Printing Guide of TPU Filament with Creator 4

In daily life, there are many users who need to print TPU flexible filament, while "delicate" TPU has high requirements for 3D printers and printing experience. Therefore, in order to achieve excellent printing results, we need to consider all aspects of the configuration.

1 Filament Introduction

▶ TPU (thermoplastic polyurethane) is a plastic filament that is commonly used in FDM 3D printing to manufacture flexible parts. Its glass transition temperature at about 80℃ is higher than that of PLA (60-65℃). It is also resistant to low temperatures, which means that it does not become brittle and difficult to work with. TPU flexible filament has a variety of properties such as vibration damping, shock absorption and shape restoration. Its uses include model car tires (or tank tracks), cushion pads, sports and fitness equipment, uppers and soles, etc.

▶ TPU flexible filament has different hardness, indicated by Shore A and Shore D. The lower number indicates it is softer; the higher number indicates it is harder; for example, TPU 90A is softer than TPU 95A.

# 2 Suggestion for Equipment Selection

| Extruder type

As TPU is a flexible filament, a 3D printer with a direct-drive extruder should be selected for use. The shorter the distance between the hot end and the nozzle, the less likely it is that the filament will get tangled or clogged. At the same time, the printing speed is also related to the design of the extruder of the equipment. Therefore, choosing the appropriate equipment (an extruder specially designed for printing flexible filaments - coordinated with printing 2.85mm filament) is more conducive to printing flexible filaments.

Creator 4's Extruder-F

Internal Structure Display of
Creator 4's Extruder-F
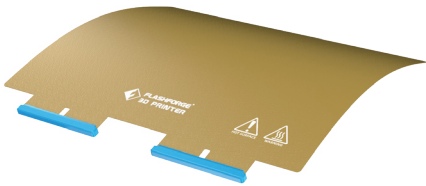
Build plate type

When printing TPU filament, choosing PEI platform or glass platform is more conducive to the removal of the printed model, so that it is not difficult to separate the model from the platform or the model will not be damaged when separated. It is not appropriate to print TPU with common rough platform film.

Installation position of spool

For printing TPU, the filament feeding resistance should be reduced as much as possible. However, the resistance will be increased when the filament is fed through the filament guide tube or the filament spool is at the lower position. Therefore, Creator 4 provides

a variety of ways to place filaments. When printing flexible filaments, the filament spool holder can be placed directly above the equipment and the filament can be delivered on the top, which realizes less resistance and is a more reasonable way of printing.
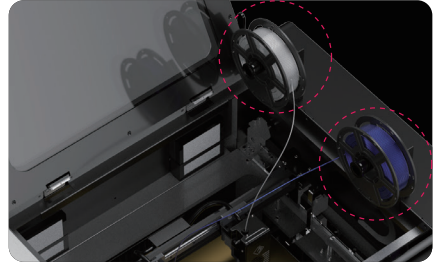
Use the correct 3D printing slice settings

various parameter settings such as speed/retraction/support will affect the quality of models printed with TPU filament.

3 Notes on Setting Printing Parameters

Printing Temperature

The general extruder temperature for printing TPU is: 190-230℃;
Creator 4 is suitable for printing TPU filament with the diameter of 2.85mm. It can ensure the rapid melting of filament and avoid impurity residues around the nozzle after printing for a long time.

| Printer	Machine Type	Creator 4		Basic Mode >
General	Nozzle Size	0.4 mm		Save Configuration
Shells	Material Right	Kexcelled-TPU(90A)	2.85mm	Restore Defaults
Infill	Material Left	Kexcelled-TPU(90A)	2.85mm	Import
Supports	Slice Profile	Standard		Export
Raft	Right Extruder Temperature	205℃		Remove
Additions	Left Extruder Temperature	205℃		Save As New

For the appropriate temperature range, please refer to the temperature that presents bright model surface effect.

| Retraction and Printing Speed

As TPU filament has high melt index and is soft, it is easy to cause stringing during printing. In order to ensure the normal printing of materials, it is required to set a relatively high retraction length and a relatively low printing speed.

