Contents

Introduction

This document is intended to help clear up some of the mystery regarding supports in Skeinforge. It was written from the perspective of a Makerbot Replicator 2 operator using ReplicatorG-Sailfish, so "Your Mileage May Vary". Throughout this document, I will attempt to provide guidance on any features that do not apply to other applications using Skeinforge (Makerware, ReplicatorG – MBI flavor).

The Raft plugin is used to configure two major components of a print:
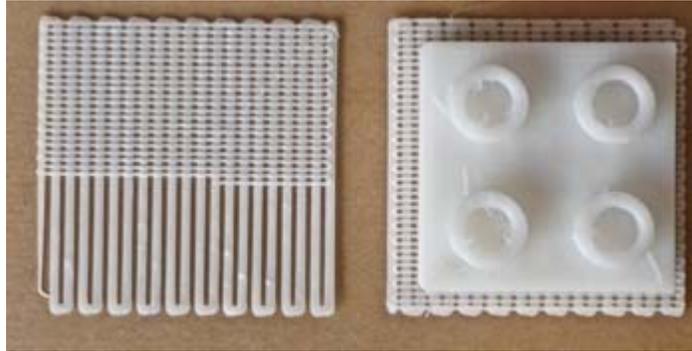
- Rafts
  -
- Supports
  -

Note that neither the raft nor supports are actually a part of your final print. They are temporary structures intended to help you print a better model.

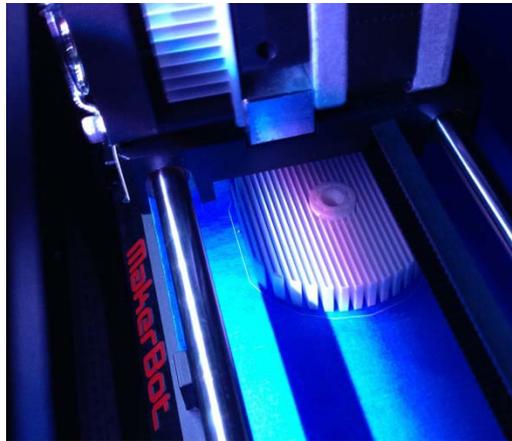I'm not going to expend any effort talking about the actual Raft part of the Raft plugin in this document (perhaps another day). Rafts, in my opinion, should never be used. There are two primary purposes of a raft: To compensate for a less-than-perfect build plate, and to help eliminate curling.

To the first point, build plate problems, those are issues that should be solved prior to finessing Skeinforge. In this day and age, there's no reason that you shouldn't be printing on a reasonably flat and smooth surface.

As regards curling, it is much more of a problem with ABS than PLA. However, both materials have some tendency to pull up at corners (especially sharp corners). The use of "Helper Disks" appears to be a much more effective way to alleviate curling. Helper Disks have more surface area in contact with the build plate, and therefore stick better. They also are much easier to remove than a raft, and don't create extra post-processing work.

The material used for a build plate also makes a difference in controlling curling. But that is a subject for another document. Now, to the settings.

Settings

The screenshots used below are for a decent middle-ground support system (actual raft is disabled), and will be used as the basis for comparing options.

☑ Activate Raft

☑ Add Raft, Elevate Nozzle, Orbit:

- Base -
Base Feed Rate Multiplier (ratio):	0.75
Base Flow Rate Multiplier (ratio):	0.6
Base Infill Density (ratio):	0.3
Base Layer Thickness over Layer Thickness:	2.2
Base Layers (integer):	0
Base Nozzle Lift over Base Layer Thickness (ratio):	0.3

☐ Initial Circling:

Infill Overhang over Extrusion Width (ratio):	0.1

- Interface -
Interface Feed Rate Multiplier (ratio):	1.5
Interface Flow Rate Multiplier (ratio):	0.7
Interface Infill Density (ratio):	0.4
Interface Layer Thickness over Layer Thickness:	1.8
Interface Layers (integer):	0
Interface Nozzle Lift over Interface Layer Thickness (ratio):	0.3

- Name of Alteration Files -
Name of Support End File:	support_end.gcode
Name of Support Start File:	support_start.gcode

Operating Nozzle Lift over Layer Thickness (ratio):	0.0

- Raft Size -
Raft Additional Margin over Length (%):	2.5
Raft Margin (mm):	2.5

- Support -

☐ Support Cross Hatch

Support Flow Rate over Operating Flow Rate (ratio):	0.72
Support Gap over Perimeter Extrusion Width (ratio):	0.005
Support Material Choice:	Exterior Only
Support Minimum Angle (degrees):	55.0

Activate Raft / Add Raft, Elevate Nozzle, Orbit

In order for supports to work at all, these must be checked (activated).

