Creator 3 materials

PLA

METAL FILL PLA

WOOD FILL PLA

ABS

PC

PA(NYLON)

HIPS

PVA

PETG

FLASHFORGE
3D PRINTER

Support Structure

Support structure is essential for creating geomentries with overhangs in FDM. The melted thermoplastic cannot be deposited on thin air. For this reason, some geometries require support structure.

Surfaces printed on support will generally be of lower surface quality than the rest of the part. For this reason, it is recommended that the part is designed in such a way to minimize the need for support.

Support is usually printed in the same material as the part. Support materials that dissolve in liquid also exist, but they are used mainly in high-end desktop or industrial FDM 3D printers. Printing on dissolvable supports improves significantly the surface quality of the part, but increases the overall cost of a print, as specialist machine (with dual extrusion) are required and because the cost of the dissolvable material is relatively high.

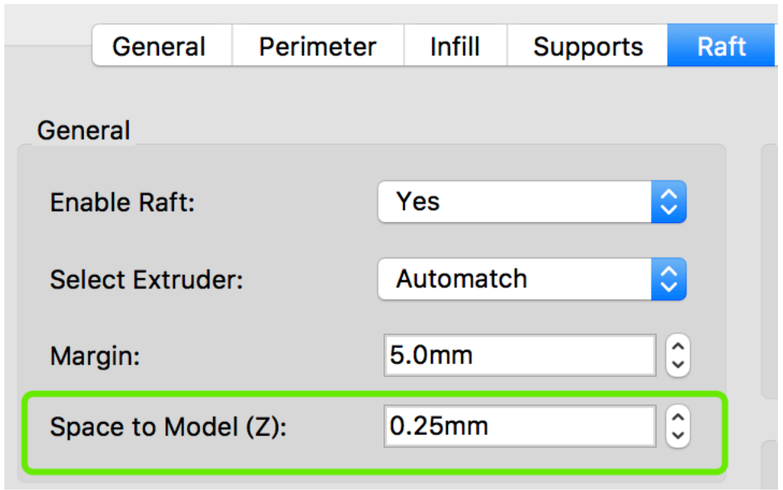
Material	Characteristics
ABS	+ Good strength +Good temperature resistance —More susceptible to warping
PLA	+Excellent visual quality +Easy to print with —Low impact strength
Nylon (PA)	+High strength +Excellent wear and chemical resistance -Low humidity resistance
PETG	+Food Safe* +Good strength +Easy to print with

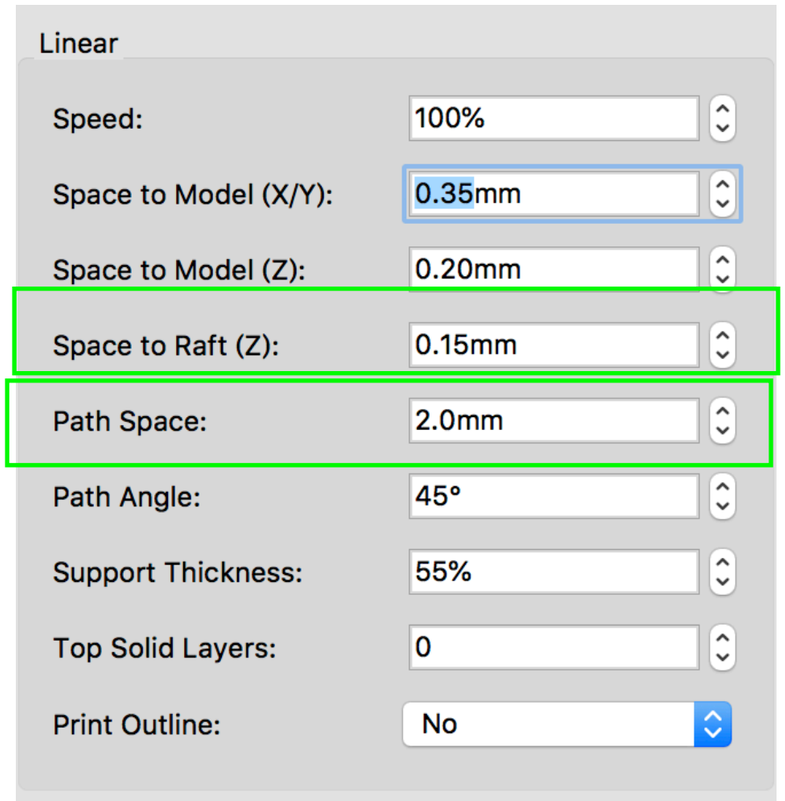
PLA

Color

luminous

transparent





Creator 3 Parameter setting recommendations

Add the raft or set the build plate temperature to 50-70 degree will improving printing success rate, it will make sure filament stick on the build plate. The raft is not a necessity

It's easier to peel off models if setting the distance between raft and model as 0.2-0.25mm when printing with PLA, especially printing larger models.

