

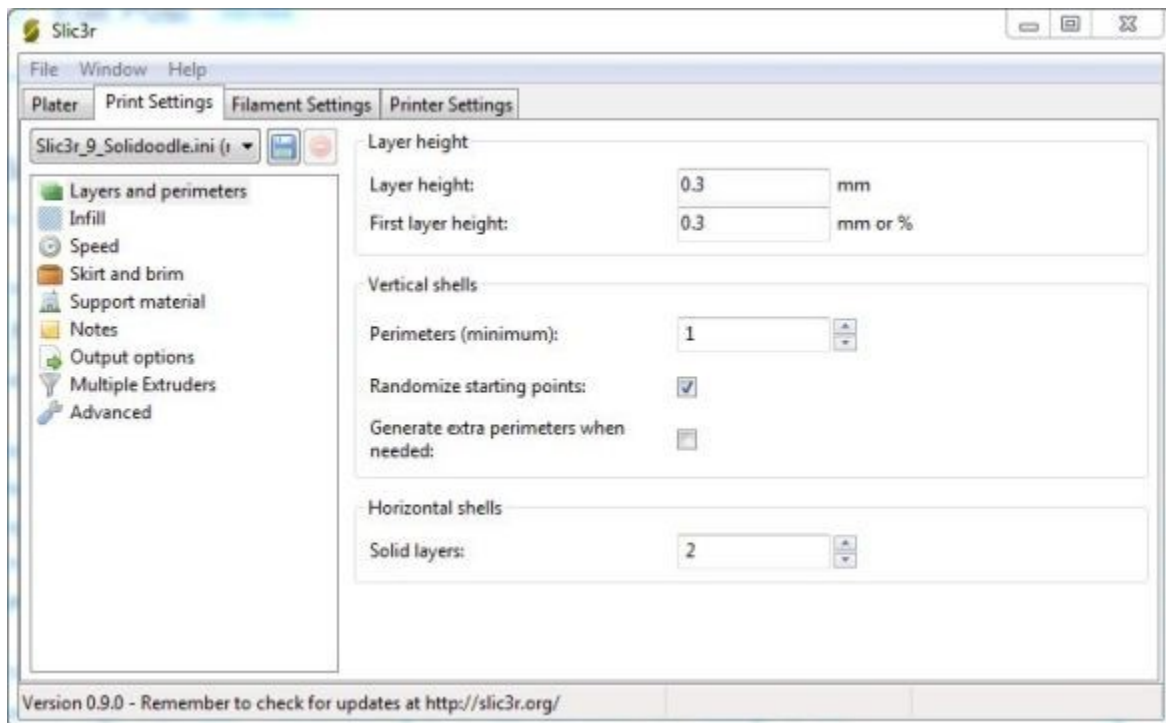
Setting the Flow Rate

Posted on [August 16, 2012](#) by [Ian Johnson](#)

