G-code Tutorial

Beijing Tiertime Technology Co., Ltd.

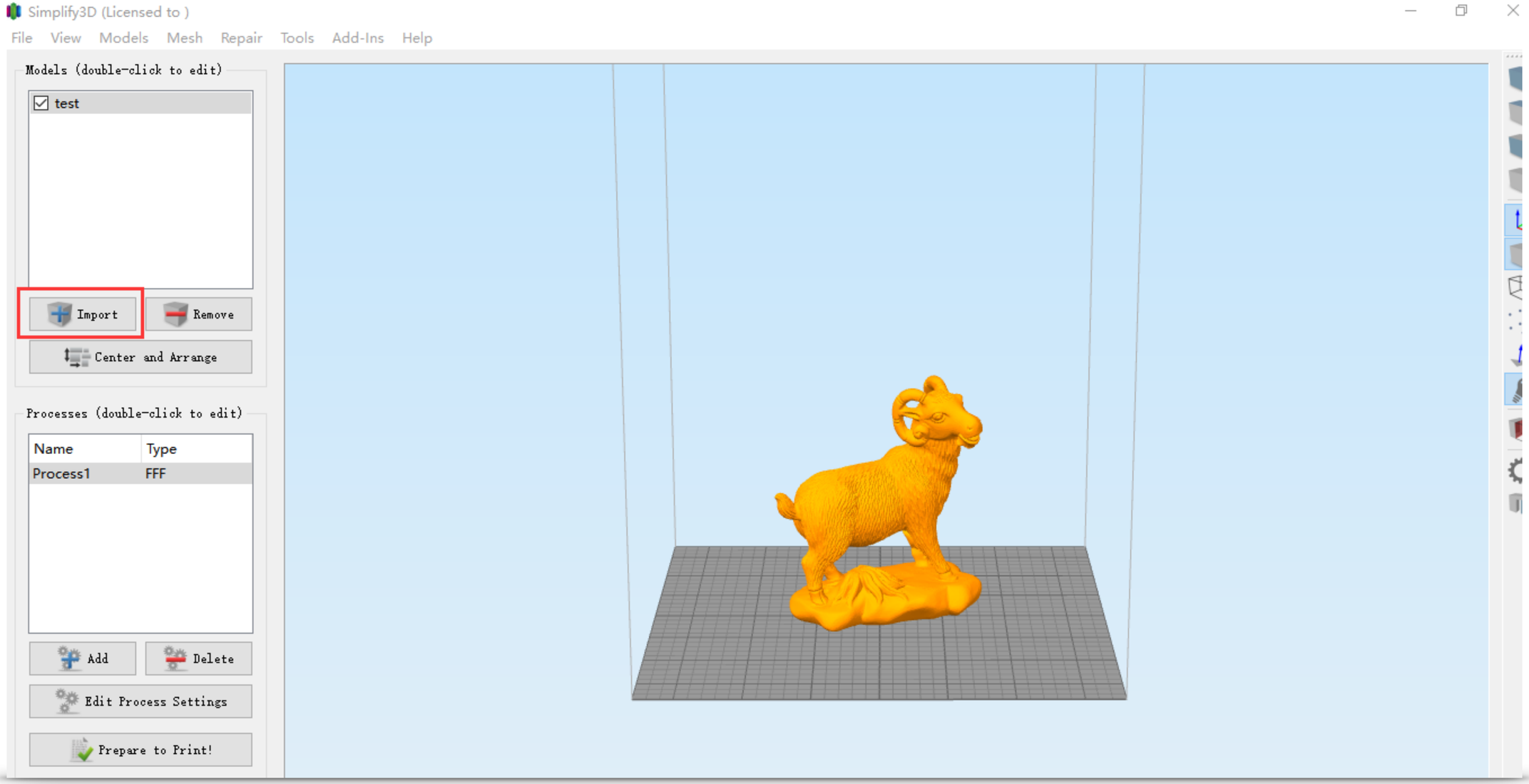
We need a slicer which can generate G-code:

You can use Simplify3D or other free slicers such as Slic3r, Cura, Kisslicer and etc.

Here we use simplify3D as an example.

