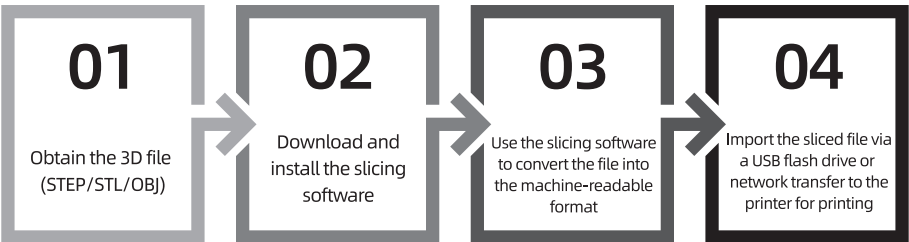


Slicing Software Instructions for Adventurer 5M Series

Note Before reading the brief instructions, please ensure you have reviewed the Quick Start Guide and completed the first print.

Before printing 3D model files, you need to configure slicing presets for the corresponding printer. Recommended slicing software: **OrcaSlicer** / **FlashPrint 5**.

Pre-printing Steps:



Open-source Slicing Software
OrcaSlicer

This slicing software, created by the open-source community, offers more open configuration options. Experienced users are recommended to use this slicing software.

Download and Installation Instructions

1. Download the latest slicing software from the official website:
<https://github.com/SoftFever/OrcaSlicer/releases>.
2. Find the OrcaSlicer software package on the USB flash drive and install the version that matches your system.

Note

Files can be imported by project, requiring a click on the project file each time you open it. Alternatively, configs can be imported, eliminating the need to do so each time, but this may not be compatible with higher software versions.

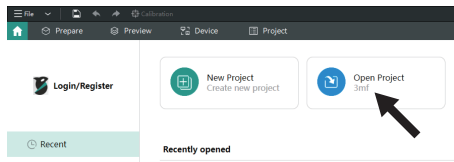
How to Use OrcaSlicer (Import Project Files)

* The steps are illustrated for one machine type.

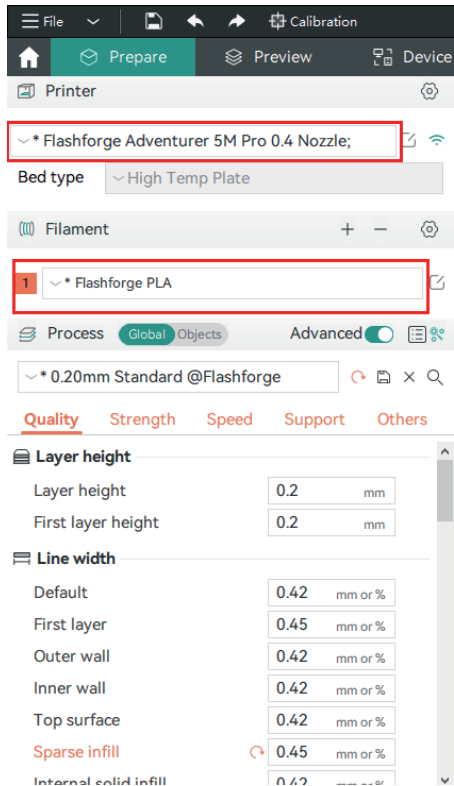
1. Open the installed OrcaSlicer.



2. Click [Open Project].

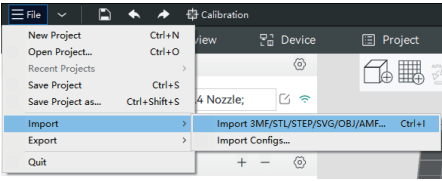


3. The profile should be in .3mf format. You can find the corresponding file on the USB flash drive. Drag it directly into OrcaSlicer, or open it directly (if the machine is not configured with a USB flash drive, please download the corresponding profile from Flashforge's official website).

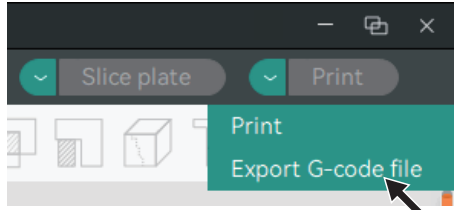
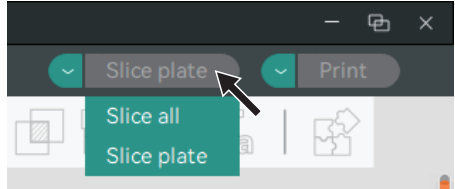


4. After importing the profile, the software interface will display the corresponding printer, and you can select the desired printing material.