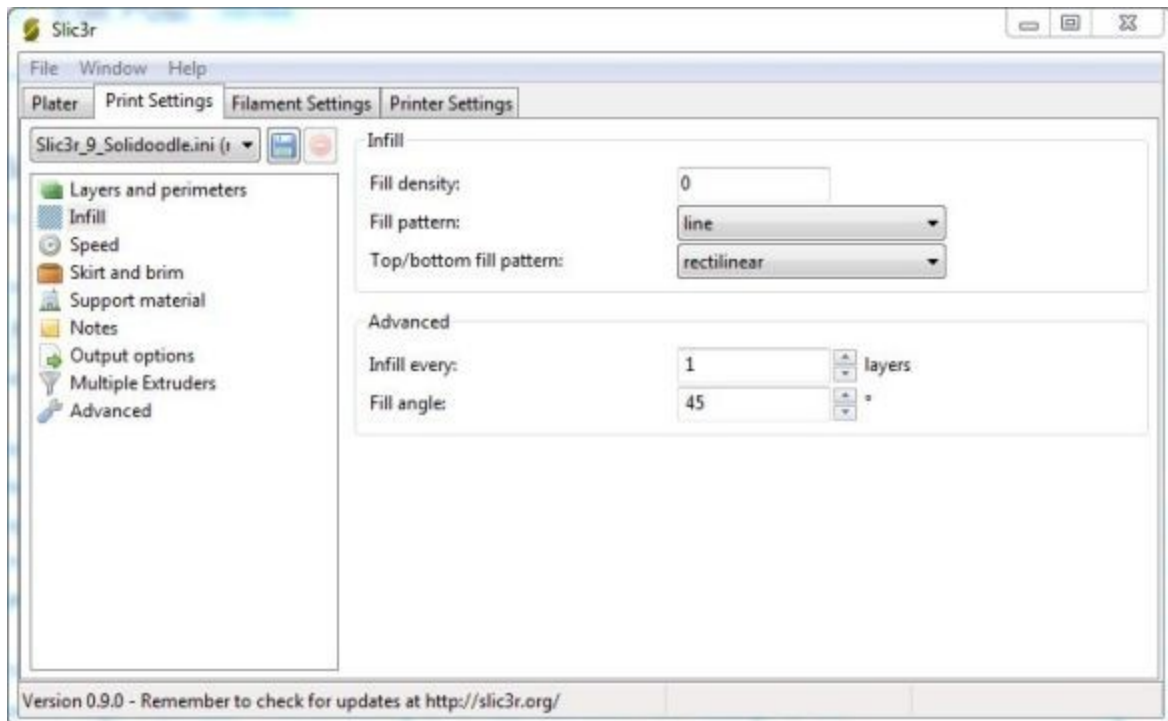
Once you get your extruder calibrated, you know that the right amount of plastic is going in, but you still need to make sure that the right amount of plastic is coming out. Slic3r will run the extruder at the speed that it has calculated will deliver the right amount of plastic to create the thread width it has planned for. However the real world doesn't always match up to those calculations.

When you are printing at .3mm layers, a good Width over Thickness Ratio is 1.4, meaning the thread is 1.4x wider than its height. This would be .42mm, and if the flow setting is right, the printer should be delivering threads at this width. Simply running the extruder and measuring the filament that comes out isn't enough, because the thread is shaped by getting pushed down onto the layer below it.

The best way to check that a given flow rate is delivering the right amount of plastic is to print a single walled part. Pick something like a cube, and slice it with settings for one perimeter, 2 solid layers and 0 fill. When it prints, the walls of the cube should be no thicker than a single thread.