Note: ReplicatorG (both the MBI and Sailfish versions) should always have this activated. You will use Print-O-Matic to determine whether supports are actually used (as Print-O-Matic will override this checkbox, depending on your selection). If you're using Makerware, then it's a bit more difficult: You have to edit the Skeinforge profile and toggle this checkbox every time.

Base

This section deals with the bottom-most part of a Raft.

Base Feed Rate Multiplier (ratio): 0.75
Does not apply to supports

Base Flow Rate Multiplier (ratio): 0.6
Does not apply to supports

Base Infill Density (ratio): 0.3
Does not apply to supports

Base Layer Thickness over Layer Thickness: 2.2
Does not apply to supports

Base Layers (integer): 0
Since this document deals with supports (without the actual raft), we want to disable it. This value effectively tells Skeinforge to use zero layers for the Base part of the raft.

Base Nozzle Lift over Base Layer Thickness (ratio): 0.3
Does not apply to supports

Initial Circling: (unchecked)
Does not apply to supports

Infill Overhang over Extrusion Width (ratio): 0.1
Does not apply to supports

Interface

This section deals with the uppermost part of a Raft, but does affect supports.

Interface Feed Rate Multiplier (ratio): 1.5
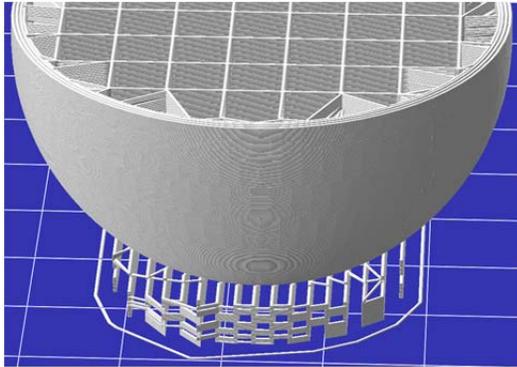Does not apply to supports

---

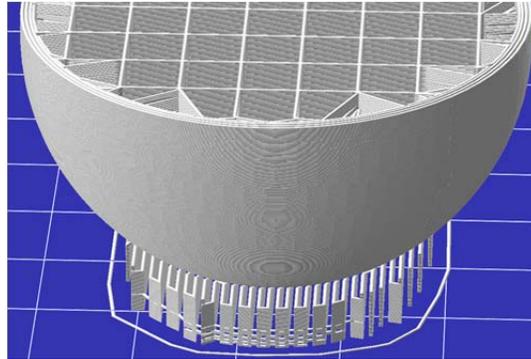## Interface Flow Rate Multiplier (ratio): 0.7
Does not apply to supports

Interface Infill Density (ratio): 0.4
It's a bit counter-intuitive, but this setting actually affects supports. Since supports would be built off of the raft (if it existed), the support density uses this setting. Here are a few examples of what this setting does:

Using 0.4:   Using 0.8

As you can see, a larger ratio provides a denser zig-zag pattern of supports. The setting of 0.4 is the default, and works fairly well. A couple of things to consider, regarding this setting:

- Denser supports will take more time and use more plastic
- Denser supports will lessen the bridging distance of the finished model, usually resulting in a better bottom surface
- Denser supports will distribute the bearing load a bit better, possibly making them easier to remove. The word "possibly" is used, because there are a lot of variables that go into making a support easy to remove.

One of the key components in deciding what density to use is the shape of the item you're supporting. In the example above (basically, a sphere), the object will grow gradually as the layers progress. So the perimeter loops are the only parts that really require support (and they don't need much). On the flip side, let's say you had an object that suddenly cantilevered horizontally by 25 mm. In that case, the perimeter/loops/infill would all require support. There's a candidate for increasing the support density.