Adjust the distance with model Z-axis is beneficial for supports removal when choosing PLA as supporting filament. But we do suggest setting space to raft(z) as 0.15-0.2mm for it's not beneficial for supports when distance is too large.
The larger the path space, the less support is needed, the less print time is needed, which depends on how big the support surface is, the smaller the support surface, the thinner the path.

Filament Temperature differences

The printing temperature required for transparent filament can be set lower, usually at 190-205 degrees;
Recommended printing temperature for white and black filament is 210 ° c
Recommended printing temperature of luminous filament is 210-220 degrees

Performance

High stiffness, good detail, affordable.Safety and environmental protection: filament can be used in contact with food.

Food contact grade original materials with complete biodegradability, no odor printing process.

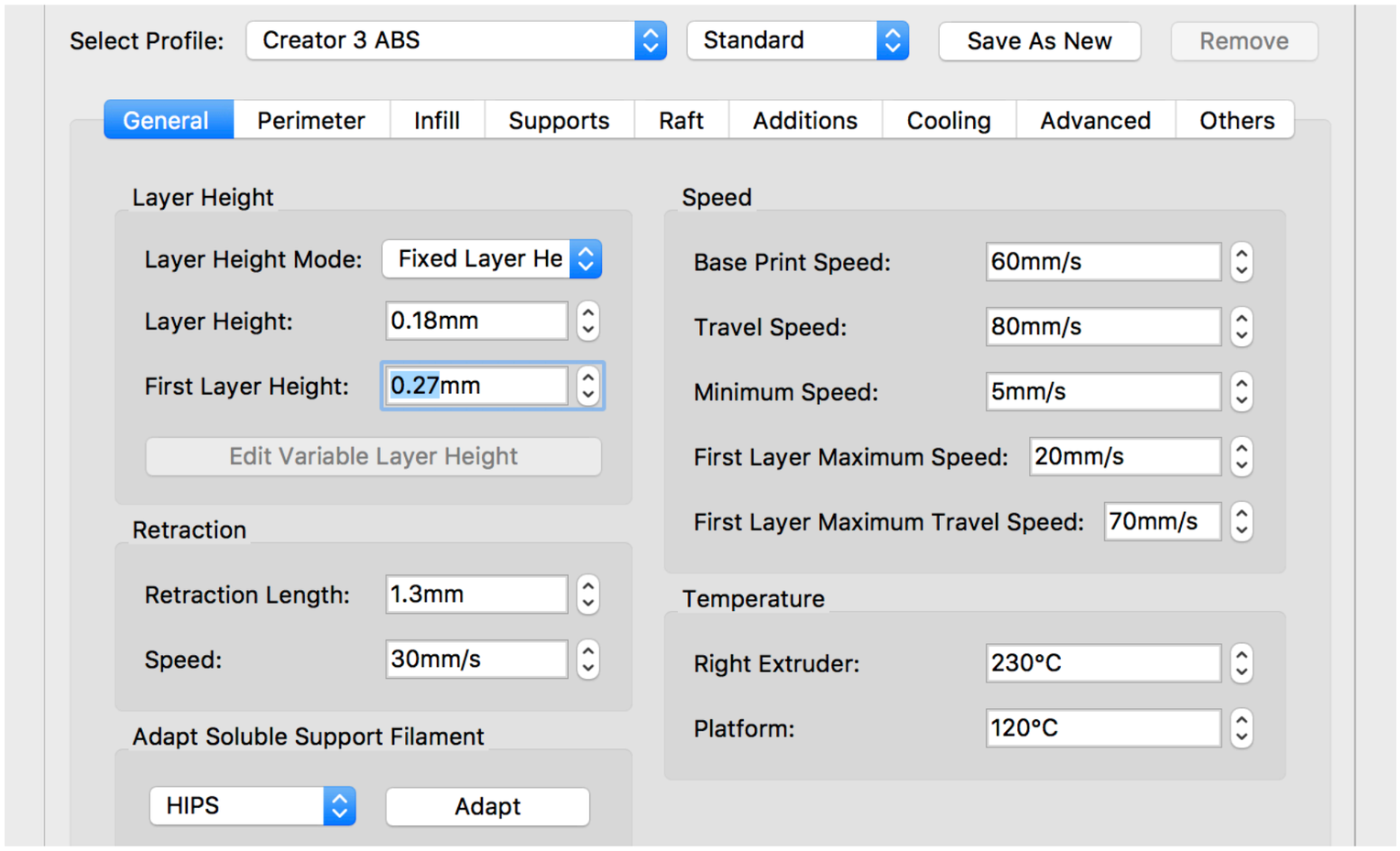
Nozzle temperature	190-220°C
Plate temperature	40-60°C
Print speed	60-90mm/s
Raft space	0.25mm
Envirement temperature	20-30
Support material	Pva

Model printing recommendations

Print of general models such as appearance design models, toy design models and architectural design models with good surface effect and low requirement for strength (PLA is easy to weather after a long time)