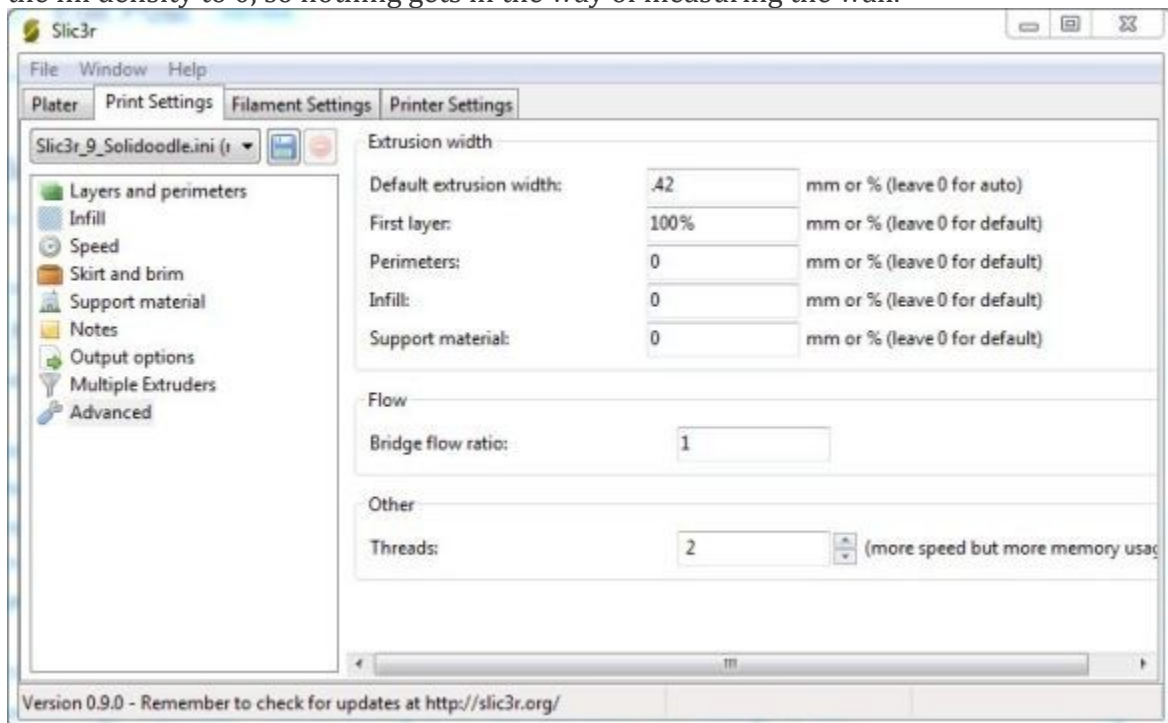


Under Layers and Perimeters, set the minimum perimeters to 1, and the layer height to .3. Make sure the "Generate extra perimeters when needed" is unchecked, or else every few layers Slic3r will throw in an additional perimeter. Set Solid Layers to at least 2. You can discount the first layer because that is affected by the Z offset, but you want to see it run at least one solid layer on top of that to judge how well the threads are fitting together.

