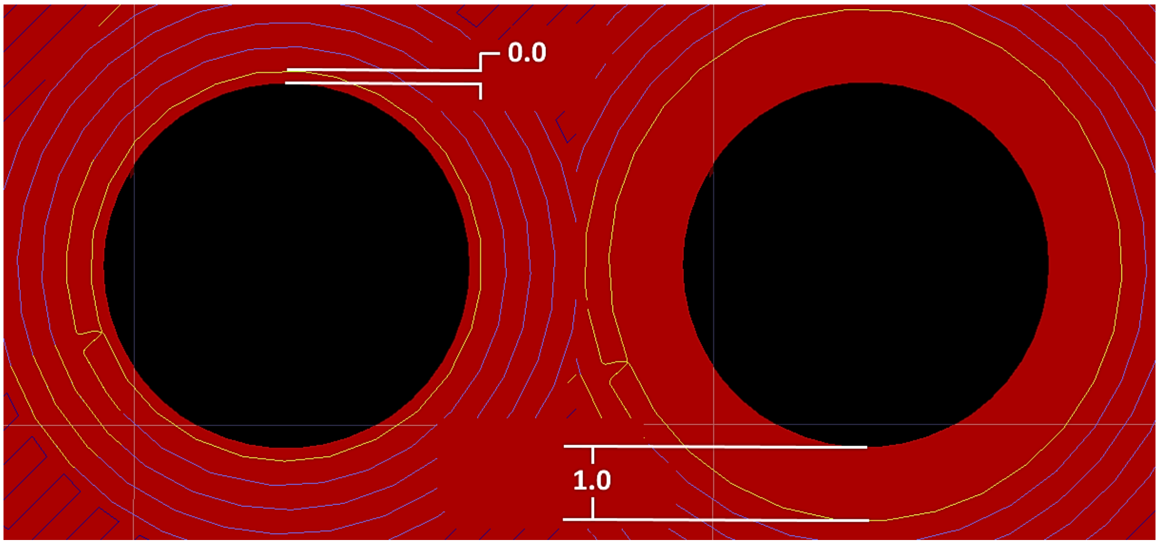
The coarse model will be less accurate than the fine STL model will be. In general, you can gauge approximate error by looking at various features in the model, and then the facet sizes to understand what the error potential can be.



Once loaded onto SSI, you can see the difference between the coarse and fine STL files, and the difference between them can be seen easily. While both of these parts will be perfectly functional, the coarser one will have visible facets in the finished print, and dimensions of size on primary features will have some error related to the chordal deviation caused by the lower number of triangles used to represent the model. If it looks good in SSI, the file precision error is lower than the process error and can be ignored in the vast majority of cases. Where facets can be easily distinguished, that does not mean the part will be bad, but it does mean there will be more dimensional error than would otherwise be a part of the printing process.

3.3.4 Inset Surface

This parameter offsets part boundary surfaces that are internal to the part, such as holes and cutouts. This is a global parameter and may not be appropriate in all scenarios. A good time to use this is when you have a part with many smaller holes that may run undersized, due to how slicing and printing currently work. The defining loops of these features will be offset by the amount you input.