The retraction length is generally set between 1.5mm and 5mm. If there are missing lines on the surface of the printed model, the retraction length and printing speed should be decreased; if there is stringing, the retraction length and printing speed should be increased.
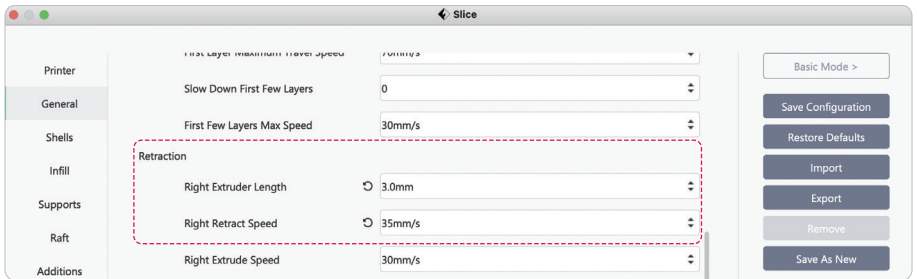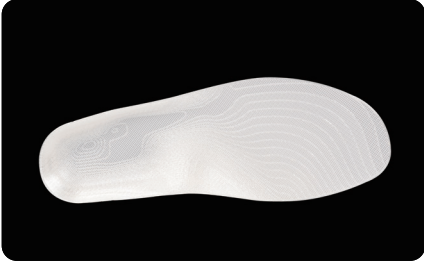

The printing speed can be set between 30mm/s and 70mm/s. However, for some simple models (e.g. insoles), the printing speed can also be set to 100mm/s. It is recommended to reduce the printing speed for complex structures with overhangs and bending shapes.
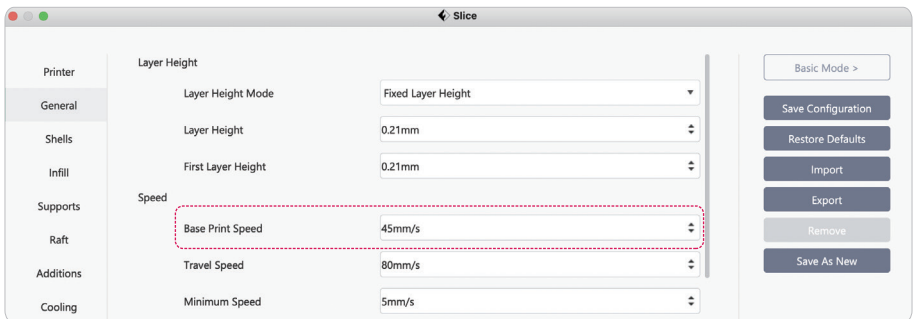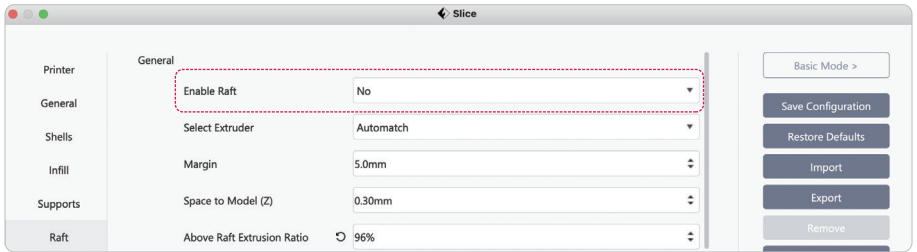
The following is the insole printed at 100mm/s:

| Raft and Support

Because TPU is sticky, the direct adhesion between layers will be firm. In order to ensure the flatness of the model raft surface, it is recommended to not enable raft printing when printing models with TPU.


At present, there is no configuration of easy-to-remove support material for TPU. Please use self-support printing for some supports, but pay attention to the gap between the support and model, which can be appropriately increased. For example, when using TPU95A to print the model, the gap between the support and model should be between 0.23mm and 0.25mm.
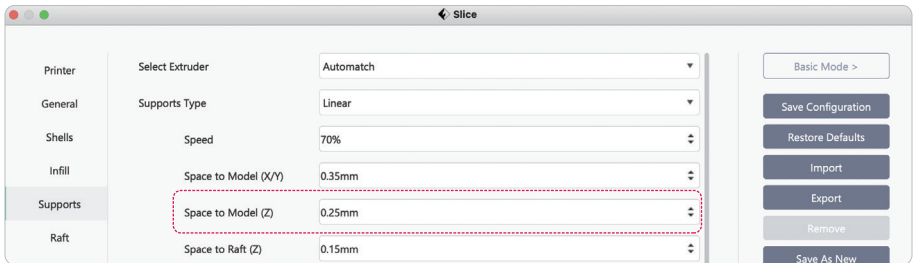
▎Shell and Top Solid

