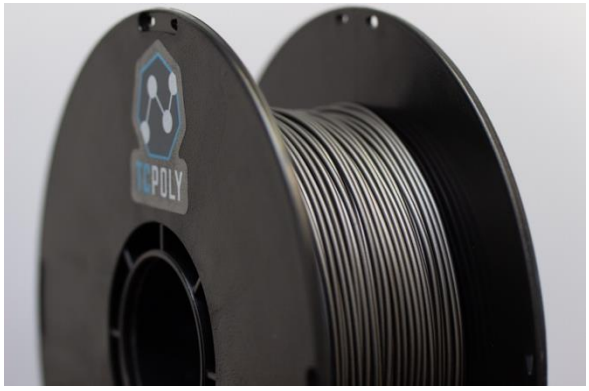
GUIDE TO PRINTING TCPOLY FILAMENTS



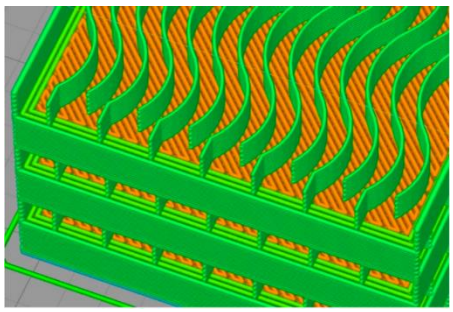
TCPoly has developed the first heat conducting plastics for open source FDM 3D printers. The 3D printing filaments have **thermal conductivity up to 50X higher than traditional plastics** and when combined with the design freedom of 3D printing, can be used to make high performance heat transfer products. This guide is designed to help you get the most out of the filament by enabling easier printing and better end parts.

TCPoly Ice9 Thermally Conductive Filaments

Ice9 Base Material	TPU	Nylon
Thermal Conductivity, In-Plane (W/m-K)	6	4
Thermal Conductivity, Through-Plane (W/m-K)	2	1
Max Use Temperature (°C)	110	175
Hardness (Shore)	Med 90A	High 70D
Volume Resistivity (ρ_v, Ω·cm)	2.97E+6	
Voltage Breakdown (V)	2500	
DI water / PEG compatible	Yes	Yes

Optimizing your parts for thermal performance and speed

TCPoly materials have the highest thermal conductivity of any FDM material, but without proper print setup and slicing, the part performance can still fall short of targets. Using a combination of our design and printing expertise along with proprietary algorithms, we create toolpath commands for the highest thermal performance, print quality, and shortest print times. Contact us for more information on our slicing and print optimization services.



GUIDE TO PRINTING TCPOLY FILAMENTS

PRINT IT BIG, SLOW, AND DRY!