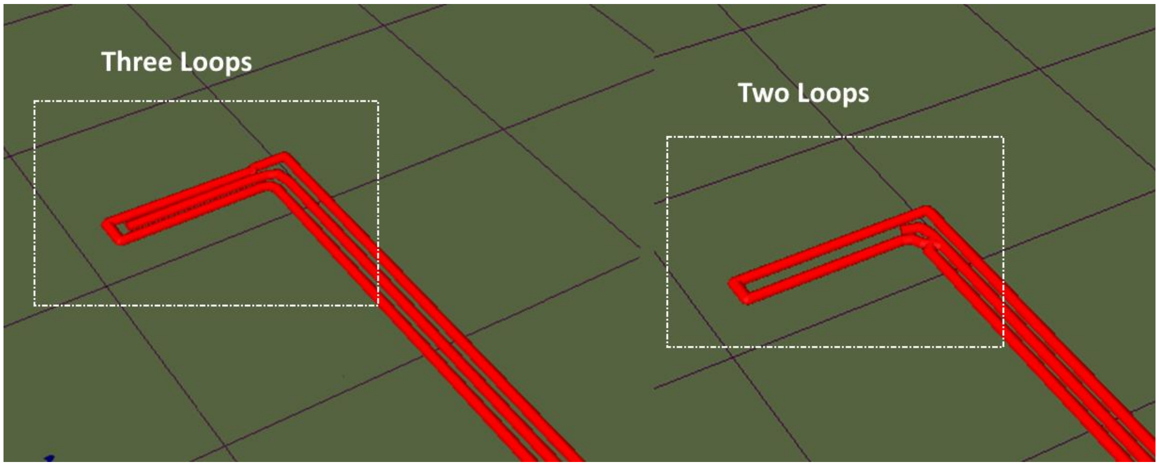


The default is no inset compensation. A large, 1mm offset is shown at right for clarity. The holes would be oversized by 1mm. In some cases, small values input here can improve small feature accuracy.

3.4 Thin Wall Parts

3.4.1 Extrusion Width

Parts with wall thicknesses less than 3 times the nozzle diameter are limited in how strong they can become. Performance on thin wall parts is primarily dictated by the nozzle diameter in use on the printer. For thin wall parts, you want 3 loops if it is at all possible to do with the nozzle in use. Where there are only two loops, consider changing the Extrusion Width to allow for three loops. By default, SSI overdrives the nozzle slightly. In general, you can under or over drive a nozzle by 25 percent with little to no change in printing performance. As you exceed 25 percent, artifacts such as fine stringing and occasional blobbing may appear. These may not be an issue, depending on the amount of labor required to remove them from the part after printing. Always Print Preview thin walled parts to see the number of loops present on the walls.



For example, 0.4mm nozzle diameter x 3 loops = 1.2mm wall thickness.

The thin wall part on the right will have poor integrity. The one on the left is improved by either a smaller extrusion width or nozzle, or both.

3.4.2 Num. Loops

Loops impact the wall thickness of the part as seen from the top view. Loops are integer numbers, and in general, they represent (# of Loops) x (Nozzle Diameter). For example, for a 0.4mm nozzle, 2.5 loops represent a mere ~0.8mm part wall.

3.4.3 Skin Thickness

Skin thickness is a decimal number, and may be rounded to the nearest layer Z, based on layer height when adaptive is not used. In general, 1.5 - 2mm skins make much stronger parts. Skin printing on many parts takes considerable time, particularly when they have large, flat surfaces. Excessive skin can increase warp, depending on material. Generally speaking, best practice here is to match the skin thickness parameter to the intended wall thickness. It’s okay to overdrive this some too. The printer will only print the required wall thickness, even when the skin thickness parameter exceeds the model geometry. For example, if wall thickness is set to 1.5mm, then setting Skin Thickness to 2.0mm will ensure the entire wall is printed without infill. A lower setting than 1.5mm may result in one or more layers with infill, which will reduce overall part integrity.

3.5 Text

For printing in soft materials:

- Simple fonts are better
- Avoid small crown paths to prevent blobbing and stringing

For lettering is different material:

- Export glyphs as their own STL bodies
- Select all bodies in slicer and assign material
- This is true for different colors and same material

For height of depth of lettering:

- Good rule of thumb is depth or height should be 10 percent of font height as seen in the X, Y plane, maximum
  (i.e.: 20mm font = 2mm protrude or cut)
- Less aggressive lettering, shallow cuts, low protrudes are best and most robust for printing
- High wear potential favors cut away lettering

For lettering standing in the Z direction, or at angles to the X, Y plane, the printer will generally require:

- Larger strokes
- Closer to square aspect ratio
- More space between glyphs
- Simple is better

Make sure STL Fil Precision is adequate for font detail:

- Too few facets = poor quality
- Too many facets = generally prints well, may be memory intensive to slice
- Evaluate STL File export

Here is an example of a test chip, sliced to evaluate paths:

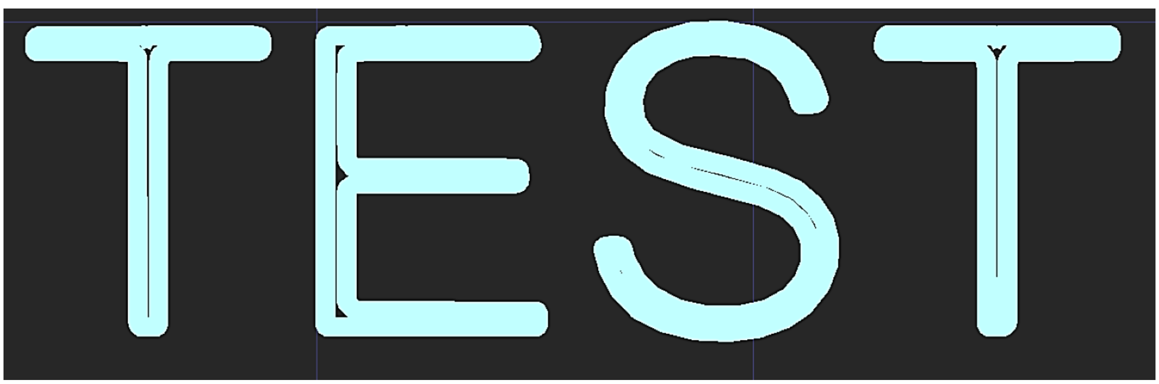
NOTE – The screenshots below are from KISSlicer, for presentation purposes. Below is the color legend.

KISSlicer Color Legend:

- **Example #1**
  Font Name: Ariel
  Height: 10mm
  Type: Protruded Lettering
  Nozzle: 0.4mm
  Lower case glyphs show some gaps and partial paths. (See the cyan sections in image below.) As a protrude lettering, with very shallow height, this would print poorly, mostly legibly.

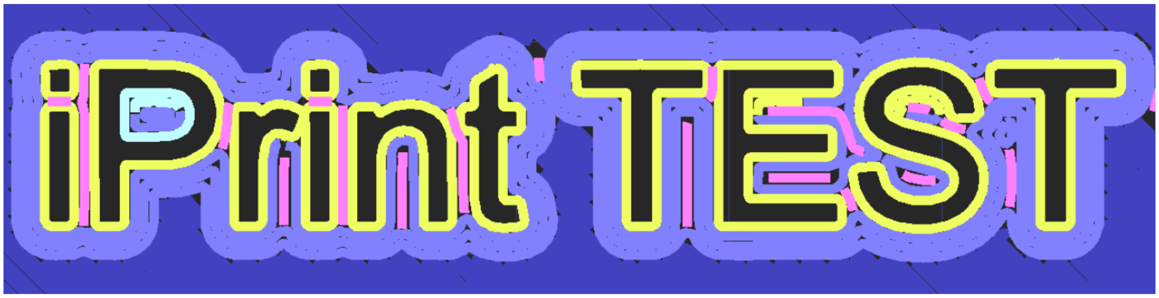
Ex #1 – Ariel 10mm Protruded Lettering

- **Example #2**
  Font Name: Ariel
  Height: 10mm
  Type: Protruded lettering
  Nozzle: 0.4mm
  Same font, upper case. Notice how all glyphs have complete printer paths. This would print well and represents very good font size relative to nozzle diameter. Much smaller, and partial paths are likely to appear. Aspect ratio could be narrowed some, as could spacing between glyphs.



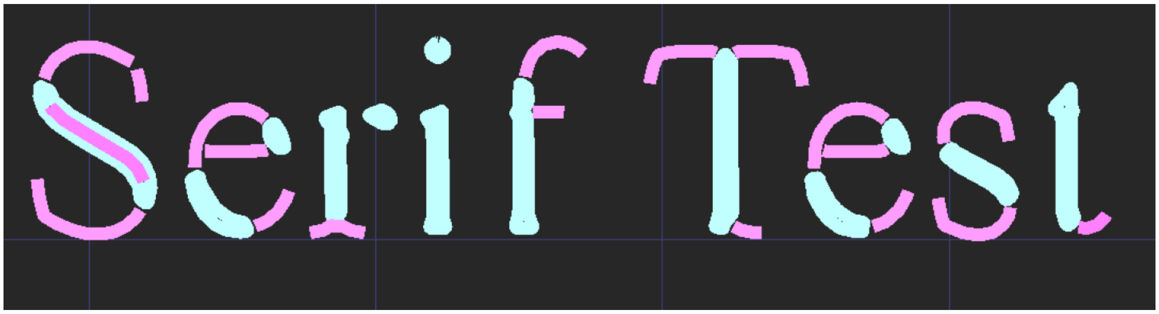
Ex #2 – Ariel 10mm Protruded Lettering Upper Case Good Paths

- **Example #3**
  Font Name: Ariel
  Height: 10mm
  Type: Cut Out Lettering
  Nozzle: 0.4mm
  As a cutout lettering, both lower and upper case will print legibly. Notice the many small "fill in paths" and the spacing between glyphs does not always allow for complete paths. This font would be best a bit larger than 10mm or expanded in the X direction for optimal printing.