ABS

Color



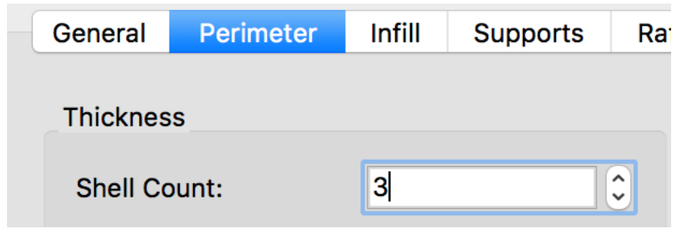
Performance

Commodity plastic, improved mechanical and thermal properties compared to PLA. ABS has good mechanical properties, with excellent impact strength, superior to PLA, but less defined details. Commonly used for enclosure prototypes.

Nozzle temperature	220-230°C
Plate temperature	120°C
Print speed	60-90mm/s
Raft space	0.2mm
Envirement temperature	20-45
Support material	HIPS

Creator 3 Parameter setting recommendations

Affected by temperature easily. The front door and upper cover should be closed when printing.
It is suggested to add a raft to printing, which has better adhesion and higher printing success rate.
If you want to print a tiny and complex structure or higher strength , we suggest to set shell count to 3or 4 to increase the strength. Or increase the fill density.

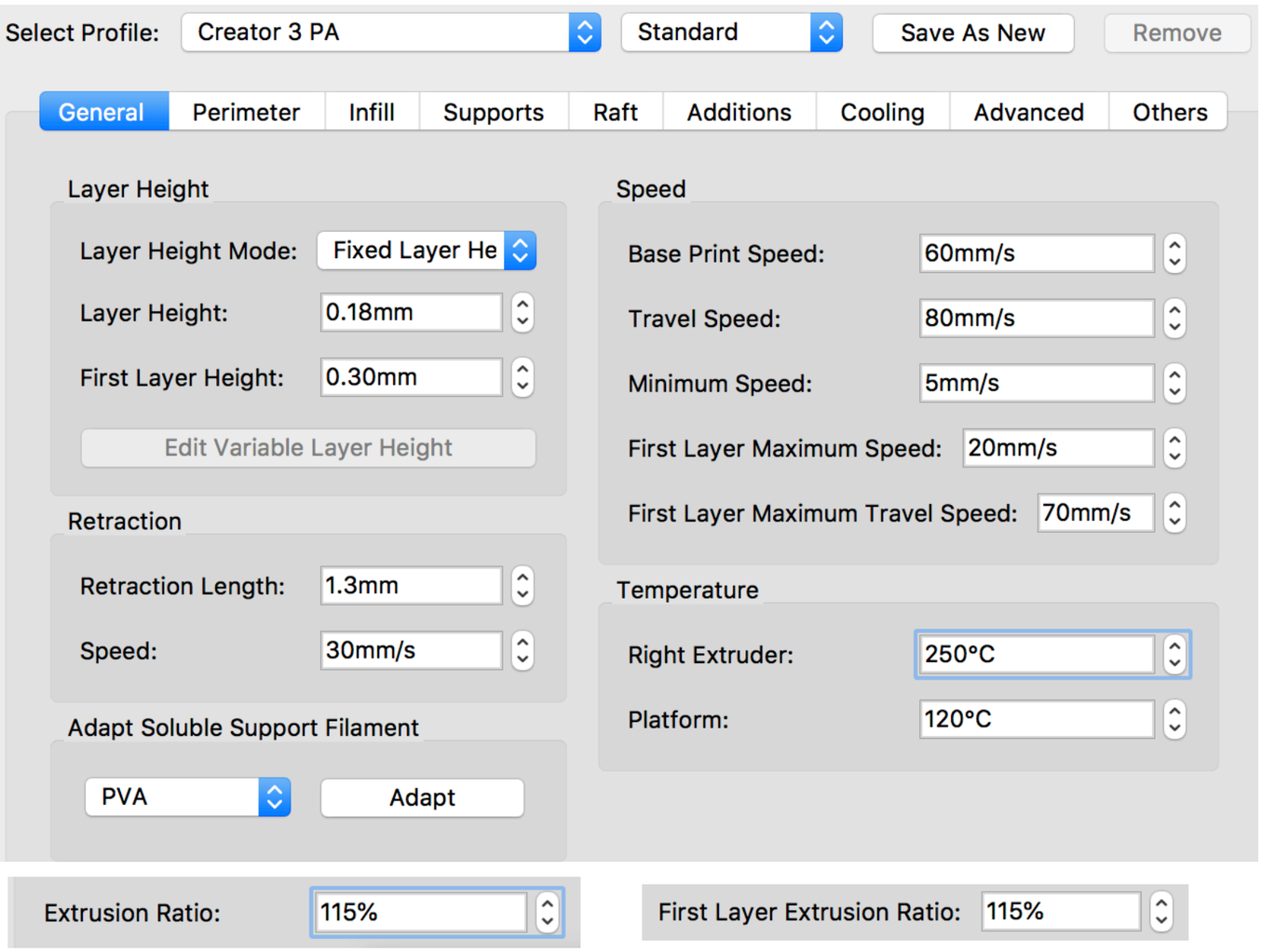


Model printing recommendations

ABS is of High strength, applied to print tooling models, auto parts, etc.

