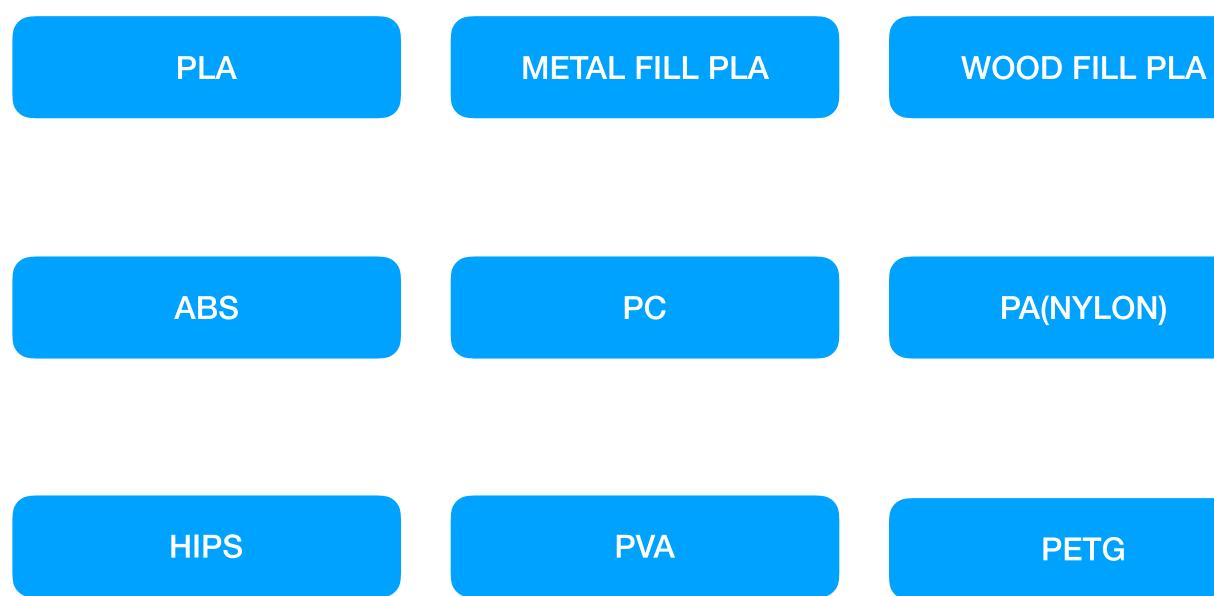
Creator 3 materials





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

nce

PLA c	olor	
General Perimete		Creator 3 Parameter setting recommendations
General Enable Raft: Select Extruder:	Yes 🗘 Automatch	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
Margin: Space to Model (Z):	5.0mm 🗘	It's easier to peel off models if setting the distance

Linear	
Speed:	100%
Space to Model (X/Y):	0.35mm
Space to Model (Z):	0.20mm 🗘
Space to Raft (Z):	0.15mm
Path Space:	2.0mm 🗘
Path Space: Path Angle:	2.0mm 🗘
Path Angle:	45°

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

transparent

erformance

gh stiffness, good detail, affordable.Safety and vironmental protection: filament can be used in ontact with food.

ocess.

Nozzle te

Plate te

Print

Raft

Envi temp

Suppor

ood contact grade original materials with mplete biodegradability, no odor printing

emperature	190-220°C
emperature	40-60°C
t speed	60-90mm/s
t space	0.25mm
irement perature	20-30
rt 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)