5. Select the model file to be printed. You can drag it directly into the software, or click [File] - [Import] to import the model file (STL/STEP/OBJ/3MF, etc.).



6. Click [Slice all]. Once slicing is completed, click [Export G-code file], save the file to a USB flash drive, and then insert it into the printer for printing.



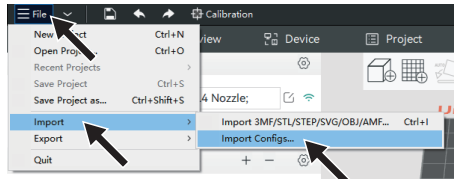
How to Use OrcaSlicer (Import Configs)

* The steps are illustrated for one machine type.

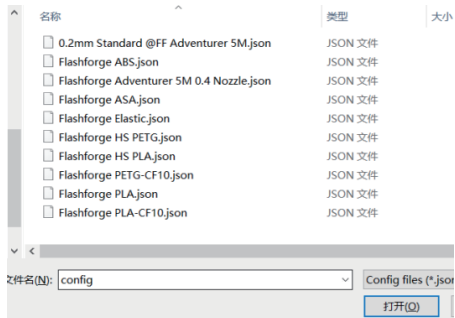
1. Open the installed OrcaSlicer.



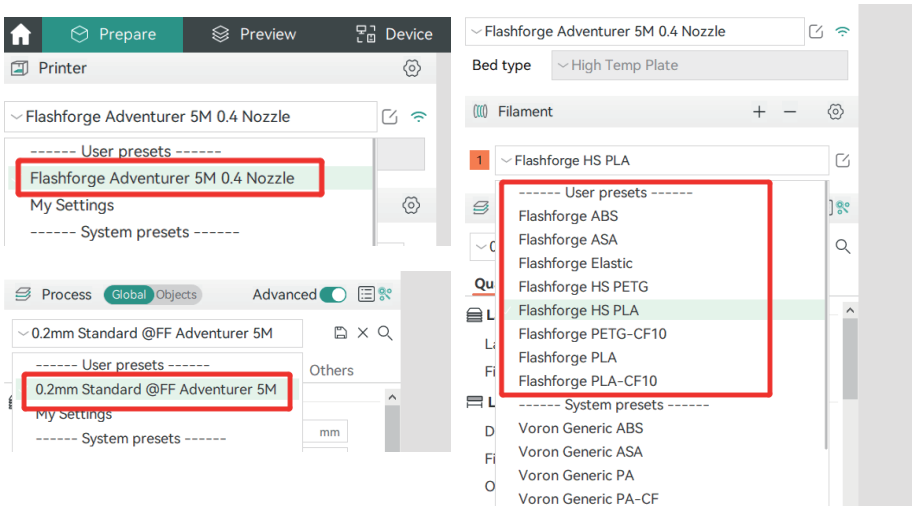
2. Click [File] - [Import] - [Import Configs...].



3. The config file should be in .json format. You can find the corresponding files on the USB flash drive, select all, and open it (if the machine is not configured with a USB flash drive, please download the corresponding config files from Flashforge's official website).

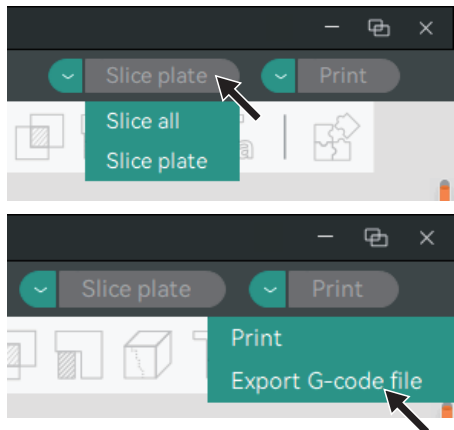


4. After importing, the corresponding printer, available filaments, and recommended parameters will be displayed.



5. Select the model file to be printed. You can drag it directly into the software, or click **[File] - [Import]** to import the model file (STL/STEP/OBJ/3MF, etc.).

6. Click **[Slice all]**. Once slicing is completed, click **[Export G-code file]**, save the file to a USB flash drive, and then insert it into the printer for printing.



Flashforge's Official Slicing Software - FlashPrint 5

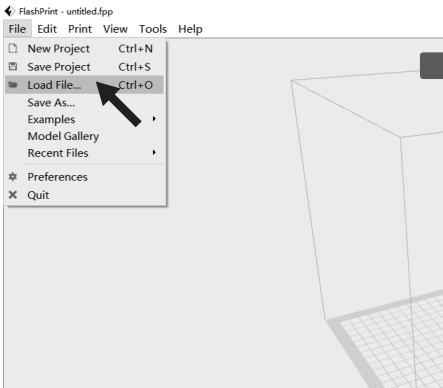
* The steps are illustrated for one machine type.