PA

Color



Performance

Used to substitute functional injection moulded parts, good chemical resistance.

PA has superior mechanical properties than ABS and high chemical and abrasion resistance. Used for functional parts requiring high fatigue strength.

Nozzle temperature	240-260°C
Plate temperature	90-120°C
Print speed	30-60mm/s
Raft space	0.2mm
Envirement temperature	20-45
Support material	Pva

Creator 3 Parameter setting recommendations

When printing with PA, the equipment should be full enclosed. When used PVA as the support material, the thickness of the first layer can be set to 0.3-o.4mm, which is conducive to the adhesion with filament .(or keep the layer height, but set the first layer extrusion ratio to 110-120% to get better a adhesion)
If you use third-party material we suggest to set extruder temperature to 250-260degree. And set extrusion ratio to 110-120%.

Storage recommendations

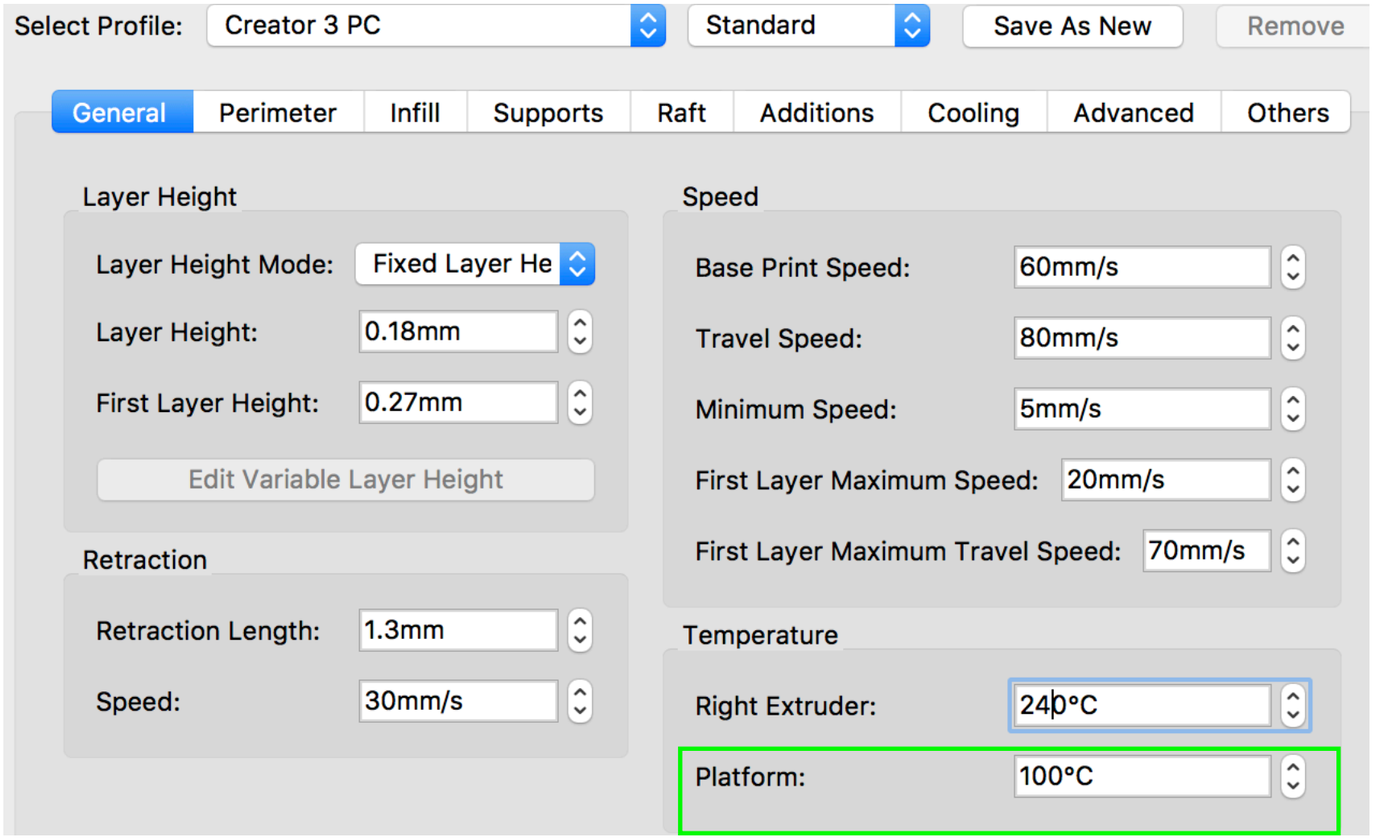
PA material is subject to moisture. Pay attention to storage and use it as soon as possible after unpacking. If the material has been damped. It will accompanied by small bubbles with squeaky sounds during printing. It can be dried in a 50 degree oven.