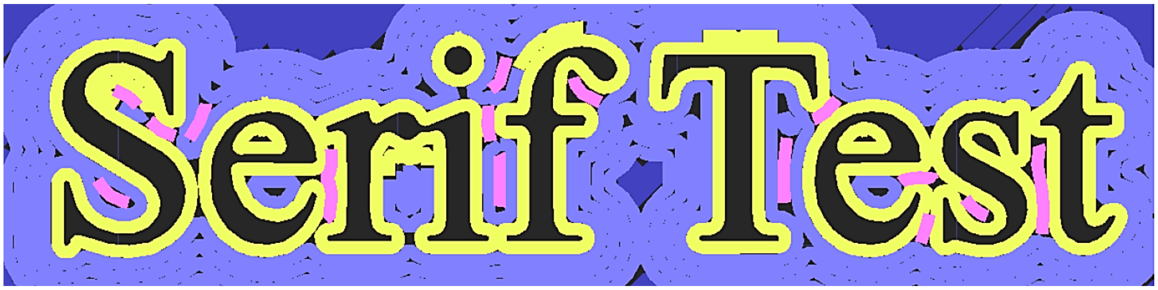
Ex #3 – Ariel Cut Out Lettering

- **Example #4**
  Font Name: Times New Roman
  Height: 10mm
  Type: Protrude Lettering
  Nozzle: 0.4mm
  This same font size, in a more stylized font is not viable at this print size and nozzle diameter. Some paths are missing, others just single, best guess paths. Increase font size, until all paths are seen.



Ex #4 – Times New Roman Protrude Poor Paths

- **Example #5**
  Font Name: Times New Roman
  Height: 10mm
  Type: Cut Out Lettering
  Nozzle: 0.4mm



Ex #5 – Times New Roman Cut Out Lettering

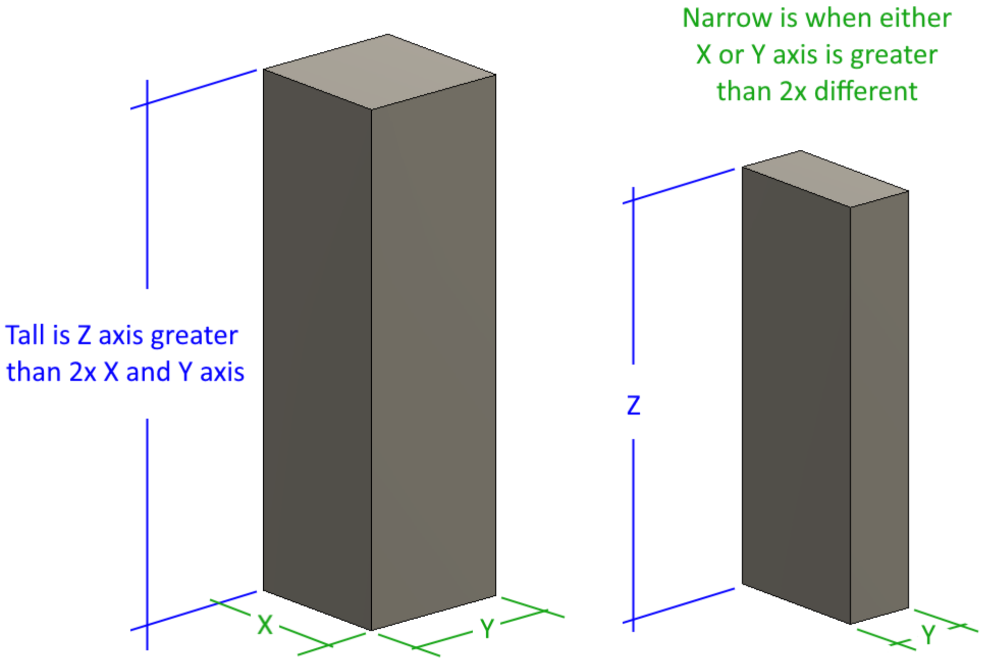
For best results, take the information you find in this form, size your font, choose cut out or protrude and consider the nozzle diameter you are working with. Then, print the lettering out on a simple test chip to validate everything. Once you have a good test chip, you can apply the same parameters to most parts and expect legible prints.

