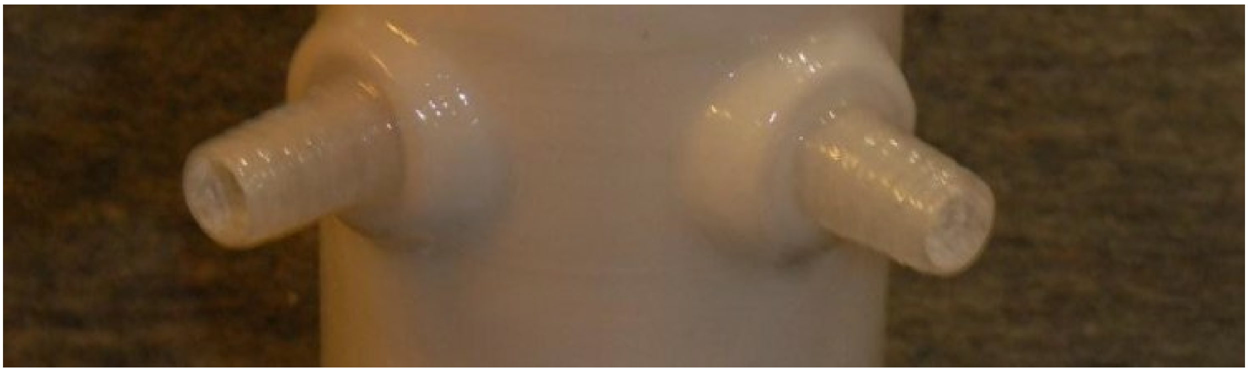
Speeding up that Large Objects: Maximum Flow and a 0.8mm Nozzle

craftbot.nl/2015/06/29/speeding-up-that-large-objects-maximum-flow-and-a-0-8mm-nozzle



I have been printing really large PLA objects the last month, mainly Reactors for the Laboratory. They are basically Pots with lids and a couple of tulle's. They are varying in height between 20 and 30 centimeter. Printtime per Reactor was about 30 hours.

There was a demand to make Reactors with a higher temperature resistance @60 degrees. As PLA allready softens @50 degrees I had to test other print materials, which had a higher Glas Transition Temperature (GT temp= the point meterial softens, become rubbery. The melting point is much higher).

I made a **comparison of the Glass Transition Temperature** of that materials:

Material:	PLA	ABS	Easyfill ABS	HIPS	REAL PETG	PETG	Colorfab XT-copolyester	FlexiFill	Crystal Flex™	PolyCarbonate	Taulman Nylon 645	Taulman Nylon 645	T-Glase
	EasyFil™ PLA - PolyLactic Acid	Acrylonitrile Butadiene Styrene	EasyFil™ ABS - Acrylonitrile Butadiene Styrene	EasyFil™ HIPS - High Impact Polystyrene		HDglass™ (PETG compounded)		Thermoplastic Co-Polyester	Crystal Flex™				
Transparency:	Opaque	Opaque	Opaque	Opaque		Transparent		Opaque	Transparent				
Diameter:	1.75 mm	1.75 mm	1.75 mm	1.75mm		1.75 mm		1.75 mm	1.71 mm				
Diameter Tolerance:	± 0.05 mm	± 0.1 mm	± 0.05 mm	± 0.05 mm		± 0.05 mm		± 0.05 mm	+0.09 mm / -0.11 mm				
Print Temperature Guideline:	± 210° C	~235°C to 256°C	± 220-260°C	± 235°C	250	215 - 225°C	240-260	± 210° C	± 240°C	280°C	242°C	245°C	228°C
HPB	60°C	110°C	110°C	85°C	30	60°C	70	60-100°C		115°C	60°C	60°C	68 °C max
Melting Temperature:	210° C ± 10° C	~235°C to 256°C	240° C ± 10° C	180°C - 260°C	245			180° C	~215°C - ~240°C		217	218	207
Glass Transition Temperature:	± 50°C	~ 85° C	± 103°C	± 89°C	70°C < GT < 80°C	70°C < GT < 80°C	75	-	-	120?°C	52°C	48°C	76°C
Ovalidity Tolerance (max):	≥95%	± 5%	≥95%	± 5%		± 5%		0.05 mm	± 5%				
Density (21.5°C):	1240 kg/m³	1010 kg/m³	-	1040 kg/m3		1270 kg/m³		1150 kg/m³	1020 kg/m³	1200			
Print Speed Guideline:	40 - 80 mm/s	40 - 80 mm/s	40 - 60 mm/s	40 - 60 mm/s		40 - 80 mm/s		10 - 20 mm/s	40 - 80 mm/s				
Impact Strength:	7,5 KJ/m2	16 KJ/m2	36 KJ/m2	15 KJ/m2		7.2 KJ/m2		-	3 KJ/m2	11-60 kj/m2			
Tensile Modulus:	3310 Mpa (MD), 3860 Mpa (TD)	2000 Mpa	2000 Mpa	1550 MPa		2147.6 MPa		95 MPa	1550 MPa	2350			

As temperature warping (and splitting of layers) with other materials is much more a challenge when printing large objects, I have tested some of them.