Interface Layer Thickness over Layer Thickness: 1.8
This setting does affect supports, but I've never been able to figure out exactly how or why. The official description is:

> Defines the ratio of the height & width of the interface layer compared to the height and width of the object infill. The feed rate will be slower for raft layers which have thicker extrusions than the object infill. Using a value of 1, the base will look the "same" as the normal layer but if you put
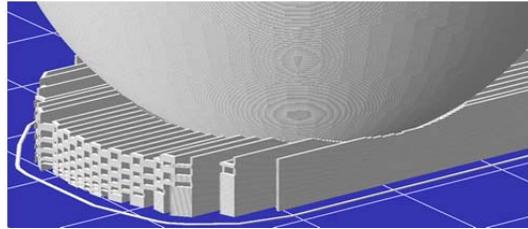
here 1.5 or 2 the base will be "fatter". The "fatness" is achieved by slower movement of the head so more filament per mm is extruded hence fatter the "line."

However, changes to this setting produce quirky results. Here's an example of what happens when you change it:
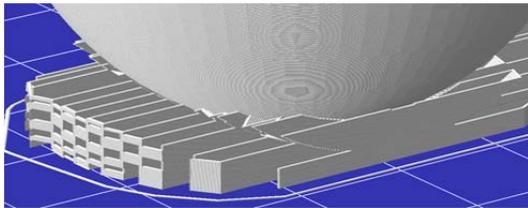
Using 1.8   Using 1.0

As you can see, the density of support lines has increased, as well as the general shapes. There's nothing wrong with tinkering with this setting, but it doesn't appear to behave consistently.

Interface Layers (integer): 0
Just like the Base of the Raft, we also don't want interface layers (the upper part of the raft). So we disable it by telling Skeinforge to use zero layers for the interface.

Interface Nozzle Lift over Interface Layer Thickness (ratio): 0.3
Does not apply to supports

Name of Alteration Files
This section allows you to specify a partial gcode file that will be inserted to your model. I don't know of any useful alteration file for supports, so these will be ignored.

Name of Support End File: support_end.gcode
No changes needed, we're not using alteration files

Name of Support Start File: support_start.gcode
No changes needed, we're not using alteration files

Operating Nozzle Lift over Layer Thickness (ratio): 0.0
Does not apply to supports

Raft Size
This section also deals with Rafts, but affects Supports.

Raft Additional Margin over Length (%): <mark>2.5</mark>

This setting, in addition to "Raft Margin", changes the over-run past the object. Increasing the value will cause the support structure to run further (it is calculated as a percentage of the length of the nominal length). The default setting works fine for supports (it only needs to be adjusted for rafts). Here's an example.
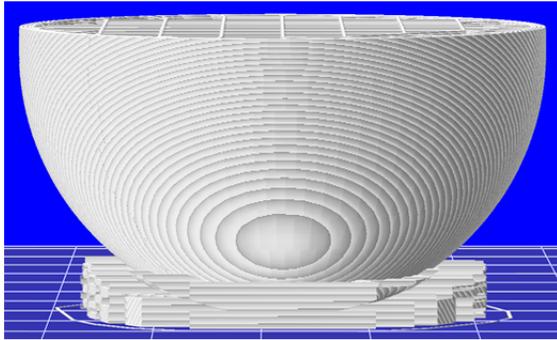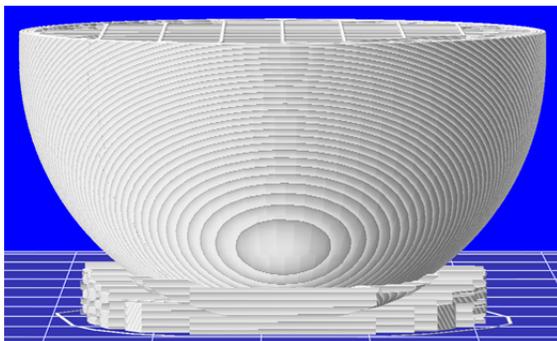
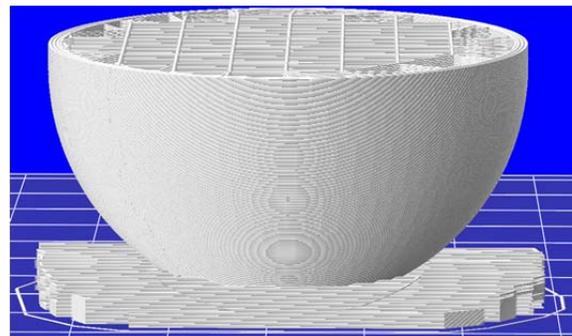Using 2.5   Using 10.0
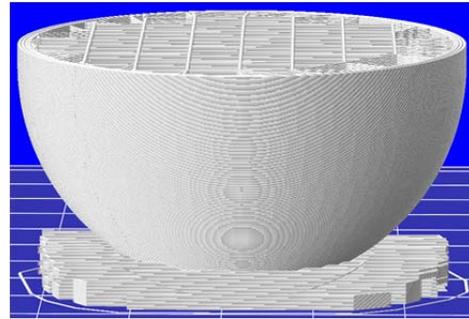
Raft Margin (mm): <mark>2.5</mark>