In order to obtain a soft TPU model, we can minimize the number of shells and the top solid layers, and even change the number of the top solid layers and the bottom solid layers to 0 when printing some models.
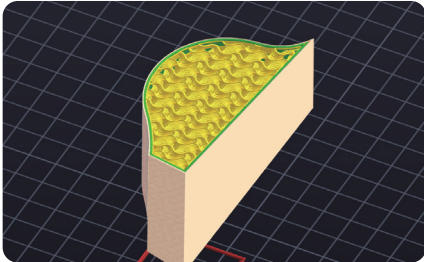
▎Infill Density and Infill Method

When slicing, the greater the infill density is, the higher the printed model strength (hardness) is. When at the same infill density (20%), the model strength is in the following order: 3D Infill > Hexagon > Line > Triangle, and the printing time is in the following order: 3D Infill < Line < Hexagon < Triangle.
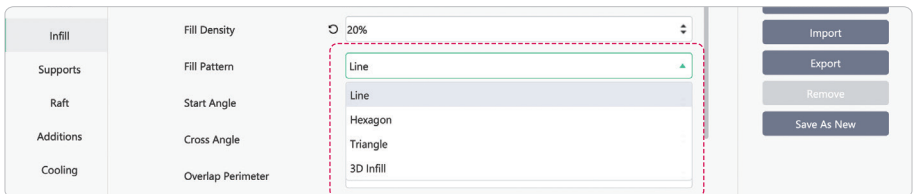
4 Demonstration Process of Adjusting Printing Parameters

As the most important link to improve the printing success rate and printing effect, printing parameters need to be constantly adjusted and explored by users according to the actual situation. FlashPrint 5, Flashforge's slicing software, also provides users with some default parameter configurations of filaments for your convenience. At the same time, you can save the relevant configuration with one click, so that the parameters can be reused.

The following is a demonstration of the process of adjusting the parameters of TPU filament printed with Creator 4. It follows the principle of starting with simple models and progressing to complex models.

| Confirm basic parameters: extrusion ratio, printing temperature, printing speed, top solid, etc.

In the FlashPrint 5 slicing software, find the default configuration parameters of TPU filament that is consistent with the hardness of the actually used filament (for the default parameters, Kexcelled-TPU(90A) is taken as an example) for slicing.

⚠️ **Note: FlashPrint 5 presets the printing parameters of some filaments, including third-party filaments. If the brand of filament actually used by users is not within the preset range, TPU hardness type can be used as the standard as the initial choice.**