Heated Chamber
As droughts // air movement round the print are another cause of temperature warping, I closed the CraftBot as much as possible: at the front (fancy magnet mounted perspex), and added a less fancy cardbox box on top to create a heated chamber.
It turned out that a complete closed chamber was causing an overheated (and clogged) extruder (for the original extruder, as well the E3D hotend), as the temperature raised more than 50 degrees. I cut a hole in the top of the box and was able to keep the temperature of the chamber at 35-40 degrees. This just by heating by a 100 degrees HPB.

For that large print have I tested: Easyfill ABS, HIPS, ABS and PETG.
the results of **temperature warping of materials**:

- absolute drama: ABS and Easifill ABS
- better, but still splitting: HIPS
- best: PETG

PETG prints like PLA, but has a much higher GT. It softened with me only at 75 degrees C. The picture above is actually the PETG print. It's printed in two pieces, as it's 30 centimeter height and sealed with epoxy on the in- and outside. Some of the tulles are printed separately and some directly with support. Both solutions printed well.
BTW: It turned out after all that PETG didn't need that heated chamber for those huge prints.

The printtime > 30 hours is way too long, as Murphy tells the printer that: when it fails it has to fail at least at 90% of the print.
So I have exchanged the 0.4mm nozzle by a 0.8mm E3D nozzle. I tested it both with the original hotend as well with an E3D extruder.

Maximum Flow
First I tested what the maximum flow actually is of the extruder. That's the maximum flow of the melted plastic and is expressed as PathWidth(PW) x LayerHeight(LH) x PrintSpeed(PS). So an standard 0.4mmPW and 0.3mmLH with a printspeed of 50mm/s is resulting in 0.4×0.3×50=6, so a flow of 6.
The melting capacity of the 40W heater turned out quite impressive: there is a maximum flow of 40 possible with PLA and PETG!
To put that in perspective: that would result in a maximum theoretical XY print speed of 333mm/s!

Kisslicer Flow and Speed Calculator

Only fill in the colored fields according the KS input fields!

Stacked Layers	1			
	Perimeter	Solid+Sup	Sparse	Travel
Head Speed	60	80	80	100
Layer Thickness	0,30			
Path Width	0,40		0,40	
Filament Diameter	1,72			
Minimal Flow	0,1			
Maximum Flow	40			
Results				
F value Max Head Speed mm/s	3600	4800	4800	6000
Resulting Flow because of Head Speed mm3/s	7,2	9,6	9,6	
Resulting Feed Because of Head Speed mm/s	3,1	4,1	4,1	
Min Head Speed Resulting of Min Flow mm/s	0,83	0,83	0,83	
Max Head Speed Resulting of Max Flow mm/s	333,33	333,33	333,33	
Min Feed Speed Resulting Min Flow mm/s	0,0620	0,0620	0,0620	
Max Feed Speed Resulting Max Flow mm/s	17,2	17,2	17,2	

I have made that Openoffice Calculation sheet in the past for mainly Kisslicer, but it can be used for all slicers actually.
It can be found here:
Kisslicer Flow and Speed Calculator BtH

Unfortunately: that speed is not possible yet with the CraftBot. I can print at a maximum of 80mm/s as a higher speed is resulting in shifted layers because of the steppers missing steps. Even when turning the Vref of the Stepperdrivers higher.

Testing Marlin with the CraftBot
As I blew up the CB of one of my CraftBots -and had to wait for a replacement, I have temporarely build in a RepRap//Marlin CB (megatronics)
It turned out that the mechanics of the CraftBot are capable to print at much higher speed: 160mm/s actually, as I have tested that with a large (230x190mm) straight object. For that I tweaked the Marlin firmware at 3000mm/s2 for the acceleration and a jerk (derivate of acceleration) at 20.