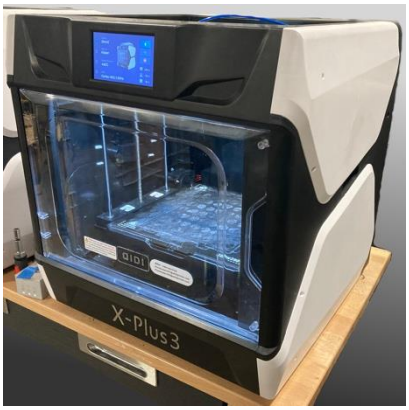
0.4 and 0.8 mm nozzles

- **PRINT BIG:** Although you can print down to 0.4mm, try to use a 0.8 or 1.0 mm nozzle if possible. Also use a large layer height (>50% of the nozzle diameter).
- **PRINT SLOW:** Reduce the linear print speed to get good layer adhesion and prevent nozzle buildup. Note: lower linear speed does not have to be a lower print rate: printing at 1.0/0.7 mm (nozzle/layer) at 20 mm/s is the same volumetric rate as 0.4/0.25 mm at 150 mm/s!
- **PRINT DRY:** Our filaments will absorb some moisture over time, so for the best quality prints store the material in a dry sealed container and dry at 70°C for several hours to dry out your spool. For filament drying we use the PrintDry but there are also lower cost options available, including food dehydrators on Amazon.
- **PRINT WITH RETRACTION:** Use a lot of retraction. Because Ice9 melts quickly, you need to retract it far to prevent oozing. Try 5 mm on 1.75 mm diameter, and 8 mm on 2.85 mm diameter and considering using a coast distance of 0.3-0.5 mm if available in your settings. Minimizing travel distance where possible will help too.
- **REDUCE ACCELERATION:** Fast starting and stopping will increase nozzle buildup. If you can reduce the acceleration limits on your printer, this will allow for higher linear print speeds at the same quality.
- **PRINT ON PAINTER'S TAPE:** The Ice9 TPU filament will adhere strongly to glass and tack print beds. To facilitate easy removal, considering printing on painters tape. Heating the bed up to 90-110°C will also help remove the part from a bed (with or without tape). Extra wide printers tape can cover a large portion of your print bed.

Our filament is compatible with a wide range of printers from: Ultimaker, Creality, Prusa, Flash Forge, Raise 3D, Bambu, Qidi 3 series, and many more.

For best results on Ice9 TPU, we recommend direct drive printers designed for flex materials.

Dual gear direct drive extruders work very well with our entire filament catalog. We use Qidi-Plus 3 and Qidi-Max 3 in house.



TROUBLESHOOTING & FAQ

PROBLEM	POSSIBLE CAUSE	POSSIBLE SOLUTIONS
Inconsistent extruding	Too much resistance to flow	This can be solved by some combination of increasing the nozzle diameter and reducing the flowrate. If you are more concerned with fine detail and need a 0.4 or 0.5 mm nozzle, then consider reducing the speed to 15-20 mm/s. Otherwise move to a larger nozzle size, the material flows best at 0.6-1.0 mm nozzle diameter.
	Excess moisture in filament	When the filament is left out in ambient (~20°C 50% RH) it will slowly absorb moisture and eventually become saturated. When filament saturated with moisture is melted in the hot end, the water evaporates causing the extruded filament to appear porous or bubbly. If the filament has been left out for more than a few days we recommend drying for 2-4 hours at 70°C for the highest quality prints.
	Nozzle is partially clogged	Occasionally the nozzle may become partially blocked during printing. To remove the blockage we recommend running pure plastic filament through the nozzle until it runs clear with the color of the new filament. ABS or PETG work well at 240°C, but you can also use PLA as well. If the nozzle doesn't come unclogged with this, you can use a nozzle cleaning bit.
Unable to remove part after print	Cold adhesion too strong	The Ice9 TPU filament typically has very good bed adhesion, especially to glass, so we recommend printing on painters tape when possible to facilitate easy part removal. Even so, removal is best accomplished when the part is warm. Our recommended procedure is to let the part cool down to room temperature after the print, then heat the bed back up to 90°C and gently peel the painters tape off the bed and then off the bottom of the part. If not using painter's tape use a spatula to remove the part from the bed. If printing directly on glass add 10-20°C to the bed removal temperature.