For example, if you actually use TPU filament (here, take Polymaker-TPU(90A) as an example) to print, you can save the parameters as the name of the actually used filament when slicing.
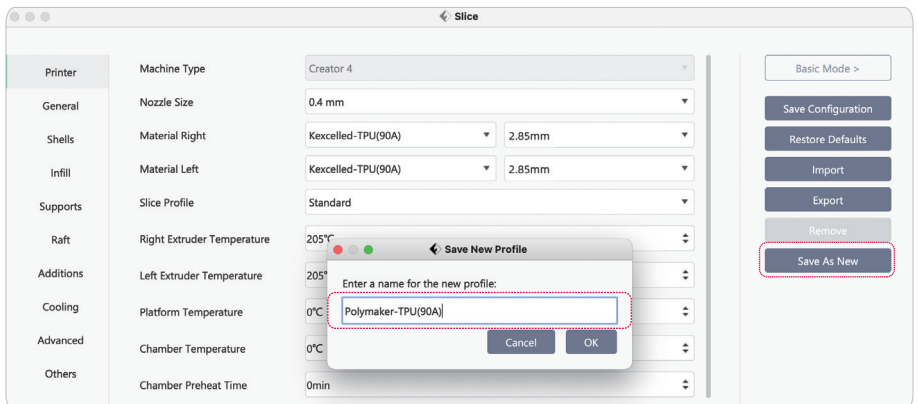
For the box model used as the example, the access path is:
File → Examples → 20mm _ Box.stl.
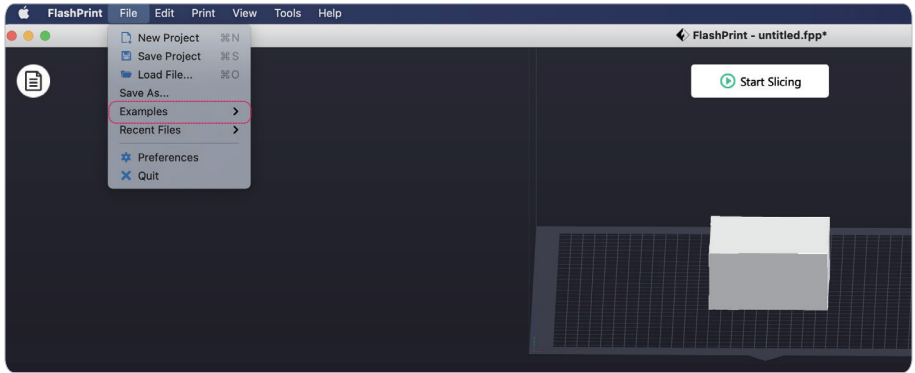
After finishing printing the sliced model, we checked the actual surface effect of the model: the overall filament extrusion is even; it has glossy surface finish; but the top solid effect is not good enough (that is, the internal infill texture can be seen).
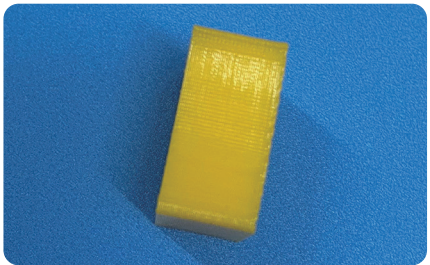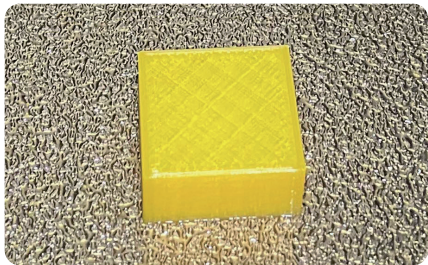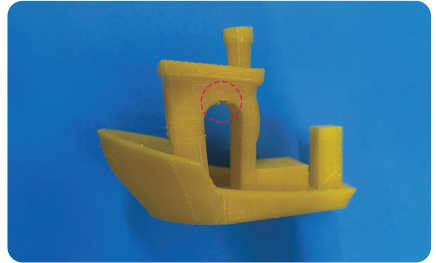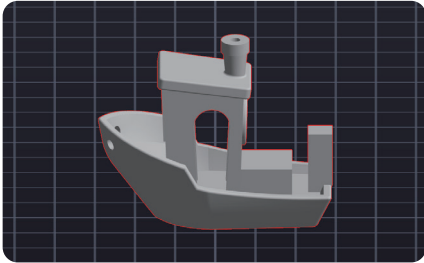

Therefore, we adjusted the number of top solid layers. We changed the original top solid layers from 2 to 3.
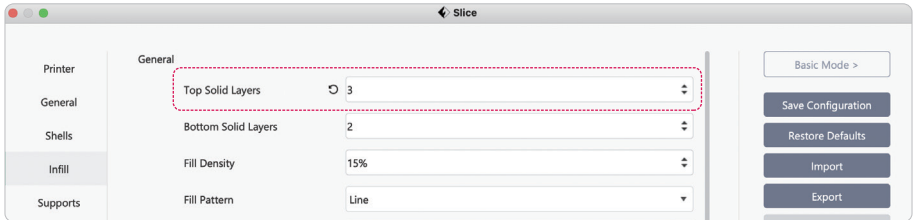
On the basis of step 1, reconfirm the basic parameters: retraction, shell, cooling, etc.