This setting adds a specified length to the otherwise-calculated support length (see "Raft Additional Margin over Length" above). The default setting works fine, but if you're tempted to play with it, here's an example of how it will look:

Using 2.5   Using 10.0

Support

Support Cross Hatch: <mark>(unchecked)</mark>

As you can probably guess, checking this box will create cross-hatched supports (90 degrees to each other, every layer). Not recommended, unless you like cutting yourself while cursing.

Support Flow Rate over Operating Flow Rate (ratio): <mark>0.72</mark>

This is a pretty important setting. It determines how much plastic is extruded for each line of supports. A setting of 1.0 would extrude it like a "normal" wall. The default is 0.7, which makes for a nice, light support wall. But at accelerated speeds, I've found it necessary to bump the setting just a bit. Otherwise, I get random areas that are wispy. Oddly enough, the wispy parts have never caused

problems in the final print, as they always mend themselves. Here's a picture of supports with a 0.7 flow rate.

Yes, this was on blue tape (not recommended for raftless supports unless you've done a good job prepping the tape). See the wispy part?

Support Gap over Perimeter Extrusion Width (ratio): <mark>0.005</mark>

As the title implies, this is a ratio. Using a value of 1 would set the supports back by the distance of a full perimeter extrusion. As I understand it (without intensive testing), it only applies to inclined surface. In other words, supports underneath flat area will not be affected. Change this at your discretion, but keep in mind that increasing the ratio will do two things: Make it easier to remove, and allow the overhang to move a bit before settling onto the support. If you play with it, keep the ratio low (maybe below 0.5).

Support Material Choice: <mark>Exterior Only</mark>

The choices are as follows:

- None
- Empty Layers Only
- Everywhere
- Exterior Only

Those are pretty descriptive. The "Empty Layers Only" option isn't terribly useful – only if you want to raise the item off the bed for some unknown reason (or stack items on top of each other). "Exterior Only" will only provide supports that come from the build plate. "Everywhere" will also generate supports within an item – that might be a good option when a part hangs over another part.

Note: ReplicatorG (both the MBI and Sailfish versions) have this option in Print-O-Matic. So regardless of what you choose here, PoM will override it (assuming you've enabled Raft/Supports). PoM only offers the following choices (None, Exterior Support, and Full Support). Using Makerware, you need to edit the profile and make this change manually.

Support Minimum Angle (degrees): <mark>55.0</mark>

This setting specifies how much of your model needs to be supported. It is an angle taken from vertical. If you use 1 degree, then practically everything except vertical walls will get support. Conversely, a setting of 89 degrees will generate supports for horizontal overhangs only.

The standard rule of thumb is 45 degrees. I've found that this can be cheated a bit. 55 is a good number for me, but it is also subject to the use of multiple shells. See below (Shells and Thread Sequence Choice) for more info.

Other Skeinforge plugins that will affect Supports

There are tons of plugins, each with tons of settings. It goes without saying that other plugins will affect the generation of supports. Here are a few that you should know about.

Shells

This setting resides in the "Fill" plugin (under the "Extra Shells" section). It is really three settings – one for Alternating Layers, one for Base Layer(s), and one for Sparse Layers. If you use ReplicatorG (MBI or Sailfish), then Print-o-Matic will override these settings.