FlashPrint 5 is not open-source. It is user-friendly and suitable for users with no 3D printing experience or those who don't require parameter adjustments.

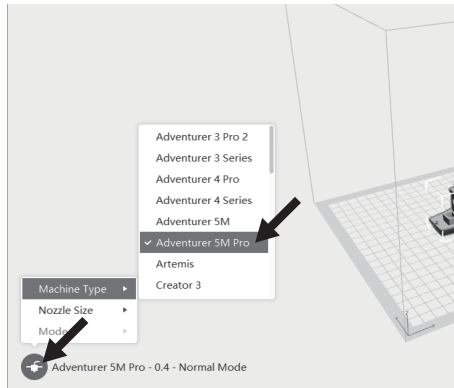
Download Instructions

1. Download the latest slicing software from the official website: <https://www.flashforge.com/download-center>.
2. Find the FlashPrint 5 software package on the USB flash drive and install the version that matches your system.

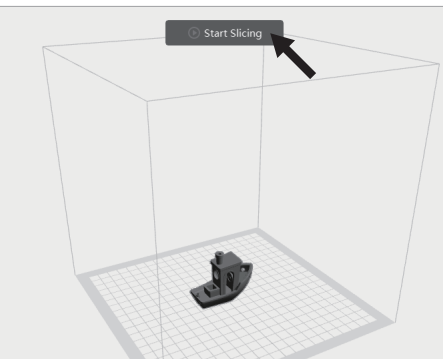
1. After installing the slicing software, import the model file.



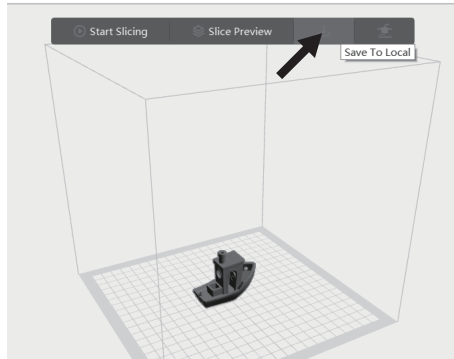
2. Select the corresponding printer type.



3. Click [Start Slicing].



4. After slicing is completed, save the file to a USB flash drive for printing.



Note

The slicing profiles available in FlashPrint 5 are configured based on extensive testing with various types of filaments. We recommend using the recommended temperature settings provided in the profiles. If you believe a specific filament requires a different temperature, you can make minor adjustments and print smaller objects at the set temperature for testing to ensure smooth operation.

冒险家5M系列 配套切片软件简要说明

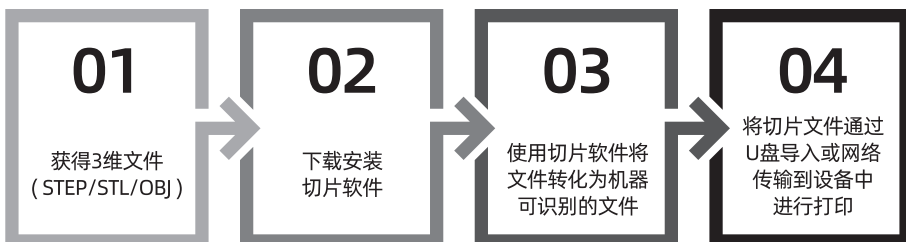
⚠ 注意事项

阅读软件简要说明前，请确认已查看快速启动指南，并已经完成第一次开机打印。

三维模型文件打印前需要先进行对应机器的切片预设才能进行打印。

推荐使用切片软件：**OrcaSlicer / FlashPrint 5。**

打印前置步骤为：



开源切片软件
OrcaSlicer

该切片软件由开源社区创建，拥有更多的开放配置参数。

用户若有一定的3D打印经验，建议使用此切片软件。

下载安装路径

1. 从官方网站 <https://github.com/SoftFever/OrcaSlicer/releases> 下载最新的切片软件。
2. 在U盘中找到OrcaSlicer软件安装包，选择对应的系统版本进行安装。

提示

注：文件可按项目导入，每次打开都需要点击一次项目文件。

文件按预设导入，按预设导入时无需每次打开，但是软件更高版本时将无法兼容预设文件。

如何使用OrcaSlicer (按项目文件导入)