TROUBLESHOOTING & FAQ

PROBLEM	POSSIBLE CAUSE	POSSIBLE SOLUTIONS
Poor Layer adhesion	Temperature too low	Strong layer adhesion needs high enough temperature for the new layer to melt into the underlying layer. To improve layer adhesion increase the nozzle temperature by 5-10°C and also increasing the bed by 10°C.
	Nozzle too high	If the nozzle is too high in relation to the print bed, the filament will not be pushed into the previous layer creating a strong melt bond. To get the correct height relevel the print bed.
Buildup of filament on nozzle	Over extrusion	If too much material is extruded the layer height will rise and material will catch on the nozzle. Fix this by reducing the extrusion multiplier (in software) or flow (on the printer). You can also reduce the infill percentage slightly (e.g. reduce from 100 to 95%) and leave the extrusion multiplier and flow settings unchanged.
	Acceleration too high	Although TCPoly filaments can print without acceleration limits in place, reducing the max acceleration can sometimes reduce nozzle buildup since this is often caused by rapid starting and stopping. Many printers have hardware acceleration limits that can be set, we recommend limiting X and Y acceleration to 200 mm/s² if you are having issues with nozzle buildup.
Excessive oozing	Incorrect retraction settings	Thermally conductive filaments will tend to ooze more than normal filaments because the hot end melting zone will be larger. To combat this keep the retraction distance high (at least 4 mm on 1.75 mm filament and 7 mm on 2.85 mm filament) and consider using a fast retraction speed (~40 mm/s). Small coast and restart distances of 0.2-0.5 mm a recommended.
	Extruder temperature too high	If adjusting the retraction settings do not improve the oozing, consider lowering the hot end temperature 10°C.

TROUBLESHOOTING & FAQ

PROBLEM	POSSIBLE CAUSE	POSSIBLE SOLUTIONS
Filament kinking in extruder	Excessive resistance In hot end/nozzle	In extruders with a gap between the gear and the tube guiding the filament to the hot end, it is possible for the filament to kink and stop pushing downwards through the nozzle. This is typically caused by too great a resistance in the nozzle and can be solved by some combination of increasing nozzle diameter, increasing temperature, and reducing print speed.
	Using a Bowden extruder with 1.75 mm Ice9 TPU	Our flexible filaments may not work in some Bowden style extruders due to the extra stiffness required in these systems If the above steps to reduce resistance do not work it may be better to move to a direct extrusion printer.
Filament grinding in extruder	Extruder tension set too tight	If the filament is being chewed up by the extruder geared tooth, this usually means that the tension in the extruder is too high (i.e. there is too much pressure holding the filament against the gear). Most extruders have an adjustment so the tension can be reduced, but make sure it is not reduced so much where the filament is slipping. If this doesn't solve the problem see the explanation for excessive resistance under "Filament kinking."
Part is not air or watertight		Printing a part that seals against fluid is quite complex and requires careful setup and often post sealing. Please contact us if you are attempting to make an air or water tight part.
Part thermal performance is not good	Print oriented incorrectly	The thermal conductivity of our filament is different depending upon the print direction. Often times, reorienting the print will allow for higher thermal conductivity at a heat source and improving the performance of the part. As a rule of thumb, you would like print lines extending in the direction that you need to move heat (not perpendicular). For more in-depth support on this subject, please contact us.

TROUBLESHOOTING & FAQ

PROBLEM	POSSIBLE CAUSE	POSSIBLE SOLUTIONS
Part is improperly formed with too much melting	Not enough cooling between layers	For smaller parts or parts with only a few features at certain heights, it may take more time for a layer to cool than is proscribed in the standard print settings. You can use standard print settings to reduce speed and increase cooling in these areas. Typical settings involve decreasing a print speed or increasing a fan speed if a layer print time is below a certain value (e.g. 15 sec.). If these settings doing work or aren't available try enabling the cooling fan for all layers after the first (for a small part) or just for the fine features.
Poor bed adhesion	Nozzle too high/first layer too fast	The Ice9 TPU filament typically has very good bed adhesion, so the culprits for poor bed adhesion will be common to general FDM printing. Ensure that the bed is level and the first layer is printed slower with the nozzle closer to the bed if you are having issues. We typically recommend first layer speed ~60% of the print speed in subsequent layers and having the nozzle 25% closer on the first layer if possible.
Nozzle clogged		Occasionally the nozzle may become partially blocked during printing. To remove the blockage we recommend running pure plastic filament through the nozzle until it runs clear with the color of the new filament. ABS or PETG work well at 240°C, but you can also use PLA if needed. If the nozzle doesn't come unclogged with this, you can use a nozzle cleaning bit.