As discussed elsewhere: the acceleration in relation to the planner in the CraftBot firmware is not optimal yet, probably resulting in random unaccelerated moves, which are causing the missed steps IMO. Luckily CratBot has announced that this will improve in the next firmware release.

Anyway, when using the CraftBot firmware and a 0.8mm nozzle and a max. flow of 40, I was able to print at 80mm/s with a pathwidth of 0.8mm and a layer height of 0.6mm.

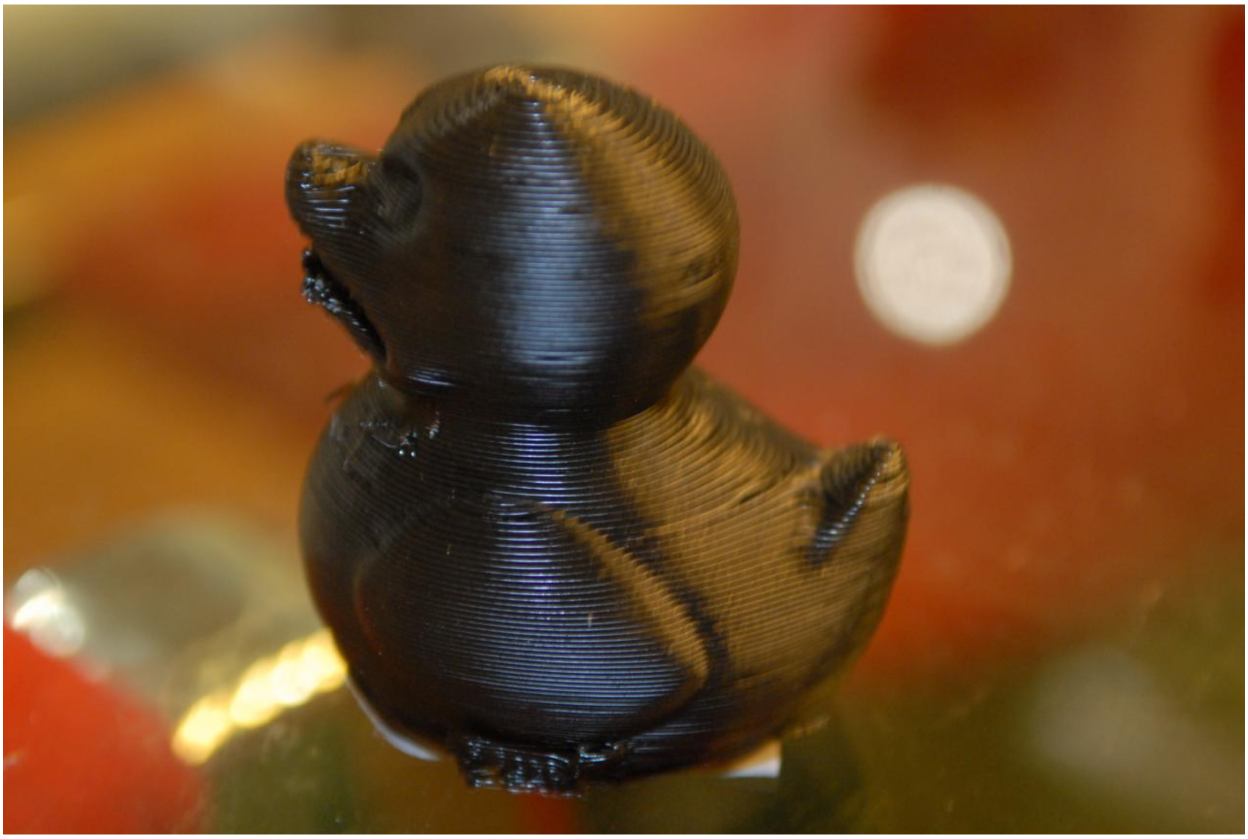
	A	B	C	D	E
1	**Kisslicer Flow and Speed Calculator**				
2	*Only fill in the colored fields according the KS input fields!*				
3					
4	Stacked Layers	1			
5					
6		Perimeter	Solid+Sup	Sparse	Travel
7	Head Speed	60	80	80	100
8	Layer Thickness	0,60			
9	Path Width	0,80		0,80	
10	Filament Diameter	1,72			
11	Minimal Flow	0,1			
12	Maximum Flow	40			
13					
14					
15	**Results**				
16	F value Max Head Speed mm/s	3600	4800	4800	6000
17	Resulting Flow because of Head Speed mm3/s	28,8	38,4	38,4	
18	Resulting Feed Because of Head Speed mm/s	12,4	16,5	16,5	
19					
20	Min Head Speed Resulting of Min Flow mm/s	0,21	0,21	0,21	
21	**Max Head Speed Resulting of Max Flow mm/s**	83,33	83,33	83,33	
22	Min Feed Speed Resulting Min Flow mm/s	0,9916	0,9916	0,9916	
23	Max Feed Speed Resulting Max Flow mm/s	17,2	17,2	17,2	
24					