* 图中仅用一种机型示意使用步骤讲解

1. 打开安装好的OrcaSlicer软件。

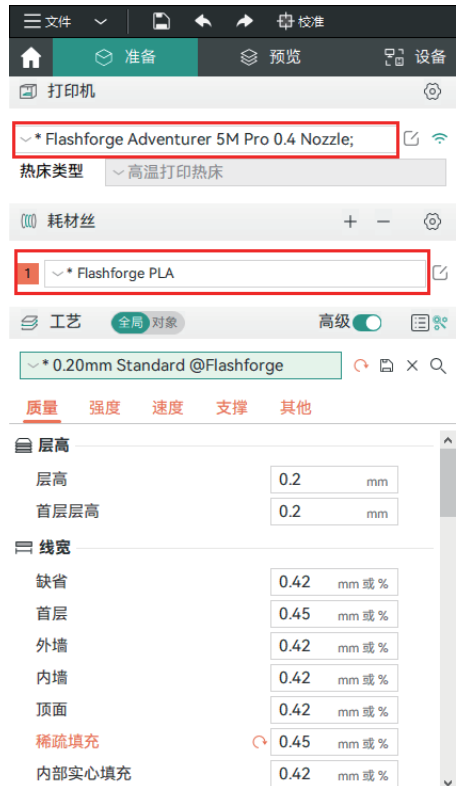


2. 点击[打开项目]。

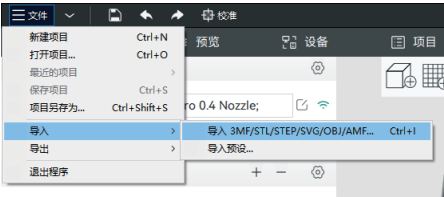


3. 配置文件的格式为.3mf, 可从U盘中找到对应配置文件, 直接拖入Orca软件或直接打开 (若设备未配置U盘请从闪铸官网下载对应机型配置文件)。

4. 导入配置文件后, 软件界面将会显示对应机型, 可选择需要打印的材料。



5. 选择需要打印的模型文件，可直接拖入软件或者点击[文件]导入STL/STEP/OBJ/3MF等格式的文件。



6. 点击切片按钮，待切片完成后点击[导出G-code文件]，将文件存入U盘，然后将U盘插入设备即可打印。



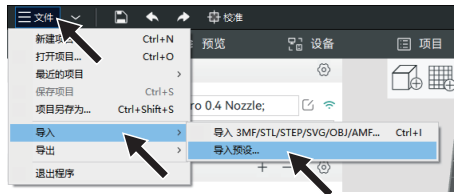
如何使用OrcaSlicer (按预设导入)