Model printing recommendations

PA has strong toughness to print small buckle models. It is suggested to print with PA, thus the buckle structure is not easy to be broken.

PC

Color



Performance

PC has better mechanical properties than the commonly used PLA and ABS.
The deformation and temperature resistance of PC is higher than ABS

Nozzle temperature	230-260°C
Plate temperature	100-120°C
Print speed	50-80mm/s
Raft space	0.2mm
Envirement temperature	20-45
Support material	/

Creator 3 Parameter setting recommendations

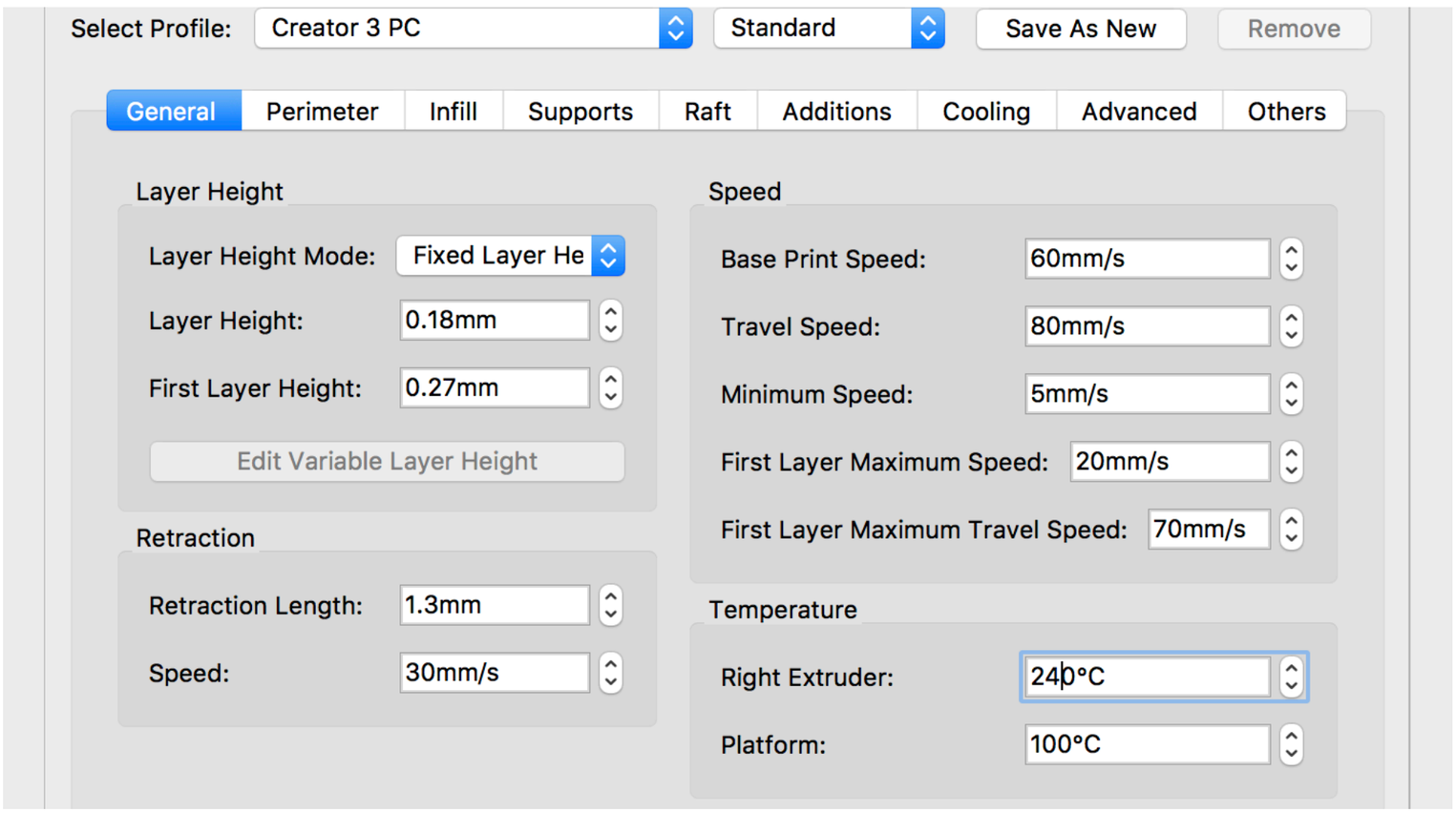
The temperature of the bottom plate required by PC material is high. It is suggested to set the temperature above 100 °C for printing and keep the device full enclosed.

Model printing recommendations

The deformation and temperature resistance of PC is higher than ABS.
It is suitable to print some models such as lamp decorations and lampshades, which is not easy to deform under higher temperature

PETG

Color



Performance

Good for mechanical parts with high impact resistance and flexibility. Sterilizable.