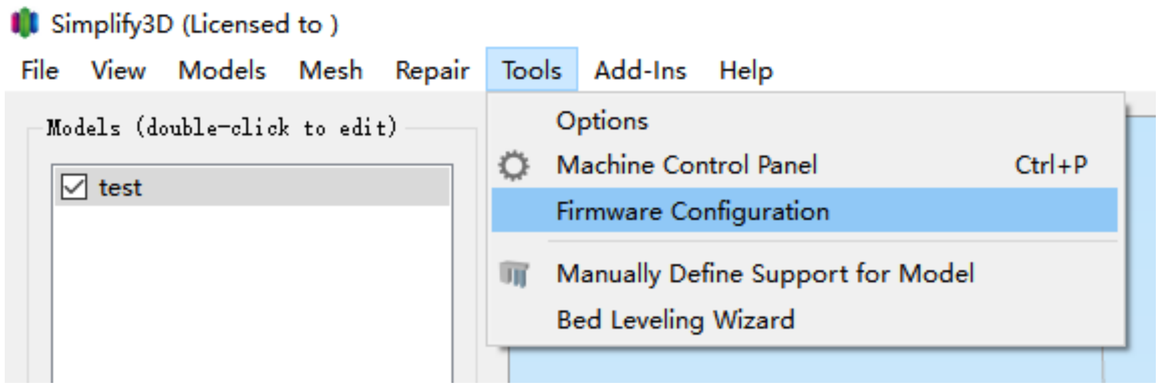
*G-code feature is only available in UP Studio **Version 2.0** and later.

