**Finding Max Possible Printing Speed**

So my interest the last couple days has been this: how to reach the maximum print speed possible on my printer? I started having some more extrusion problems, and after watching a print closely realized I was under-extruding because some of my print speeds were higher than my hardware could handle.

I'm not talking about finding a speed that produces a certain quality -- I mean finding the maximum possible physical print speed on a given printer before you start to lose precision (no skipped steps, no [extruder](http://www.soliwiki.com/Extruder) hitching, etc.)

So I did some research on this, and it makes sense that your maximum speed is normally going to be limited by the maximum speed at which you can extrude filament. There are several factors that come into play (and maybe more):

* Nozzle size
* [Extruder](http://www.soliwiki.com/Extruder) power
* Frictional losses (due to a bent bowden tube, clogs, or other physical impedence)
* Filament viscosity
* Heater power
* Hotend mass
* Layer height

Filament viscosity is influenced by the type of material, and also the temperature you use. Higher temps (within limits) reduce the viscosity. A larger nozzle means more filament can extrude faster. If your [extruder](http://www.soliwiki.com/Extruder) is underpowered or lacks grip, it may slip or miss steps prematurely before you've maxed out the hotend's limitations. And clearly if you have any physical resistance contributing to that, the [extruder](http://www.soliwiki.com/Extruder) will slip earlier.

Now on the hotend side, the more massive your hotend is, likely the faster it can transmit heat to the filament. Likewise the more power your heater uses, the faster it can heat the hotend. As the filament sinks heat from the hotend in order to melt, it cools the hotend, and a bigger heater should be able to compensate faster (assuming your PID is well-tuned). As the hotend cools because of heat conduction, the viscosity of the filament goes up and it gets more likely that your [extruder](http://www.soliwiki.com/Extruder) will start slipping.

So, I think all of that makes sense, and it should take into account most of the physical forces involved here. I'm exploring this topic, so if any of you with more expertise, or perhaps degrees in physics want to add to or correct any of this, please do smile

In my case with my printer, I guessed that my hotend (E3D Lite6) was the limiting factor, with the amount of heat getting sucked out of the nozzle limiting how quickly I can extrude before it becomes too viscous and the [extruder](http://www.soliwiki.com/Extruder) starts to slip.

To get a baseline, I chose a filament that I've been printing with happily at a known temperature: XYZprinting clear red, at 195c. I heated the hotend to that temperature, then extruded a known length of filament at a known feed rate using G1 commands. Using simple math I determined the volume of plastic that came from out from that extrusion, and slowly increased the feed rate until I reached the point where the [extruder](http://www.soliwiki.com/Extruder) slipped.

That math is just the volume of a cylinder:

volume = (filament diameter)/2 \* pi \* (raw extruded length)

So in my case that's about 2.41mm^3 per extruded mm.

Default feed rates are measured in mm/min, so an 'F60' in a G1 command means to extrude at a rate of 1mm/s.

Once I found the feed rate at which slippage occurs, I doubled the length of filament being extruded to check whether it slipped when extruding more filament (sucking heat out of the hotend for a longer time). If it did slip, I backed off the feed rate slightly until I reached a point where extruding 60mm of cold filament did not slip at all. This number was my baseline.

I also repeated the test at three temperatures to confirm whether, and by how much, temperature affected the extrusion rate.

My test showed the maximum feed rate for extruding 60mm of the material at the given temperatures:

@195c: Feed rate 130mm/min (2.17mm/s), volume: 5.21m^3/s

@200c: Feed rate 150mm/min (2.5mm/s), volume: 6.01m^3/s

@205c: Feed rate 170mm/min (2.83mm/s), volume: 6.81m^3/s

There's a clear trend here. I'm not an expert on fluid dynamics, nor did I take more samples than this, so I don't know if this is a linear trend across a wider range of temperatures, but what I have here appears roughly linear.

The problem is that this baseline doesn't account for the pressures involve when actually printing at a given layer height -- this is straight extrusion into thin air -- ideal extrusion.

But this baseline gave me a place to start. From these volumes, I calculated the maximum ideal extrusion speed (mm/s) to reach that rate of extrusion using this simple (and slightly idealized) formula:

speed = volume / (nozzle diameter) \* (layer height)

This means my maximum ideal extrusion rates are:

0.1mm layer @195c: 130 mm/s

0.2mm layer @195c: 65 mm/s

0.3mm layer @195c: 43 mm/s

0.1mm layer @200c: 150 mm/s

0.2mm layer @200c: 75 mm/s

0.3mm layer @200c: 50 mm/s

0.1mm layer @205c: 170 mm/s

0.2mm layer @205c: 85 mm/s

0.3mm layer @205c: 57 mm/s

Now, I never expected actual print speeds to match this, so I started doing some empirical testing. I started back at 195c and started plugging values into [Slic3r](http://www.soliwiki.com/Slic3r). I set the Perimiters, Infill and Support Material speeds to my chosen value, and used a test model consisting of a sphere 60mm in diameter sitting atop a disk to act as a [raft](http://www.soliwiki.com/Raft). I used 30% infill. This model let me see the perimeters (outside of the sphere), infill (inside of the sphere) and 30% rectilinear support material (around the bottom of the sphere) to see if the extrusion could keep up under each condition.

I watched the print, and if I heard anything more than rare slippage in the [extruder](http://www.soliwiki.com/Extruder), cancelled the print and backed off the speeds a bit until I achieved a speed where slippage was rare (at most one or two "clunks" per layer).

Interestingly, I found that infill was the hardest thing for my [extruder](http://www.soliwiki.com/Extruder) to keep up with. It normally did better with support material, but I think that's because support material is often more sparse, so there's more chance for the extruded plastic to squeeze out around the existing layers rather than being smushed down on top of the last layer. Likewise for the bottom half of the sphere where the perimeter is open, and in this shape often being laid on top of support material.

Empirically, at 195c, I arrived at these maximum speeds and corresponding extrusion rates:

0.1mm layer height: 50 mm/s = 2m^3/s

0.2mm layer height: 40 mm/s = 3.2m^2/s

0.3mm layer height: 40 mm/s = 4.8m^3/s

Anything higher than this produced an unacceptable level of [extruder](http://www.soliwiki.com/Extruder) slippage.

At 0.1 and 0.2mm layer heights, these values are close to half the idealized ("extrude into thin air") rates. At 0.3mm layer height, it seems we're getting into territory where there's enough room and low enough back-pressure to reach at least 90% of the idealized extrusion rate (which was 5.21m^3/s, which you can think of as roughly a 0.4 layer height, which is the nozzle diameter).

I'm thinking that this kind of procedure to measure the maximum extrusion rate at a given temperature would be useful for calibrating any new material you try to print with, or any new hotend or modifications affecting the hotend or [extruder](http://www.soliwiki.com/Extruder).

For example, we all know that 190-220c is the recommended temperature range for most [PLA](http://www.soliwiki.com/PLA) -- but how do you decide whether to use the lower temps or higher temps? This might be one way. Run this extrusion test to find out what temperature gives you a comparable ideal extrusion rate to some reference material that you know works well.