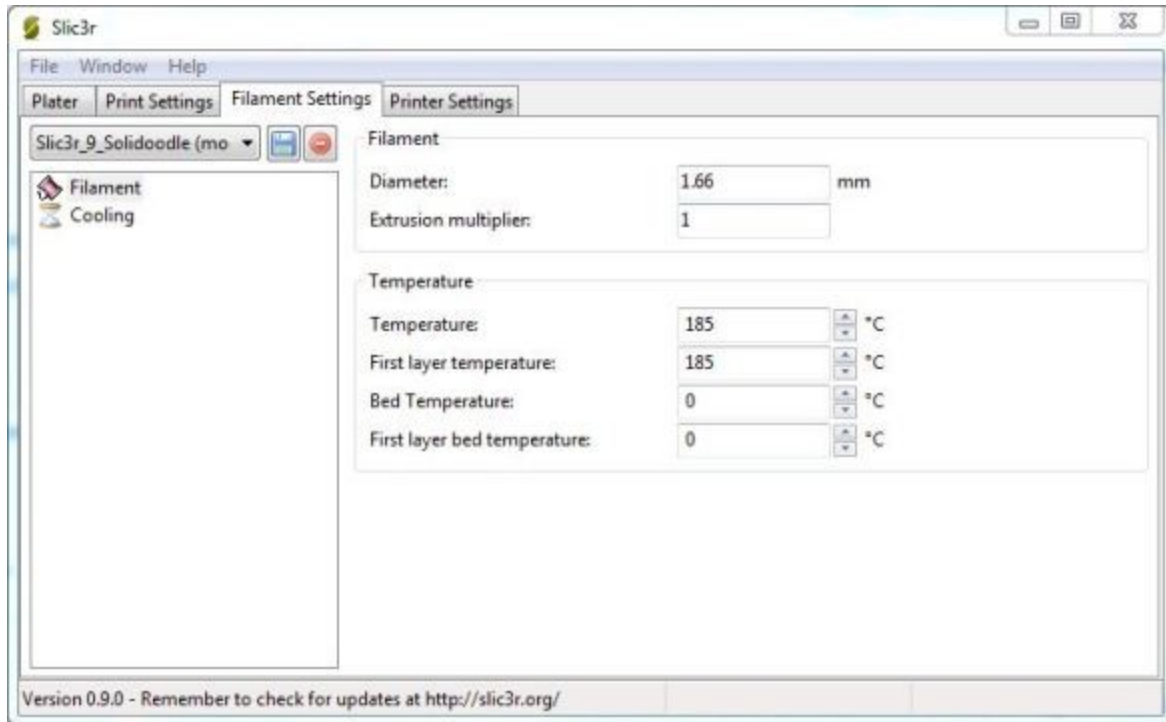


Set

the fill density to 0, so nothing gets in the way of measuring the wall.

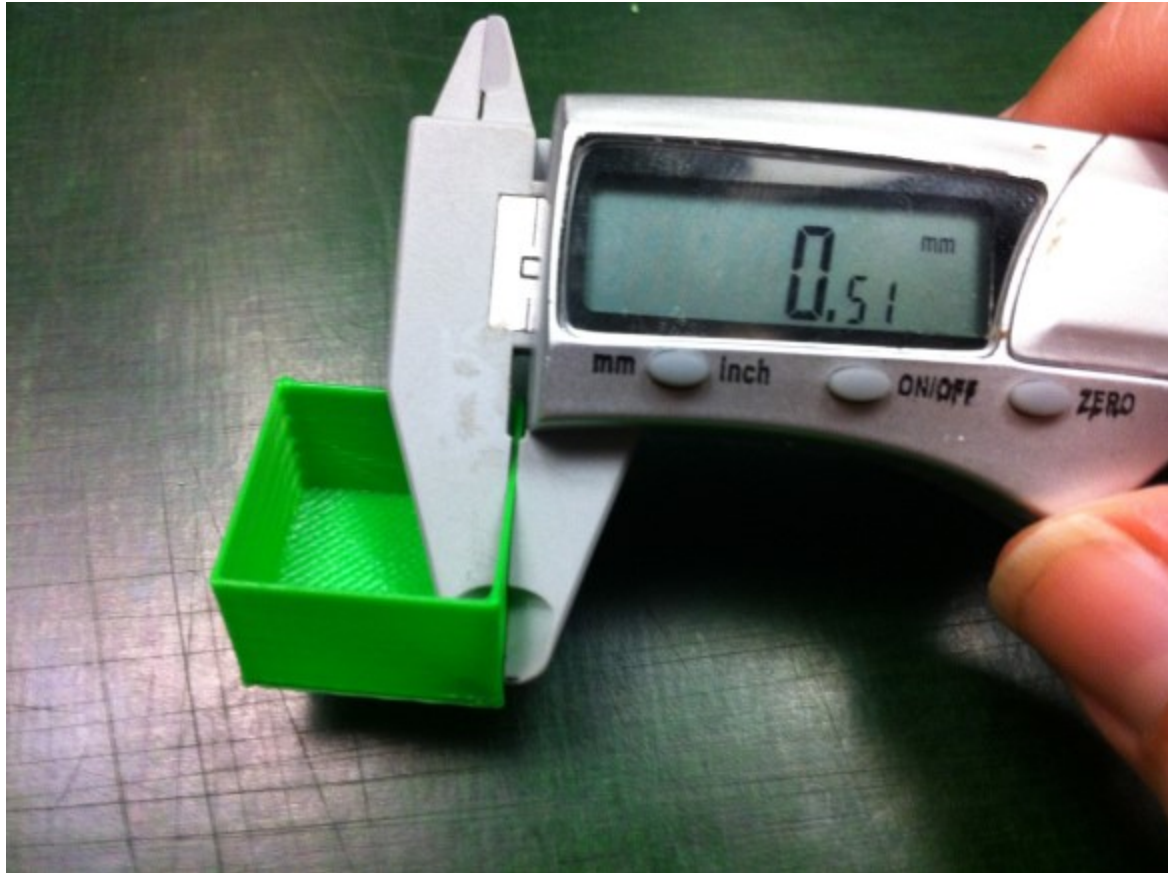


Under Advanced, set the Default Extrusion Width to .42. Slic3r can calculate this automatically, but if we set it explicitly, we will know what width we are looking for.