PETG is a thermoplastic material with improved mechanical properties over PLA and excellent chemical and moisture resistance.

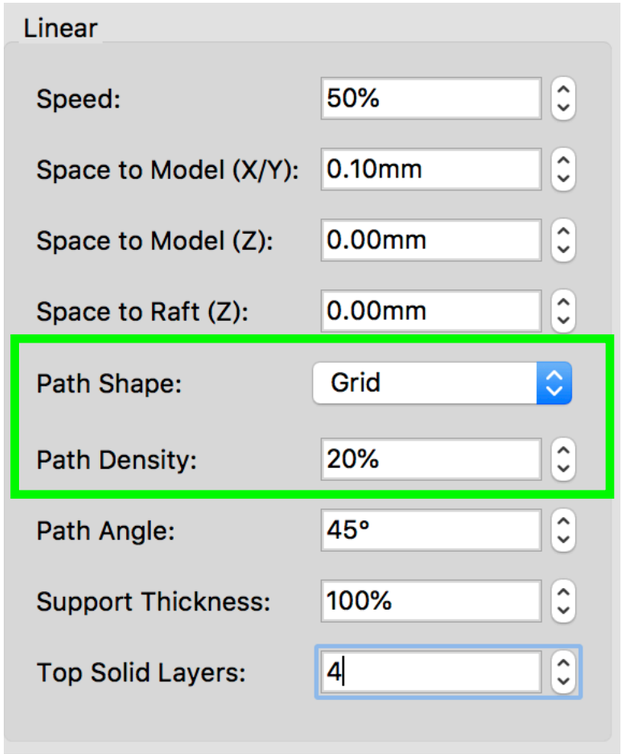
Nozzle temperature	220-240°C
Plate temperature	80-100°C
Print speed	60-80mm/s
Raft space	0.2mm
Envirement temperature	20-45
Support material	/

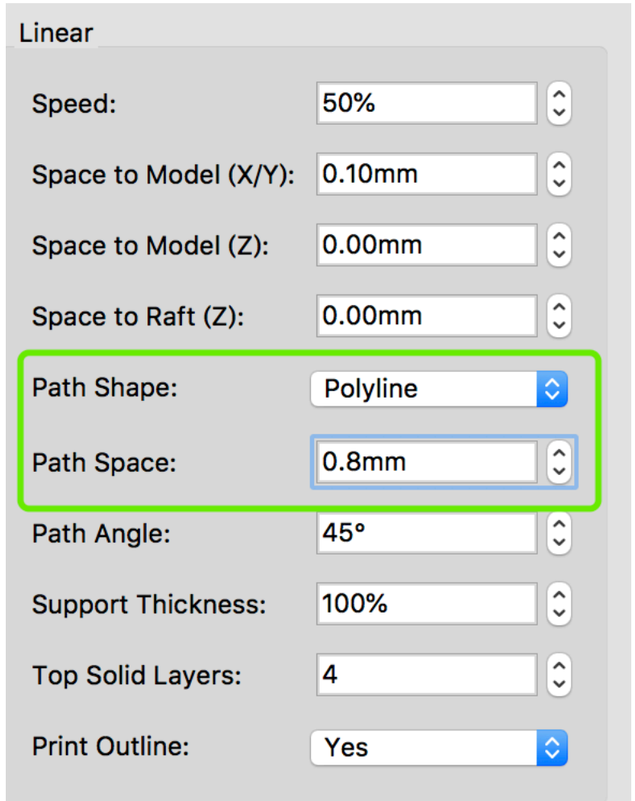
Model printing recommendations

applied to print some storage tanks, medicine storage tanks, etc.It can also print some objects that need higher impact and toughness, and its overall anti-corrosion and anti-weathering performance is better than PLA, with better printing performance.

PVA

Color





Model printing recommendations

As supporting filament, when printing large models, such as the volume of 100mm or larger supporting surface, grid mode is recommended to save the use of supporting filament.

However, the poly line mode is recommended when printing models with complex, fine, small structures.

The smaller the path spacing, the denser the support, the better the printing effect, but the more filament used, the longer the time required.

Dissolution time

If the model is big, we can remove some support first, than soak the model in water directly, the dissolution time maybe 4-6 hours(it is depends on the volume of support(small model may be 1-2 hours), flowing running water will accelerate the dissolution.

Tips: After soaking for 6 hours, PVA has been softened. It is recommended to use small tools such as brush to remove it, which can save time.