* 图中仅用一种机型示意使用步骤讲解

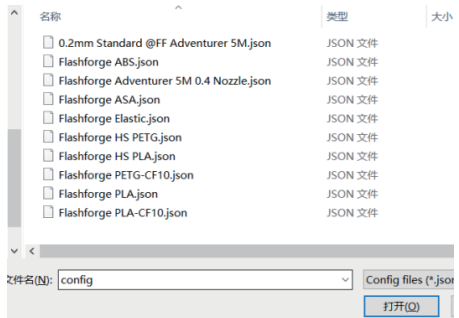
1. 打开安装好的OrcaSlicer软件。



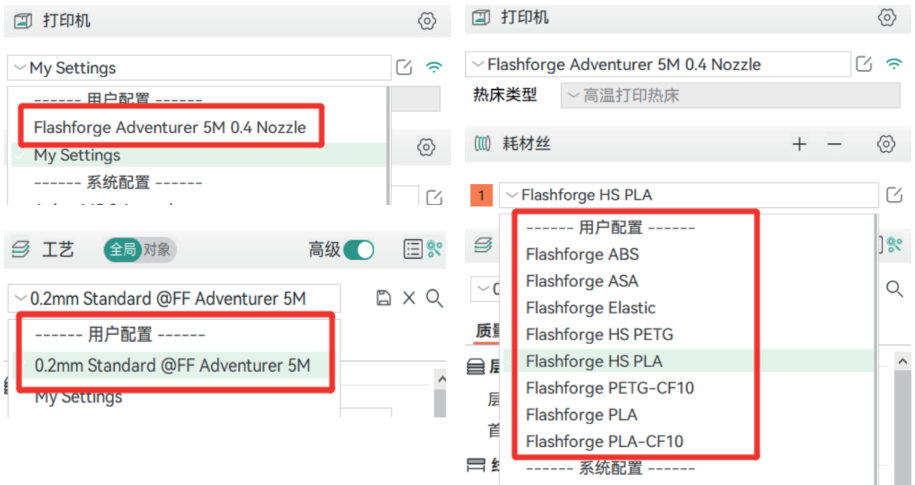
2. 点击[文件] - [导入] - [导入预设]。



3. 预设配置文件格式为.json，可从U盘中找到对应预设文件，直接全选打开即可（若设备未配置U盘请从闪铸官网下载对应机型配置文件）。

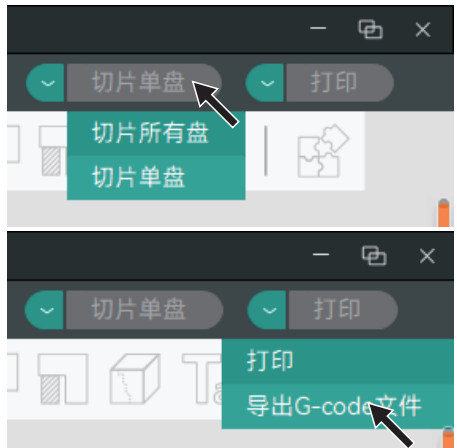


4. 导入完成后，在打印机、耗材丝、工艺栏的用户配置里分别会显示对应机型、可选材料及推荐参数。



5. 选择需要打印的模型文件，可直接拖入软件或者点击[文件]导入STL/STEP/OBJ/3MF等格式的模型文件。

6. 点击切片按钮，待切片完成后，点击导出G-code文件，将文件存入U盘，然后将U盘插入设备即可打印。



闪铸科技官方匹配的切片软件FlashPrint 5

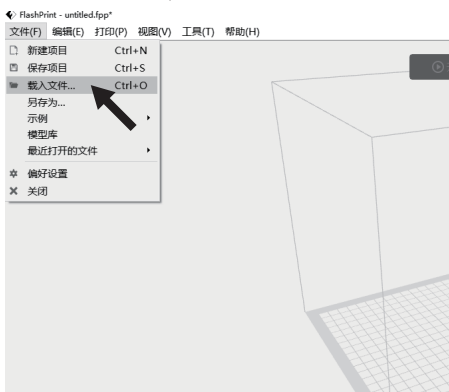
* 图中仅用一种机型示意使用步骤讲解

FlashPrint 5非开源软件，该软件简单易用，适合无3D打印经验或无参数调试需求的用户使用。

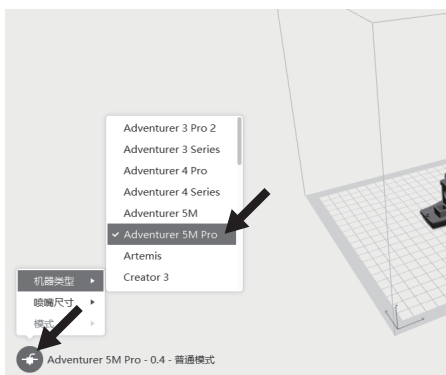
下载地址

1. 从官方网站 <https://www.sz3dp.com/download-center> 下载最新的切片软件。
2. 在U盘中找到FlashPrint5软件安装包，选择对应的系统版本进行安装。

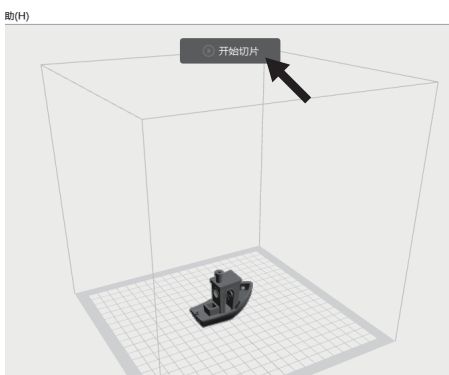
1. 安装切片软件后，导入模型文件。



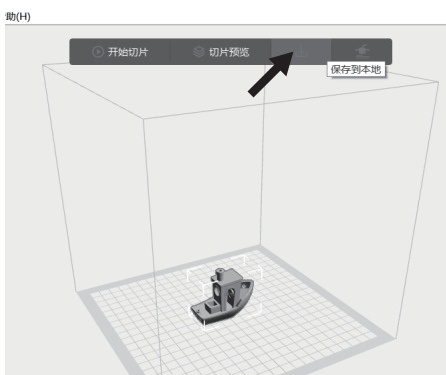
2. 选择对应机型。



3. 点击开始切片。



4. 完成切片后保存文件至U盘，可使用U盘进行打印。



⚠ 注意事项

FlashPrint 5中可用的切片配置文件是基于对各种类型耗材的广泛测试而配置的。我们建议使用配置文件中包含的推荐温度设置。如果您认为特定的耗材需要不同的温度，可进行微调，使用设定温度打印较小物体进行测试，确保一切顺利运行。