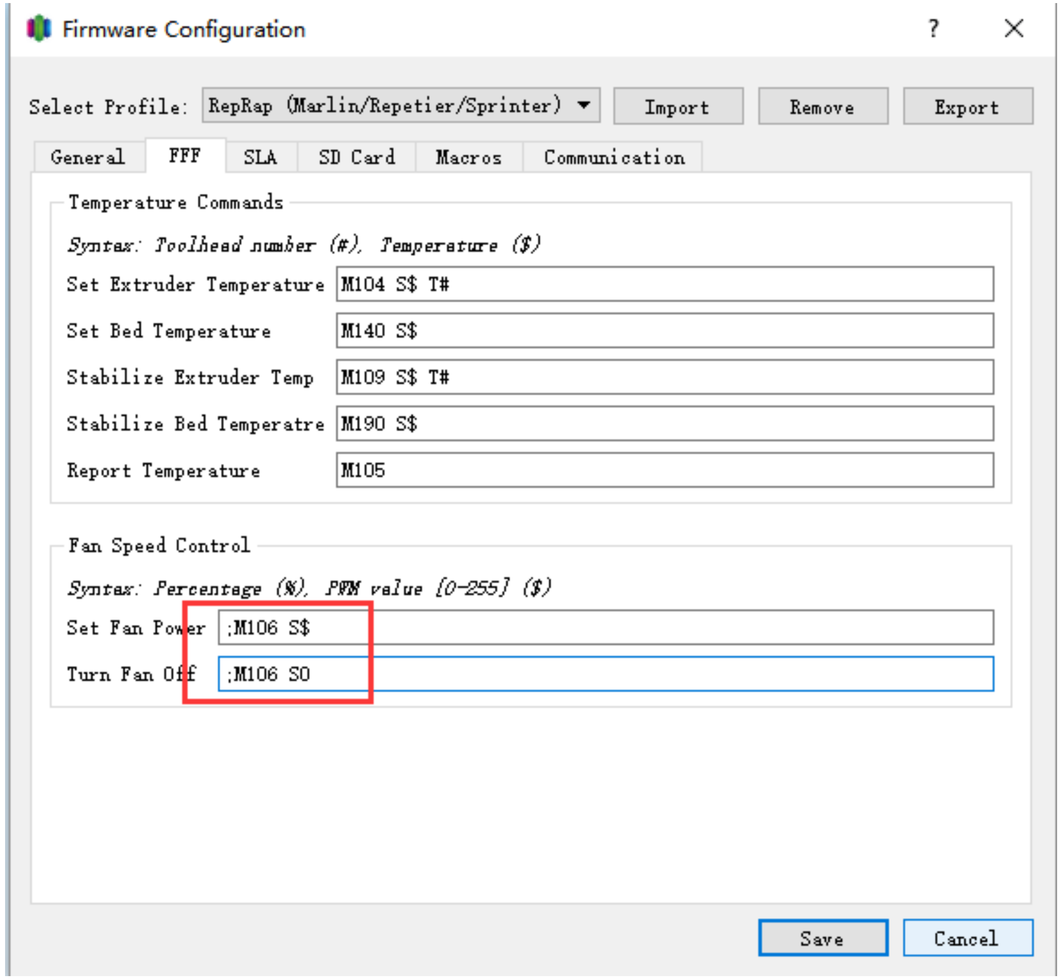
Step 1: Load a model



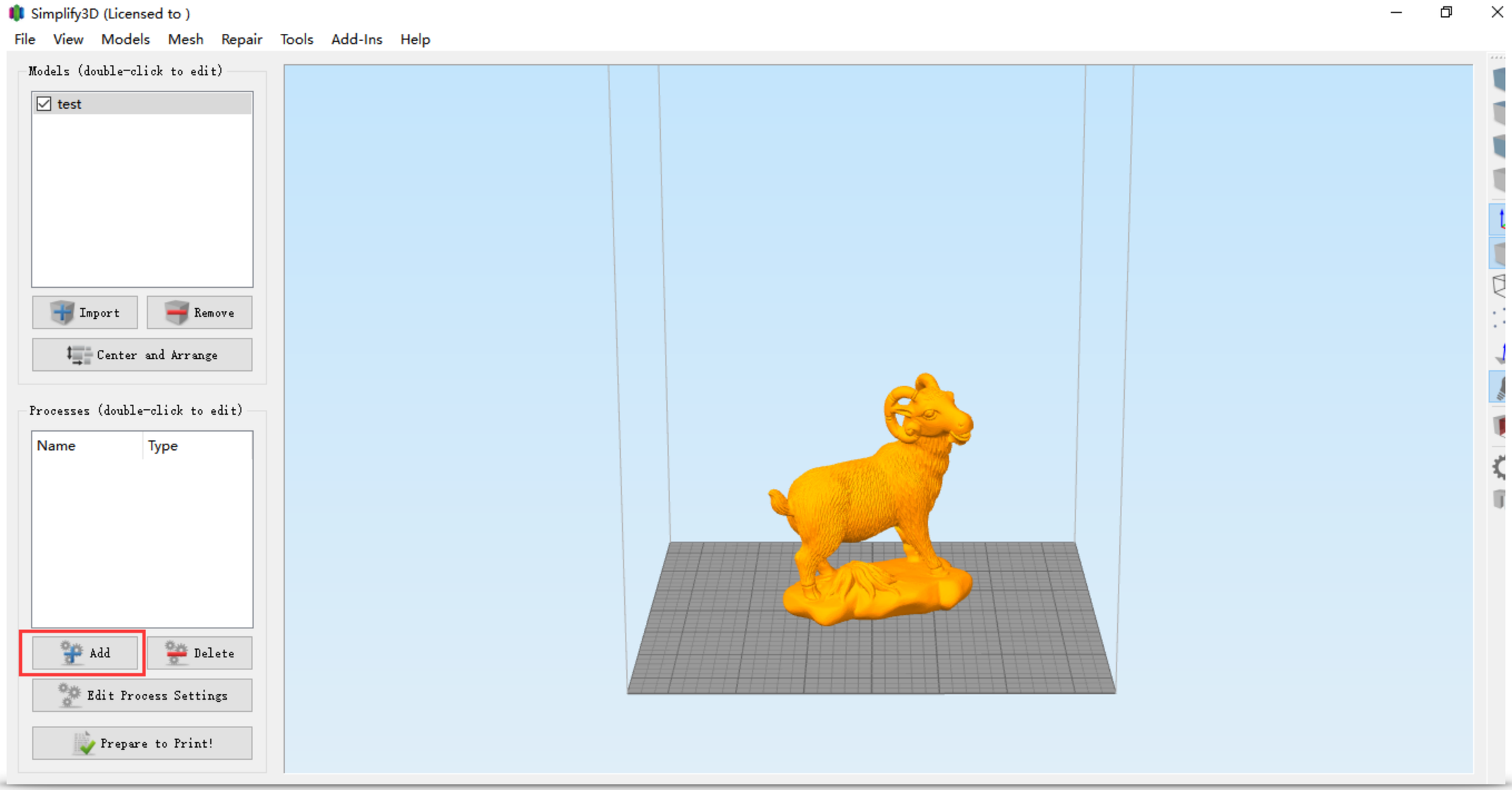
Step 2: Configure firmware

Since there is no software fan control，use“;”to disable to corresponding commands，remember to click Save when finish.