Performance

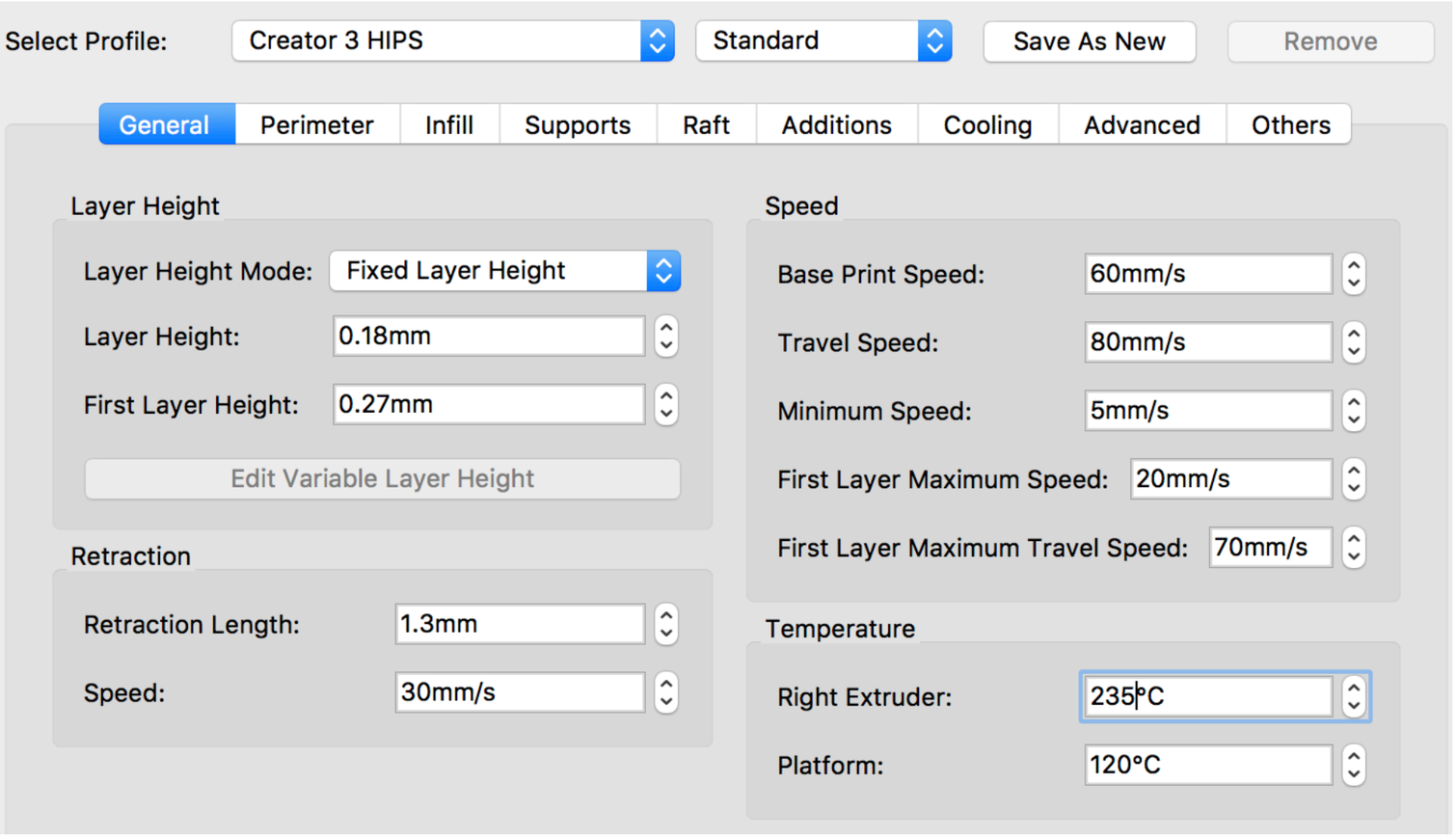
PVA is soluble in water, apply to used as supporting filament.

Nozzle temperature	200-220°C
Plate temperature	60-70°C
Print speed	60-80mm/s
Raft space	0.2mm
Envirement temperature	20-45
Support material	PLA/TPU

Storage recommendations

Direct exposure to the air, it is easy subject to moisture and softening, print bad easily after softening ,so it is best to store and print in dry boxes.

HIPS Color



Performance

HIPS is not soluble in water, dissolved in limonene, without contact toxicity or irritation.
Apply to used as supporting filament.

Nozzle temperature	230-240°C
Plate temperature	120°C
Print speed	30-80mm/s
Raft space	0mm
Envirement temperature	20-45
Support material	ABS

Model printing recommendations

Affected by temperature easily. The front door and upper cover should be closed when printing.
As supporting filament, when printing large models, such as the volume of 100mm or larger supporting surface, grid mode is recommended to save the use of supporting filament.

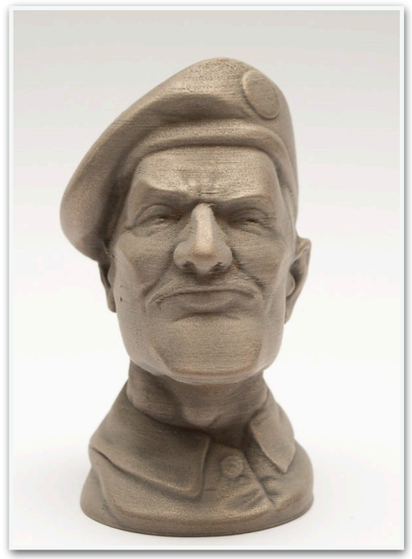
he poly line mode is recommended when printing models with complex, fine, small structures.
The smaller the path spacing, the denser the support, the better the printing effect, but the more filament used, the longer the time required.