Select Profile:	Creator 3	ABS		\$	Standard	٤ (Save As New	Remove
General	Perimeter	Infill	Supports	Ra	ft Additions	Coolin	g Advanced	Others
Layer Hei	ght			ę	Speed			
Layer He	eight Mode:	Fixed Lay	er He ᅌ		Base Print Spee	d:	60mm/s	•
Layer He	eight:	0.18mm	•		Travel Speed:		80mm/s	\$
First Lay	er Height:	0.27mm	Ĵ		Minimum Speed	:	5mm/s	
E	dit Variable	Layer Heigl	nt		First Layer Maxi	mum Spe	ed: 20mm/s	
Retraction	۱				First Layer Maxi	mum Trav	vel Speed: 70mm	n/s 🗘
Retractio	on Length:	1.3mm	•		Temperature			
Speed:		30mm/s	\$		Right Extruder:		230°C	
Adapt Sol	uble Suppor	t Filament			Platform:		120°C	
HIPS	\$	Ada	pt					

Fill Density:

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.

	-
Plate	temp
Pr	int sc

Envirement te

Model printing recommendations

ABS is of H models, auto p

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.

General	Perimeter	Infill	Supports	Ra
Thicknes	S			
Shell Co	ount:	3		Ĵ

25%

\$



Color

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

High	strength,	applied	to	print	tooling
parts	, etc.				

PA	Ca	olor •						
Select Profile:	Creator 3		Standa		Save As New	Remove		Performance
General Layer Hei	Perimeter	Infill Supports	Raft Ad	dditions Cool	ing Advanced	d Others		Used to substitut moulded parts, g
Layer He Layer He First Lay	eight Mode: eight: rer Height:	Fixed Layer He 🗘 0.18mm 🗘 0.30mm 🗘 Layer Height	Base Pi Travel S Minimu	rint Speed: Speed: m Speed: yer Maximum Sp	60mm/s 80mm/s 5mm/s			PA has superior r ABS and high ch resistance. Used requiring high fat
Retraction					avel Speed: 70n			Nozzle temper
Retractio	on Length:	1.3mm 🗘	Tempera	ature				Plate tempera
Speed:		30mm/s	Right E	xtruder:	250°C	Ĵ		Print spee
Adapt Sol	luble Suppor	t Filament	Platform	n:	120°C			Raft space
PVA	\$	Adapt						Envirement temp
Extrusion Ra	atio:	115%	First	Layer Extrusion	Ratio: 115%			Support mate
		arameter settir nendations	ng	S	torage reco	ommenda	tions	Model p
enclosed. Whe thickness of th which is condu- keep the layer ratio to 110-12 If you use third	en used PV ne first layer ucive to the height, but 20% to get h d-party mat	e equipment should A as the support ma can be set to 0.3-o. adhesion with filam set the first layer ex better a adhesion) erial we suggest to s 250-260degree. And	terial, the .4mm, ent .(or drusion	storage ar unpacking accompar	al is subject to n nd use it as soon . If the material nied by small bu nting. It can be c	n as possible a I has been dar bbles with squ	after nped. It will eaky sounds	PA has strong to It is suggested buckle structur

extrusion ratio to 110-120%.

Ite functional injection good chemical resistance.

mechanical properties than nemical and abrasion I for functional parts tigue strength.

rature	240-260°C
ature	90-120°C
ed	30-60mm/s
e	0.2mm
perature	20-45
erial	Pva

rinting recommendations

toughness to print small buckle models. to print with PA, thus the re is not easy to be broken.

PC Color				
Select Profile: Creator 3 PC	;	Standard	Save As New	Remove
Layer Height	Infill Supports	RaftAdditionsCSpeedBase Print Speed:	Cooling Advanced	Others
	.18mm	Travel Speed:	80mm/s	
	.27mm	Minimum Speed:	5mm/s	
Edit Variable Lay	yer Height	First Layer Maximum First Layer Maximum		
Retraction Retraction Length: 1.	.3mm 🗘			1/3 V
	0mm/s	Temperature Right Extruder:	24þ°C	Ĵ
		Platform:	100°C)

Creator 3 Parameter setting recommendations

Model printing 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.