3.6 Tall Parts

In general, a part is considered tall when its Z extent measurement is significantly greater than it is in the other axis. Tall part effects begin to be seen when the Z is more than 2x the X and Y extent dimensions.

A tall part is more difficult when either X or Y are different in the same way the Z axis is for tall parts. A part in this condition is both tall and narrow.

A tall part and tall and narrow part are shown below.

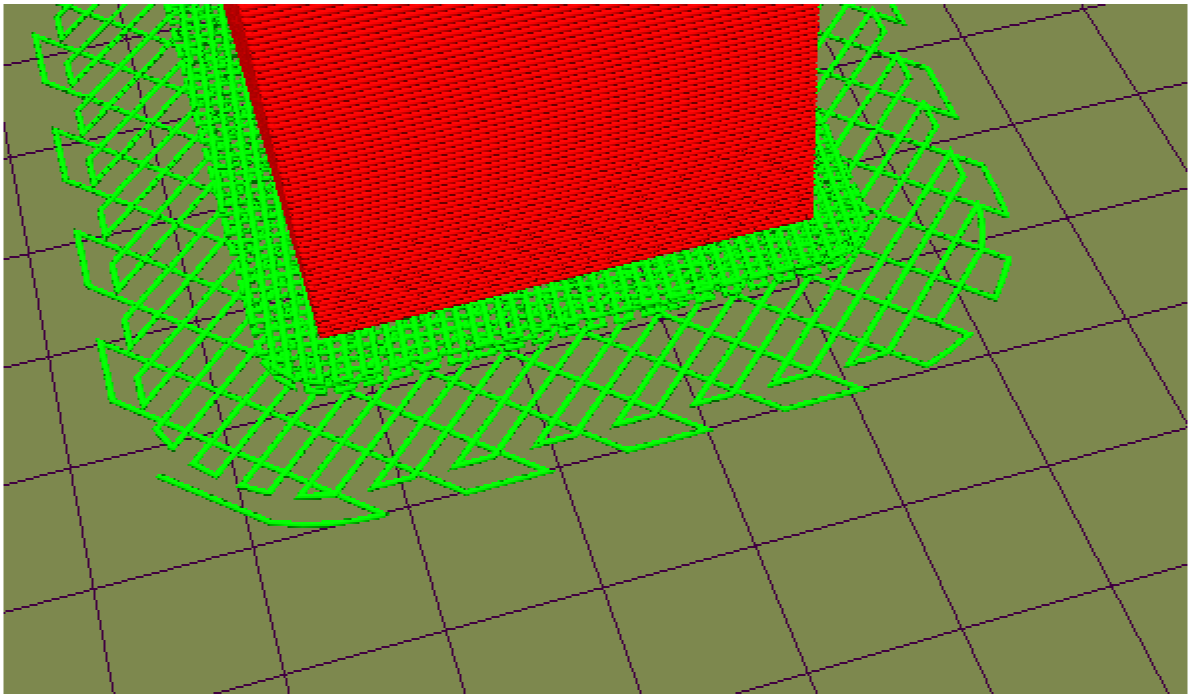


In general, tall and narrow parts both benefit from an expanded raft and can benefit from a brim, depending on their geometry. As a part gets taller, the ease it may be knocked over increases. This is truer when a tall part has an additional narrow axis as shown above.

Optionally, and where it's appropriate, a larger layer height can help with these parts due to there being more clearance between the part and the nozzle while printing.