3 Times faster print with a 0.8mm nozzle!
That is really speeding things up: the print will be ready about three times faster! So my huge print is now ready in 10 hours in stead of 30.
A comparison with a smaller object (my avatar, the evil duck of 80mm height) and different Layer heights with the 0.8mm nozzle:

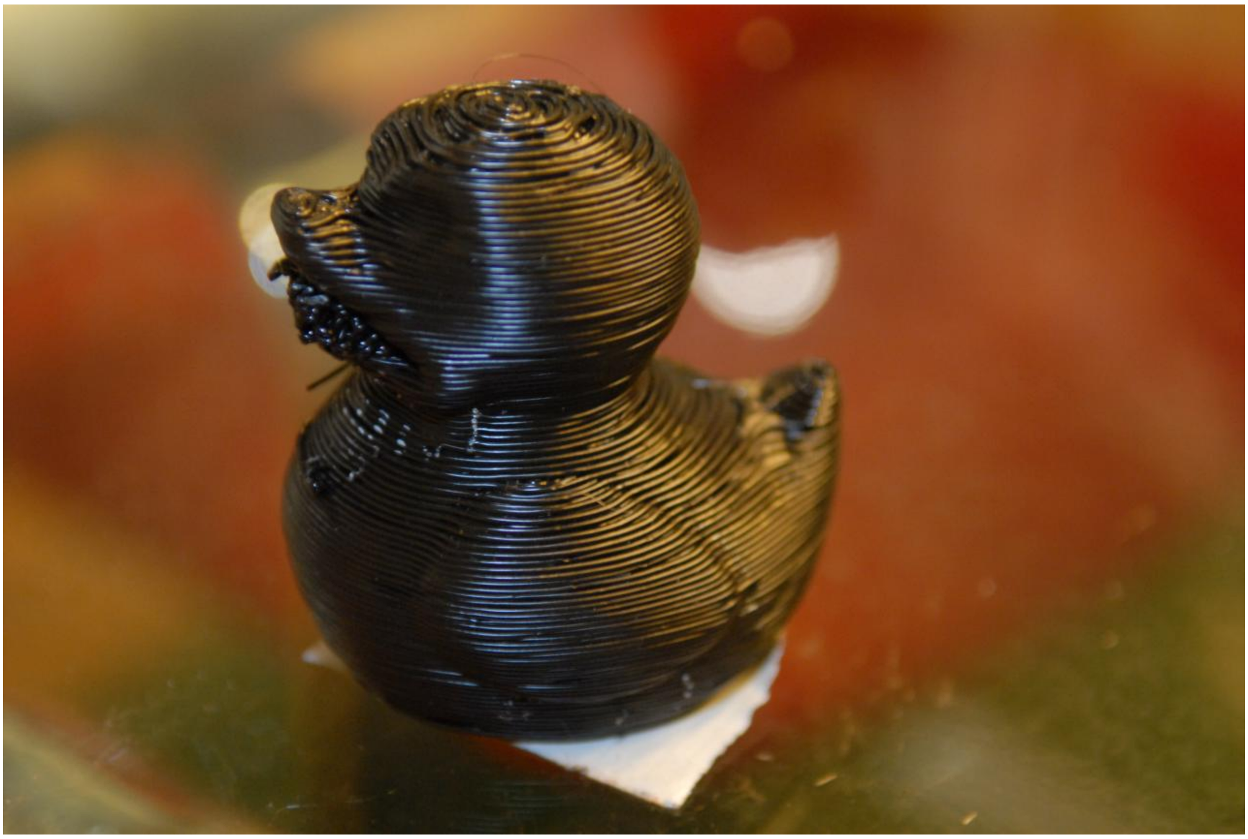
Layer Height	PathWidth	Nozzle	printtime	times faster	relative times faster	
0,3	0,4	0,4	126	1		
0,6	0,8	0,8	43	2,93	2,93	
0,8	1	0,8	34	3,71	1,26	
1	*1,2*	*0,8*	*31*	*4,06*	*1,1*	*not printable*
printing with flow max.=40, perimeter 60mm/s, (solid) infill: 80mm/s						