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 echanical properties than the PLA and ABS. and temperature resistance an ABS

re	230-260°C	
e	100-120°C	
	50-80mm/s	
	0.2mm	
ature	20-45	
I	/	

PETG Color	• • • • • • •	• • • •	
Select Profile: Creator 3 PC	Standard	Save As New Remove	Performance
General Perimeter Infill	Supports Raft Additions C	Cooling Advanced Others	Good for mechan resistance and fle
Layer Height Layer Height Mode: Fixed Layer Layer Height: 0.18mm	Speed yer He Base Print Speed: Travel Speed:	60mm/s 80mm/s	PETG is a therm improved mechar excellent chemica
First Layer Height: 0.27mm Edit Variable Layer Heig	Image: Minimum Speed:Image: Minimum Speed:Im	5mm/s 🗘	Nozzle temp
Retraction	First Layer Maximum	n Travel Speed: 70mm/s	Plate tempe
Retraction Length: 1.3mm	C Temperature		Print spe Raft spa
Speed: 30mm/s	Right Extruder:	24þ°C	Envirement ten
	Platform:	100°C	Support ma

Model printing recommendations

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

nical parts with high impact exibility. Sterilizable.

noplastic material with nical properties over PLA and al and moisture resistance.

erature	220-240°C	
erature	80-100°C	
ed	60-80mm/s	
ice	0.2mm	
nperature	20-45	
aterial	/	

PVA Color

Linear	
Speed:	50%
Space to Model (X/Y):	0.10mm 🗘
Space to Model (Z):	0.00mm 🗘
Space to Raft (Z):	0.00mm 🗘
Path Shape:	Grid
Path Shape: Path Density:	Grid 🗘
·	
Path Density:	20%
Path Density: Path Angle:	20% ¢

Linear \$ 50% Speed: \$ 0.10mm Space to Model (X/Y): \$ Space to Model (Z): 0.00mm \$ 0.00mm Space to Raft (Z): Path Shape: \Diamond Polyline \$ 0.8mm Path Space: \$ 45° Path Angle: \$ 100% Support Thickness: \$ Top Solid Layers: 4 Print Outline: Yes

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 tem

Plate tem

Print s

Raft sp

Envirement t

Support material

boxes.

nperature	200-220°C	
perature	60-70°C	
speed	60-80mm/s	
pace	0.2mm	
temperature	20-45	
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

HIPS	Col	lor ●								
Select Profile:	Creator	r 3 HIPS		Sta	andard	Sav	e As New	Rem	ove	Performance
General Layer Height	Perime	eter Infill	Supports	Raft	Additions Speed	Cooling	Advanced	Others		HIPS is not limonene, witho Apply to used a
Layer Height Layer Height First Layer H	t:	Fixed Layer H 0.18mm 0.27mm	leight	 ○ ○ 	Base Print S Travel Spee Minimum S	ed:	60mm/s 80mm/s 5mm/s		0	
	Edit Varia	able Layer Hei	ght		First Layer	Maximum Sp	eed: 20mn	n/s	0	Nozzle te
Retraction					First Layer	Maximum Tra	avel Speed:	70mm/s	0	Plate ten
Retraction L	ength:	1.3mm		\$	Temperature	2				Print
Speed:		30mm/s		0	Right Extru		235°C		0	Raft
opoodi				•	Platform:	Jei.	120°C		• •	Envirement
							120 0		•	Support

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.

soluble in water, dissolved in out contact toxicity or irritation. is supporting filament.

emperature	230-240°C
mperature	120°C
speed	30-80mm/s
space	0mm
temperature	20-45
t material	ABS

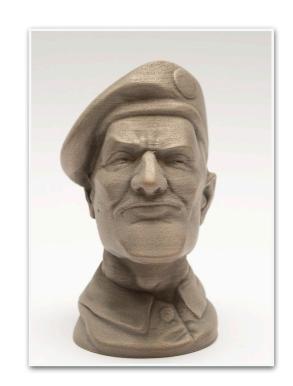
Woodfill PLA

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).

