ProtoCycler

Extrusion Theory Guide

Rev. 1.0

ReDeTec

Table of Contents

Extrusion Theory Guide

Purpose

The purpose of this guide is to help users find the optimal settings for extruding new plastics that they may have little or no experience with. Extruding plastic is a relatively complex subject matter, and it's important to note that this guide is just a brief nudge in the right direction, to help you get started – in no way is it a comprehensive explanation either of plastic extrusion, or ProtoCycler itself. That being said, if you're completely stuck at even getting plastic to come out, this guide should help you get things into the rough ballpark. Please note that getting quality filament can be a time intensive process, with many iterations required to get things "just right". Patience is definitely a requirement! There are also times when you may be forced to go backwards a few steps based on something you've learned at a later stage. That being said, following the procedure outlined below should help you get to useable filament than without.

Safety

Never extrude plastics that you do not fully know and understand the MSDS for. If you don't know what an MSDS is, you should not be experimenting! Many plastics can release harmful and even lethal fumes. **Always** ensure you have adequate ventilation when experimenting with new plastics, and ensure you are fully aware of the risks being taken.

Similarly, please note that you may well void your warranty or completely damage your machine by attempting to extrude any new plastic. While we want to encourage experimentation, we are not able to cover any damage caused from using the "manual" extrusion option of ProtoCycler – only automatic profiles released by ReDeTec are certified for use with ProtoCycler. Abrasive additives such as carbon fiber, metal powder, etc are particularly damaging and should be avoided at all costs.

Procedure

1. Pressure and Temperature

The first step in getting quality filament from your ProtoCycler is to tune the pressure and temperature, so that the plastic can flow at the right rate, and at the right viscosity. The viscosity is purely a function of temperature, with the flow rate also depending on pressure. It's generally best to find a "sweet spot", where both the temperature and pressure are in the middle range of what would work. However, some plastics prefer to be heated a bit more and then extruded at a lower pressure, while others prefer to be heated slightly less and extruded at a higher pressure. It goes without saying, of course, that the plastic should be 100% clean and dry! It is also always recommended to use virgin pellets when experimenting with a new plastic. Last but not least, the spooler should not be used when experimenting with new parameters.

To set the temperature, consult the datasheet for the plastic in question. Note that extrusion temperatures for ProtoCycler can vary drastically from those used in 3D printing, and so it's important to know what the datasheet actually recommends – not just what you're used to printing it with. Be sure to disable minimum pressure limiting by sending the command "pl000000", no quotes. Then, set the pressure to 50. This is enough that the plastic should begin to flow once it's warm enough.

At this point, start at the lowest recommended temperature on the datasheet. Once the plastic is at temperature, it should begin to flow through the nozzle. If not, or if the flow is very slow, continue to increase the temperature in 5 C increments until the plastic is flowing adequately through the nozzle, dumping into the garbage chute. At all times, be sure to note the apparent viscosity of the plastic – if it's dripping like liquid, it is not viscous enough. If it's coming out almost rigid, it is likely still too viscous. The plastic should curve and fall away from the nozzle into the garbage chute in an acceptable way.

Once you have a baseline for the temperature, it's time to set the pressure (and potentially re adjust the temperature accordingly). We find most plastics extrude best in the 60-70 range on the pressure slider. However, it is important to note the motor *speed* that is required to maintain this pressure! If the motor is spending most of its time at maximum speed – 200 or higher – than the viscosity is too low, and the temperature should be lowered. Similarly, if the motor is spending most of its time at low or minimal speeds – 100 or lower – then the viscosity is too high, and the temperature must be increased. Ideally, the motor spends almost all of its time between 100 and 200.

Once the viscosity has been correctly set above, it's time to turn the auger lower pressure limit back on. Set PL to 50 using the command "pl000050", no quotes. Note that PL can, and sometimes should, be adjusted to suit the shear strength and elasticity of various plastics. However, tuning PL is covered in advanced parameters, below.

2. Flow Rate, Sensor Alignment, and Cooling

With plastic flowing out the nozzle, it's time to check the sensor alignment, flow rate, and cooling. As a safe starting point, set the puller speed to 45 on the slider, and the fan to maximum. ***Never attempt to pull plastic without the fan engaged, even for plastics that don't like being 3D printed with cooling fans!*** Trim the filament at the nozzle and remove all of the waste extrudate from the garbage chute. At this point, you should be able to start feeding the filament between the fan, puller guides, and puller wheels.

The overall goal of this second phase is to ensure a few things. The first is that the diameter sensors – particularly the nozzle sensor – are well centered on the plastic. Some plastics resist being stretched more than others, and will have increased tension in the filament as it's stretched by the puller wheel. These plastics will ride "higher" on the nozzle sensor due to the increase in tension. Similarly, other plastics will stretch out more easily, having lower internal tension, and riding lower on the nozzle sensor. It's imperative that the nozzle sensor is always able to accurately read the diameter, or it will be impossible to get quality filament. The second important check in this stage is that the flow rate is an acceptable value. If the flow rate is too low, the system will not be able to respond correctly to changes in filament, and the filament will cool and harden too quickly. If the flow is too high, the system will be unable to react quickly enough, and the filament will not harden enough by the time it reaches the puller wheels. Last but not least, the cooling rate for the plastic also needs to be determined.

To start, monitor the final filament diameter. While it doesn't have to be 1.75, it should be somewhere very roughly in that range. If it's consistently 1.5 or less, lower the puller speed until it varies between 1.5 and 2.0mm. Similarly, if it is 2.0 or more, raise the puller speed until it varies between 1.5 and 2.0mm. Once the filament is pulling down to roughly the correct size, we can set the nozzle sensor height and alignment – refer to your user manual, [INSERT SECTION HERE], for this procedure.