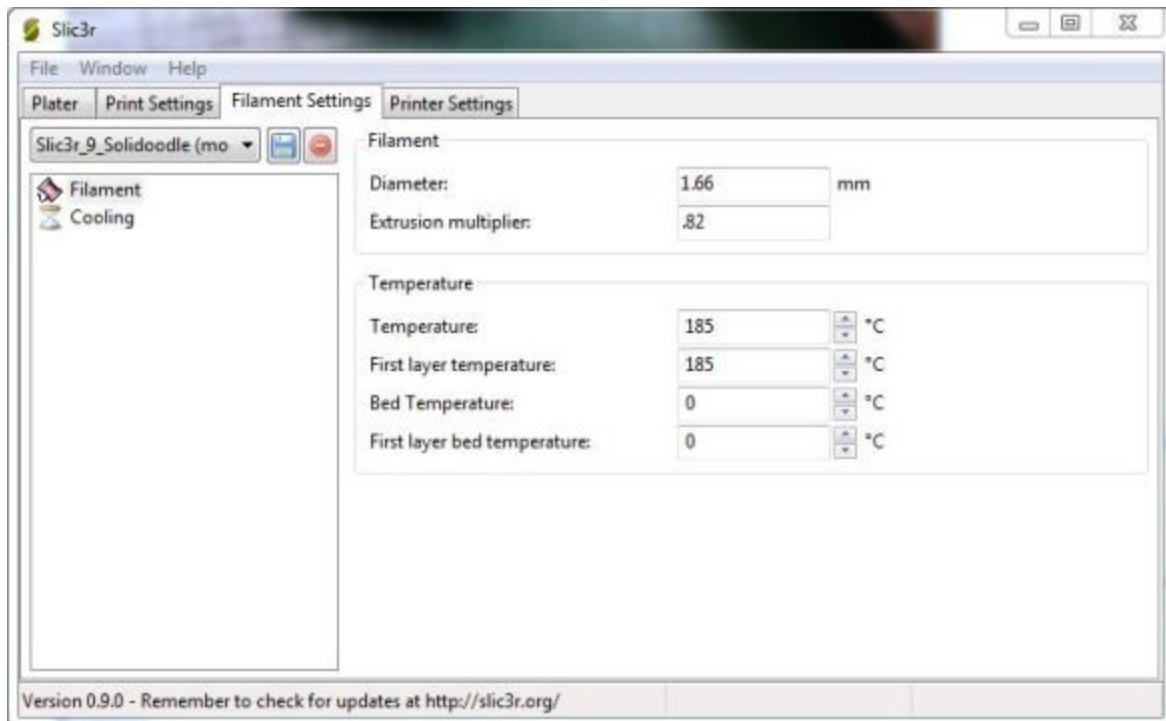


Extrusion Multiplier should be set to 1. This is where you make adjustments in case the flow calculations are not quite correct. We will see how it prints at 1, and then change it if needed.

Print the cube and stop it before it begins printing the solid layers for the top. Find the most perfect, best aligned wall and measure it with a caliper.



The first try came out to .51, which means too much plastic is getting extruded. It's a small difference, but if you want to make high resolution prints at a .1mm layer height it can become noticeable. Our target of .42 is about 82% of .51, so set I set the Extrusion Multiplier to .82. This will tell Slic3r to scale back the results of its computations by 82%.



In Skeinforge, this is a matter of adjusting the Flow Rate under the Speed tab. You could try changing the current setting by the same percentage as calculated above. For instance if my

setting in Skeinforge was 2.7, I could try dropping it by 82% to 2.21. I haven't tried it with Skeinforge to see how well it works, but I suspect it would take a little more trial and error.