I have printed all those combinations (and some more): the best 0.8mm nozzle result was with 0.6mm Layer height, an acceptable result with 0.8mm Layer height, and an unexpected curly result with 1mm layer height (although my kids love it ;-)

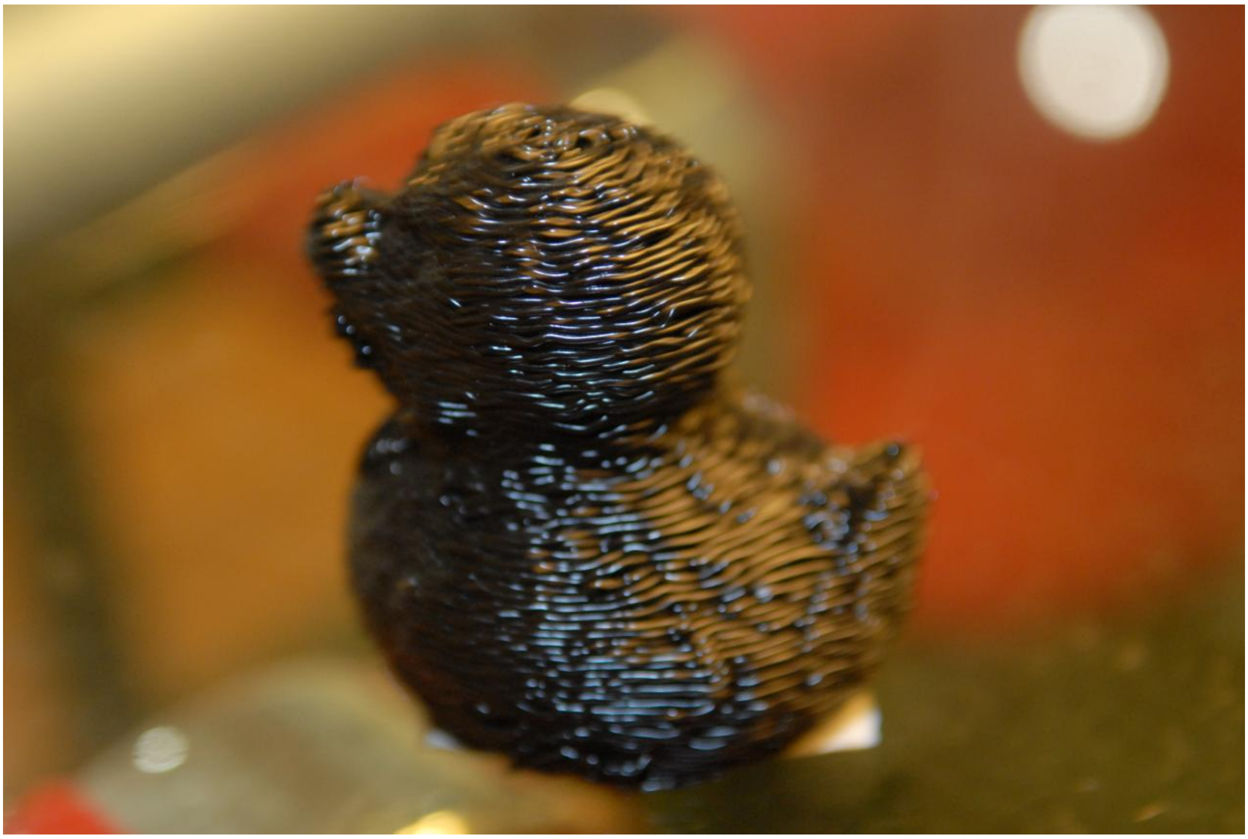
Some pictures of the 80mm evil duck test:



2.9 times faster than a 0.3mm LH print: **0.6LH 0.8PW flow 40** = about 80mm/s for the infill (long straight lines). The loops never reaching that speed for the tiny segments, as the acceleration is restricting the speed than.



0.8LH 1.0PW flow 40 3.7 times faster than a 0.3mm LH print.



1.0LH 1.2PW flow 40 a nice curly duck ;-)

About the results: of course is printing with large layer heights not intended for ‘injection mold look’ prints. They are just for more technical prints when looks are less important than an usable result. (actually, that’s how my wife picked me…)

The only thing I have changed in the slicer settings (except the max flow of 40) is a larger overlap of infill and loops (80%) to get a better connection between them.

Investment for all this:
I hope others can benefit of these tests. All you have to do is to buy a (only a measly 13 Euro) 0.8mm Nozzle, and you are good to go!

Bart