With the nozzle sensor aligned, the flow rate can now be accurately measured. Wait a few seconds for it to settle. With most plastics, a flow rate of 100-200 is ideal – plastics that require more cooling, like PLA, may need a flow rate on the lower end of this range, such as 125. Plastics that require less cooling are able to be extruded at a higher flow rate. In some cases, flow rates above 200 are possible, though diameter consistency may suffer. If the flow rate is too high, the temperature and pressure must be decreased. If the flow rate is too low, the temperature and pressure must be increased. *Note that at all times, the temperature / pressure combo must allow the motor speed to remain between 100-200 at all times, as covered in step 1!*

Finally, with the flow rate correctly set, the cooling rate can be established. The goal here is to have the filament cool enough that it completely resists deformation by the puller wheels, but is still able to easily be wrapped around the spool. This is largely a qualitative exercise – use your judgement to ensure that the filament seems pliable by the time it would reach the spooler, but is not being deformed by the puller wheels. Always wait a few seconds when adjusting the fan speed, and never lower it a significant amount – if the filament is not sufficiently cooled, it can get caught in the puller system, and cause potential damage.

Unfortunately, it's frequently the case that you will be unable to reach a cooling rate that works sufficiently…and this generally means re-adjusting the flow rate, and even the temperature, to change how the plastic behaves. For example, you may need to increase the temperature and the cooling rate, and achieve a higher flow rate (which hardens the external shell of the filament more – preventing deformation – while allowing the filament core to remain warmer for longer, promoting good spooling). However, doing this may cause the filament to "droop" too much – which requires a lower temperature or again higher flow rate to counteract. Ultimately, at this stage, it is all a fine balancing act…and becomes less of a science, and more of an art. While it can be frustrating to have everything set just right, only to find out you have to effectively restart, don't despair – you'll quickly get the hang of how all the basic parameters interact, and will be able to start adjusting things on the fly instead of starting from scratch.

3. Diameter feedback

At this stage, you should have some relatively good filament coming out. In fact, this is the best that some of our competition – without intelligent diameter feedback – can actually produce! However, there is still one final step to the puzzle…the diameter feedback. Before enabling feedback, it is extremely important to ensure that everything else works. The puller and nozzle sensors should both have a solid "lock" on their diameter. To ensure that the nozzle sensor is reading correctly, look at the "nozzle history" parameter on the graph. Any vertical lines in its output indicate a failed reading – more than a few every minute will cause issues. Similarly, the diameter lines should not look "fuzzy", or have any wild / sudden fluctuations. While the diameter feedback system is helpful for getting that final stage correct, it cannot account for poor quality extrusion in the first place. Any bubbles, moisture, etc should also be completely absent. Last but not least, the diameter sensing system does not currently support completely clear filament – dye or colourant of some form must be used if your pellets extrude completely clear.

With everything taken care of, there is a good chance that enabling diameter feedback will just work once enabled. However, a few things may need tweaking. If the diameter starts to oscillate out of control, the diameter PID system must be tuned to be less aggressive. Similarly, if the system is still very stable but is not reacting quickly enough, the diameter PID system must be tuned to be more aggressive. Please note that following "auto tuning" methods for this system will not necessarily work, for a number of reasons. Behind the scenes, the algorithm in play is far more complicated than a regular PID system, and so Ziegler-Nichols will not provide sufficient results. Note as well that, with the diameter PID system, *there is no "I" term used at this point.* Instead, simply increase the P and D terms or decrease them accordingly – in almost all cases, D should be ~60-80% of P. The stock values are 0.5 and 0.3 for P and D, respectively, which is a good starting point. Note that a change of 0.05 can have a notable effect, and rarely is a P greater than ~0.6 stable, or one less than ~0.3 adequate.

4. Advanced parameters

Beyond everything previously covered, there are a number of other variables involved in getting adequate extrusion, most of which handle occasional random events, such as a slip / shear recovery, and are not part of the normal steady state extrusion system. The theory behind these additional parameters – such as the minimum pressure limit, flow function influence, and the various timescales, is beyond the scope of this document. However, what follows is a brief explanation of what a slip / shear event is, how it affects the system, and what a few key parameters can do to help alleviate this.

Every so often, a pellet will get caught between the auger and the feed throat of the extrusion system, requiring the auger to shear it in half. This creates an artificially high pressure reading on the pressure sensor, which would normally reduce auger speed – and therefore filament flow – and therefore filament diameter. It's easy to spot a shear event because the auger speed will, for ~10-15 seconds, flat line on its lower limit (set using PL). If the filament stays within acceptable bounds during this period, there is no need to further tune the system. However, if the filament either decreases too much, or surges in diameter once the auger recovers, the system may need further tweaking. The minimum pressure limit term, or "PL", is used to compensate for this. Increasing PL effectively insists that the auger must keep spinning a certain speed. If PL is set too low, it will not maintain a high enough speed to hold pressure constant during a shear event. If PL is set too high, the auger will have an increased chance of stalling, which is very bad and should be avoided. In general, a PL of 30-70 is in the ideal range. If you find that the diameter is swinging too much during a shear event, try increasing PL by 10 (from the default of 50) and monitor the next shear event. If you find that the auger stalls during shear events, try decreasing PL by 10-20. Note that if it's not possible to find a proper balance, you may have to tweak the rest of the extrusion parameters to help – for instance tweaking everything to extrude at a lower average pressure / auger speed will, in general, help both sides of the PL equation (more consistent output with less stalling).