On the basis of step 1, confirm whether the printing parameters are appropriate again. Here, we chose the 3DBenchy model.


According to the adjusted parameters, the actual printing effect: the overall printing effect is good, but there are some missing lines on the hull.

For the above mentioned phenomenon of missing lines, we adjusted the retraction parameters. We changed the default retraction length of 5mm and retraction speed of 45mm/s to the retraction length of 1.5mm and retraction speed of 30mm/s.
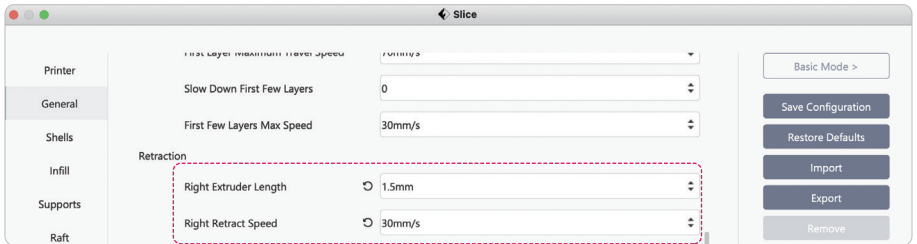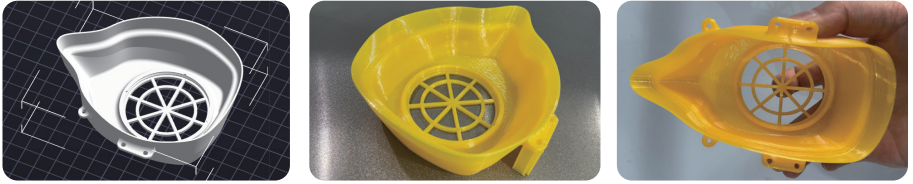

After finishing printing the re-sliced model, the actual printing effect: the model has good molding effect with smooth surface and no stringing; the hull is of strong three-dimensional effect.
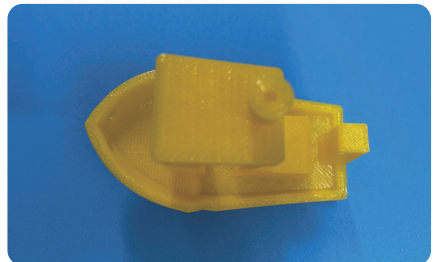
❙ Confirm the overall print quality

On the basis of step 2, we tried printing some complicated models and selected the mask shell model for printing.


Actual printing effect: the overall molding effect is good; the self-support is easy to remove; it is soft and practical.

After multiple verification, the relevant parameters can be saved as the default print data of similar filament, and then it is very convenient to print the relevant TPU filament again afterwards.

5 Suggestion for Material Preservation

TPU filament is very hygroscopic, meaning that it sucks up a small amount of moisture from the air. If there is no good storage environment, TPU filament will start to get wet in just a few days, so that the model printed with TPU has poor surface effect, with tiny holes and stringing. We can remove the accumulated moisture in the filament through our filament drying station, and restore the properties of TPU filament, so as to obtain smooth printing effects. For daily storage, it is also recommended to store it in a dry environment or directly use a special filament dryer.
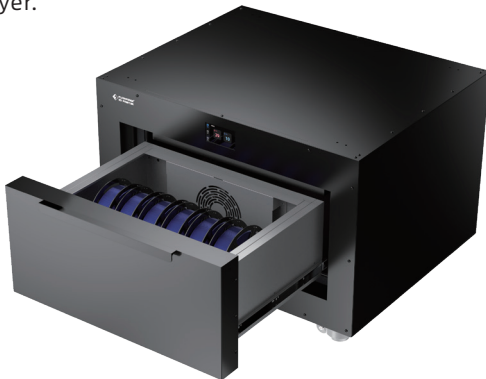
The above is the guide for printing TPU filament with Creator 4, which can be used as a reference for the actual printing of similar filaments to obtain better printing results.

FLASHFORGE®
3D PRINTER

Follow us

Zhejiang Flashforge 3D Technology Co., Ltd.

Address: No.518 XianYuan Road, Jinhua City, Zhejiang Province, China

Service Hotline: +86 579 82273989

support@flashforge.com