ReDeTec™

ProtoCycler+

EXTRUSION THEORY GUIDE

Table of Contents

Purpose

The purpose of this guide is to help users find the optimal settings for extruding new plastics. Extruding plastic is a complex subject matter, and this guide is a nudge in the right direction. It is not a comprehensive explanation of either plastic extrusion, or the ProtoCycler+ itself. That being said, if you're struggling to extrude plastics in manual mode, this guide should help you get started. Getting quality filament can be a time intensive process, with many iterations required to get things "just right". Patience is definitely a requirement! Following the procedures outlined below will reduce the time it takes to consistently produce quality filament.

Safety

Please read the following bullet points very carefully before any experimentation:

- Never extrude plastics whose MSDS you do not fully know and understand.
- Many plastics release harmful and sometimes lethal fumes when heated.
- Always ensure you have adequate ventilation when experimenting with new plastics, and ensure you are fully aware of all risks associated with the process.
- Abrasive additives such as carbon fiber, metal powder, etc. will damage the machine and should be completely avoided.

While we certainly encourage experimentation, damage caused by manual extrusion settings is not covered by the warranty. To reiterate, please be fully aware of all risks associated with your experimentation process.

Procedure

<u>Pressure and Temperature</u>

Step one for producing quality filament is tuning pressure and temperature settings so that the plastic flows at the right rate, and viscosity.
Viscosity is a function of temperature, and the flow rate is dependent on pressure. The general rule of thumb is to find a "sweet spot", where both temperature and pressure are in the middle of their operating range. Some plastics require high temperatures and lower pressures, and others vice versa.

It is recommended that you use virgin pellets when experimenting and do not use the spooler. The following steps should help you find the proper temperature and pressure settings for your experiment:

1. Ensure that the plastic is completely clean and dry.
2. Disable minimum pressure limiting by sending the command “pl000000” (no quotes), and set the pressure to halfway up the slider.
3. Set the temperature by consulting the datasheet for the plastic in question (note: the extrusion temperatures for ProtoCycler+ can vary drastically from those used in 3D printing). Begin with the **lowest** recommended temperature.
4. If the plastic is not flowing once the set temperature is reached, increase the temperature in 5 degree increments until the plastic is flowing adequately through the nozzle.
5. Pay attention to the visible viscosity of the plastic - if it is dripping like liquid, it is not viscous enough. If it is rigid, it is too viscous. The plastic should curve and fall away from the nozzle into the garbage chute.

Now that we have a baseline for the temperature - we can move on to adjusting the pressure.

6. Most plastics extrude best in the 60-70% range on the pressure scale. Make a note of the motor speed that is required to maintain this pressure.
7. If the motor is spending most of its time at maximum speed, then 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 then the viscosity is too high, and the temperature must be increased. Ideally, the motor spends almost all of its time between 50 and 100 (100 and 200 on firmware V1.04 and earlier).
8. Now that the viscosity has been correctly set, it’s time to turn the auger lower pressure limit back on. Set PL to 25 using the command “pl000025”, no quotes. With firmware V1.04 and earlier, set PL to 50. Note that PL can, and sometimes should, be adjusted to suit the shear strength and elasticity of various plastics. Tuning PL is covered in advanced parameters, below.

<u>Flow Rate, Sensor Alignment, and Cooling</u>

Now that plastic is flowing, it’s time to check the flow rate, sensor alignment, and cooling. The goal of this section is to ensure that:

- The diameter sensors (particularly the nozzle sensor) are well centered on the plastic. This is important because elasticity varies depending on the plastic, thus the puller wheels need to know at what rate to pull the filament to ensure the best quality. Less elastic plastics will ride “higher” on the nozzle sensor, and vice versa.
- The flow rate is an acceptable value. Too low, and the filament will harden too early. Too high, and the filament won’t harden in time for the puller wheels.
- The cooling rate is determined correctly.

Please read thoroughly, and follow the instructions:

1. To start, set the puller speed to ~30% on the slider, and the fan to maximum (***never attempt to pull plastic without the fan engaged, even for plastics that aren't 3D printed with cooling fans!***).
2. Trim the filament at the nozzle and remove all of the waste from the garbage chute.
3. Start feeding the filament between the fan, puller guides, and puller wheels.
4. Monitor the final filament diameter. It doesn't have to be exactly 1.75mm, but it should be close.
5. If it is consistently equal to or lower than 1.5mm, lower the puller speed until it is between 1.5mm and 2.0mm. Conversely, if it is equal to or greater than 2.0mm, raise the puller speed until it is between 1.5mm and 2.0mm.
6. Once it is in the correct range, set the nozzle sensor height and alignment. If you are unsure how to do so, please refer to the user manual.

The flow rate can now be accurately measured. 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.

7. Now we can establish the cooling rate. This is largely a qualitative exercise. The goal is to have the filament cool to the point where it won't be deformed by the puller wheels, and can also be easily wrapped around the spool (Note: 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 you might sometimes not be able to reach a cooling rate that is sufficient, and will have to re-adjust the flow rate and temperature again. It might require a few iterations before the plastic behaves acceptably.

<u>Diameter Feedback</u>

Note: please do not proceed to the following steps without having quality filament being produced from following the previous sets of instructions.

1. Look at the "nozzle history" parameter on the graph to ensure that the nozzle sensor is reading correctly. 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.
2. Please note that the diameter feedback system won't work with clear filament. Dye or colorant of some form must be used.

3. If the diameter starts to oscillate out of control, the diameter PID system must be tuned to be less aggressive. If it is stable but not reacting quickly enough, the diameter PID system must be tuned to be more aggressive.
4. In almost all cases, D should be roughly 60-80% of P. The stock values are 0.5 and 0.3 for P and D respectively (Note: 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).

Edge Cases

There are more variables involved in extrusion that are beyond the scope of this guide. Most of them are regarding random events such as slip or shear, and other variables that aren't part of a normal steady state extrusion system. What follows below, is how to deal with the edge case of a slip/shear event.

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 at 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 15-35 is in the ideal range. If you find that the diameter is swinging too much during a shear event, try increasing PL by 5 (from the default of 25) and monitor the next shear event. If you find that the auger stalls during shear events, try decreasing PL by 5-10. *For firmware V1.04 and earlier, double all PL figures - i.e. a range of 30-70, increasing by 10 from a default of 50, and decreasing 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).