Woodfill PLA

Woodfill PLA (FDM) contains wood, bamboo, or cork-based pulverized material, resulting in FDM parts with a unique wood-like appearance.

Metalfill PLA

Metalfill PLA (FDM) contains steel, copper, bronze or other metal particles that give parts a metallic surface finish and unique properties.

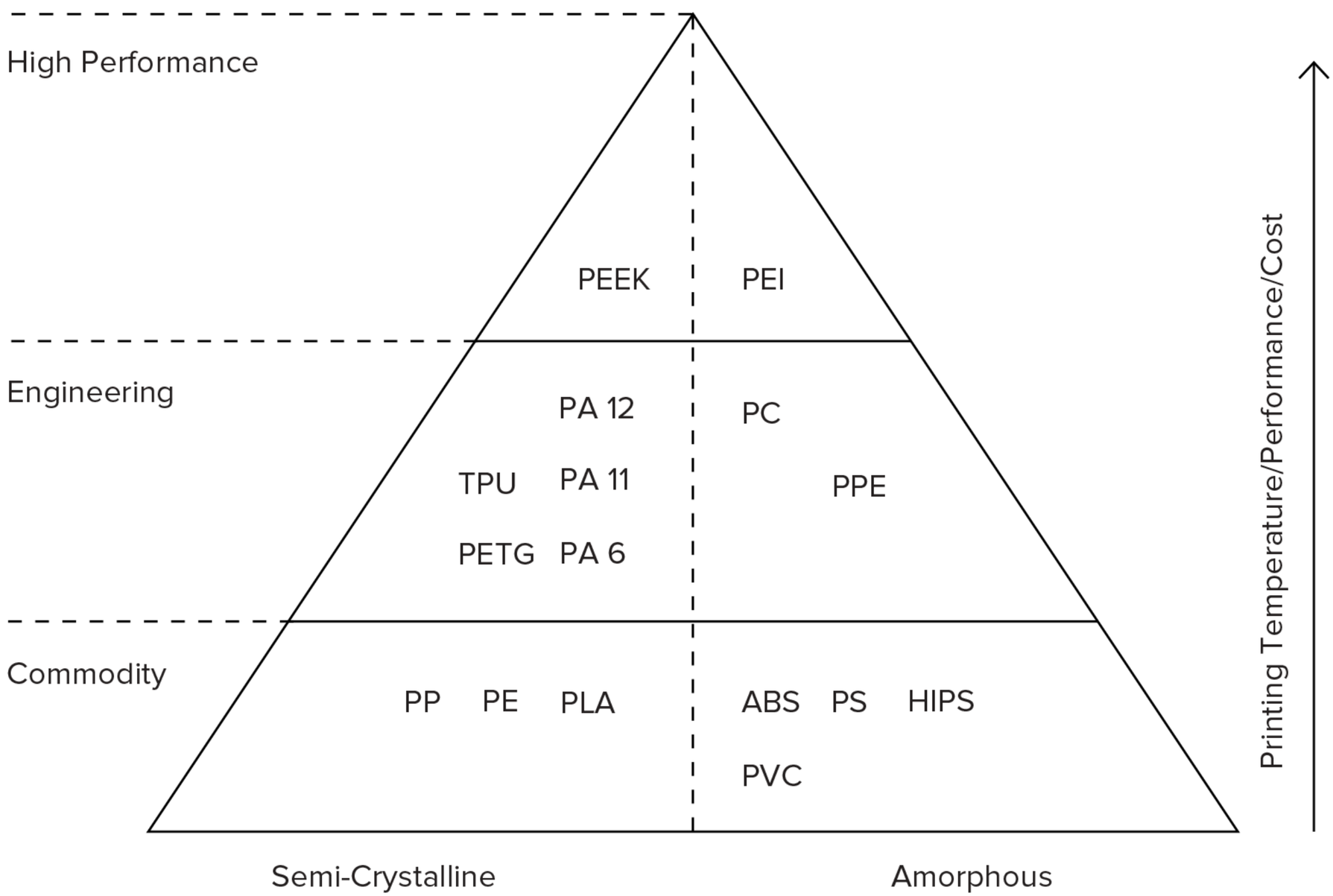


Marble-PLA

Marble PLA give parts a unique marble-like appearance.

Common FDM Materials

One of the key strengths of FDM is the wide range of available materials. These can range from commodity thermoplastics (such as PLA and ABS) to engineering materials (such as PA, TPU, and PETG) and high-performance thermoplastics (such as PEEK and PEI).



FLASHFORGE 3D PRINTER