If you're new to Skeinforge, keep in mind that "shells" really means "extra shells", as you'll always have one shell. So a setting of 1 means you'll print two.

When your model has overhangs, you definitely want to use extra shells. Maybe you don't need extra shells if your model tapers at a 10 degree angle outwards. But if you go much further, you'll end up with gaps between layers, where you can see the infill at the overhangs.

Thread Sequence Choice

Also in the "Fill" plugin, this setting could affect how well supports work. I recommend "Loops > Perimeter > Infill". There are a few other settings that can work well, but the important part is that you print loops before perimeters (when overhangs are involved). This allows the outermost bit (the Perimeter) to cling to something (the first inner Loop). If you did the reverse (Perimeter then Loops), your perimeter would have nothing to cling to – at least, initially. So it would simply fall down until it hit the supports. You'll ALWAYS get a nicer finish on overhangs when you use Loops before Perimeters. There are some other good reasons to use Loops > Perimeter > Infill, but I'll leave that discussion for a different document.

Skeinforge Supports vs. Makerware Supports

I initially considered doing a full write-up on configuring supports in Makerbot Slicer (aka Miracle Grue). But that's better left for a separate document.

The burning question in everyone's minds is: Which slicer is better for generating supports? It's a tough question to answer, since there are many scenarios where one might work better than the other.

It is my opinion that Skeinforge is currently better at this. Partially, because you have more control over the process (for example, you can specify the minimum angle whereas Makerware just puts it everywhere). But it is also because Makerware still seems to have a few bugs. I hope that this will change in the future.

Unfortunately, Makerbot Industries has made a conscious effort to "keep it simple", which by virtue of its definition means you have less options. Once you've gotten through the initial learning curve of 3D printing, you will invariably start looking for ways to print "better". This is where Skeinforge shines: It provides you with options. Yes, it's clunky to configure. And yes, it's overwhelming at first. But you have more options.

Skeinforge Bugs

Skeinforge does exhibit buggy behavior occasionally.

Extraneous supports

Occasionally, Skeinforge will produce supports in places that aren't necessary. In the example pictures below, you can see that small support remnants were produced outside the footprint of the item (which tapered inwards as it went upward, like an inverted cone). Supports had been generated to carry an inset down low, as well as the upper sections that exceeded 60 degrees. It did not cause any problems with the model, just a few extra bits to remove.
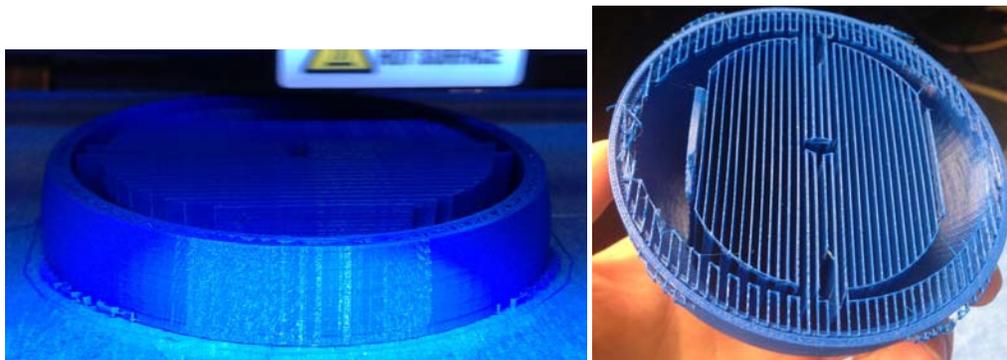
Conclusion

As you can see, there are many settings that affect the final layout of supports. You can control the following general characteristics:

- Spacing of support lines
- Thickness of supports
- Overrun past the object
- Shape (linear vs. cross-hatched)
- Minimum angle for which supports will be generated

- Gap from supports to the model

Aside from individually manipulating the auto-supports, you really can't ask for much more. So it's really a matter of figuring out which combination of settings you prefer.

References used in creating this document:

http://fabmetheus.crsndoo.com/wiki/index.php/Skeinforge_Raft

http://www.makerbot.com/support/replicatorg/documentation/skeinforge/