3.6.1 Inflate Raft

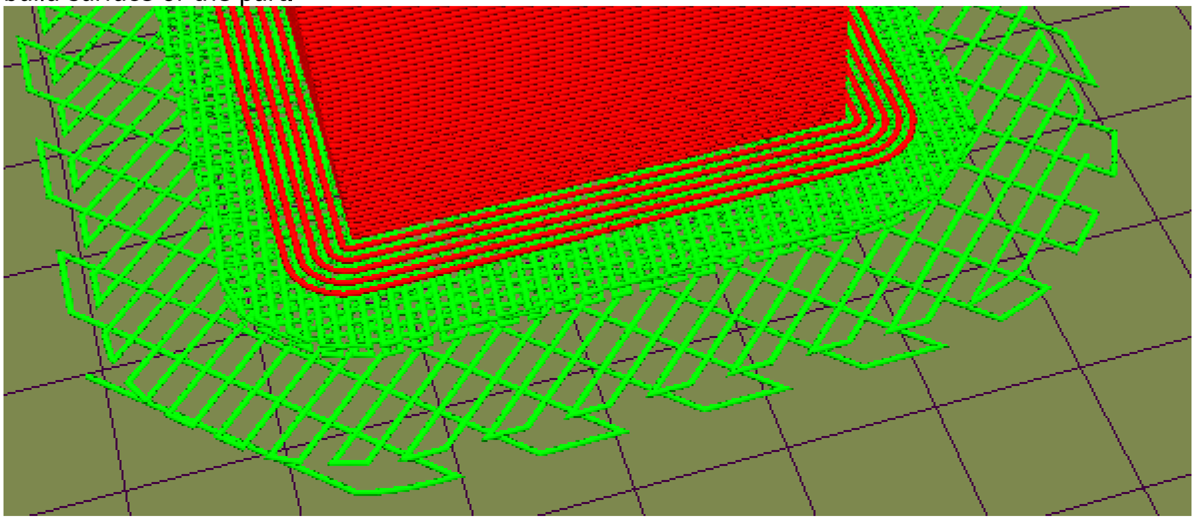
This increases the size of the raft underneath the part to a size greater than the minimum automatically generated size needed to produce the part. The default maximum value is 10mm. Be sure and check your part and raft remain within the X and Y boundaries of the build area.



Expanded raft on tall and narrow part shown.

3.6.2 Brim

A brim on tall and narrow parts can improve adhesion to the raft as well as improve flatness on the lower build surface of the part.



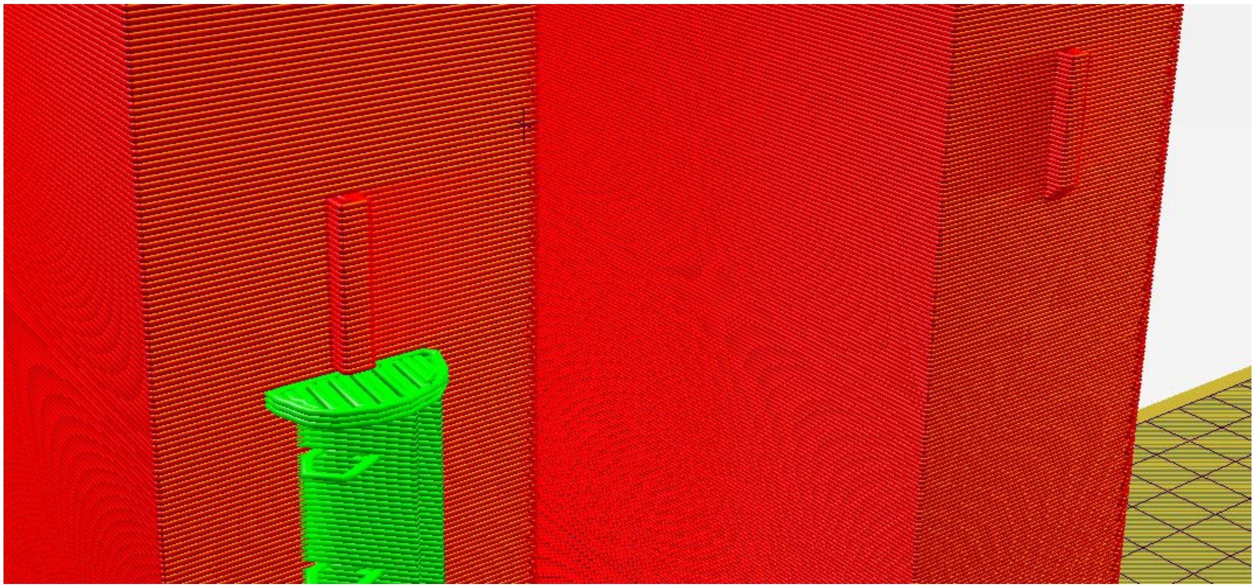
Brim and expanded raft on tall and narrow part shown.

The loops directly impact part strength. More is better. The SSI default of 2.5 loops will not make very robust, thin wall parts.

3.6.3 Inflate Support (C)

These parts may have small features on the outside, or vertical walls not quite 90 degrees. In both cases, there will often be a need to generate a small support running up the part. Narrow and tall support features can fail. Expanding them makes a larger, more robust support feature less prone to failure.

Where possible, adding part geometry to make features self-supporting can very significantly improve print time and reduce failure potential.



Expanded support and feature designed to not require support shown.