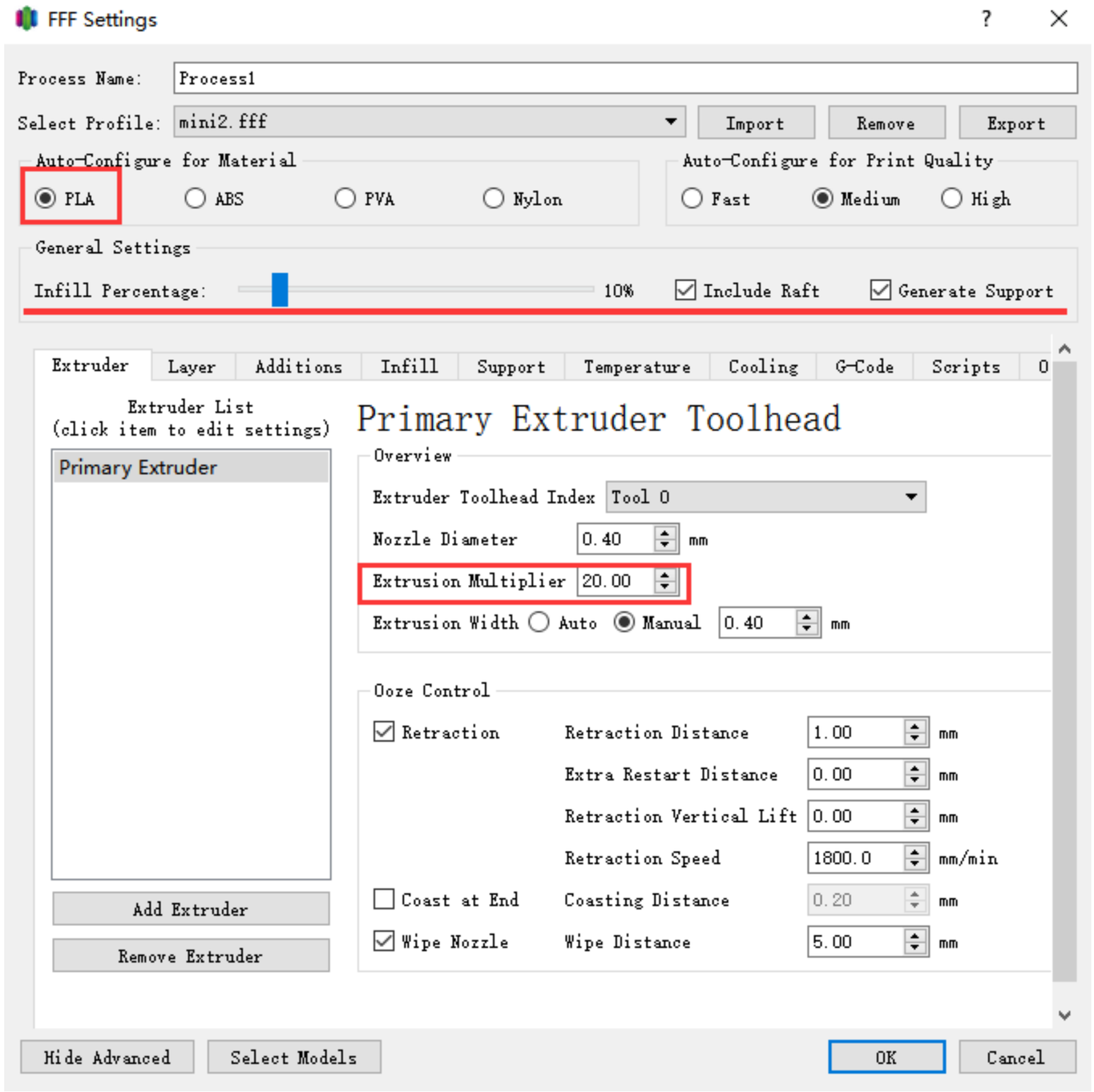
Step 3: Add a new Print Profile (FFF), or load existing profile.



Step 4: Profile Configuration

Extrusion multiplier is recommend to set to 20.

Other extrusion related parameter should also multiply 20, eg. retraction.

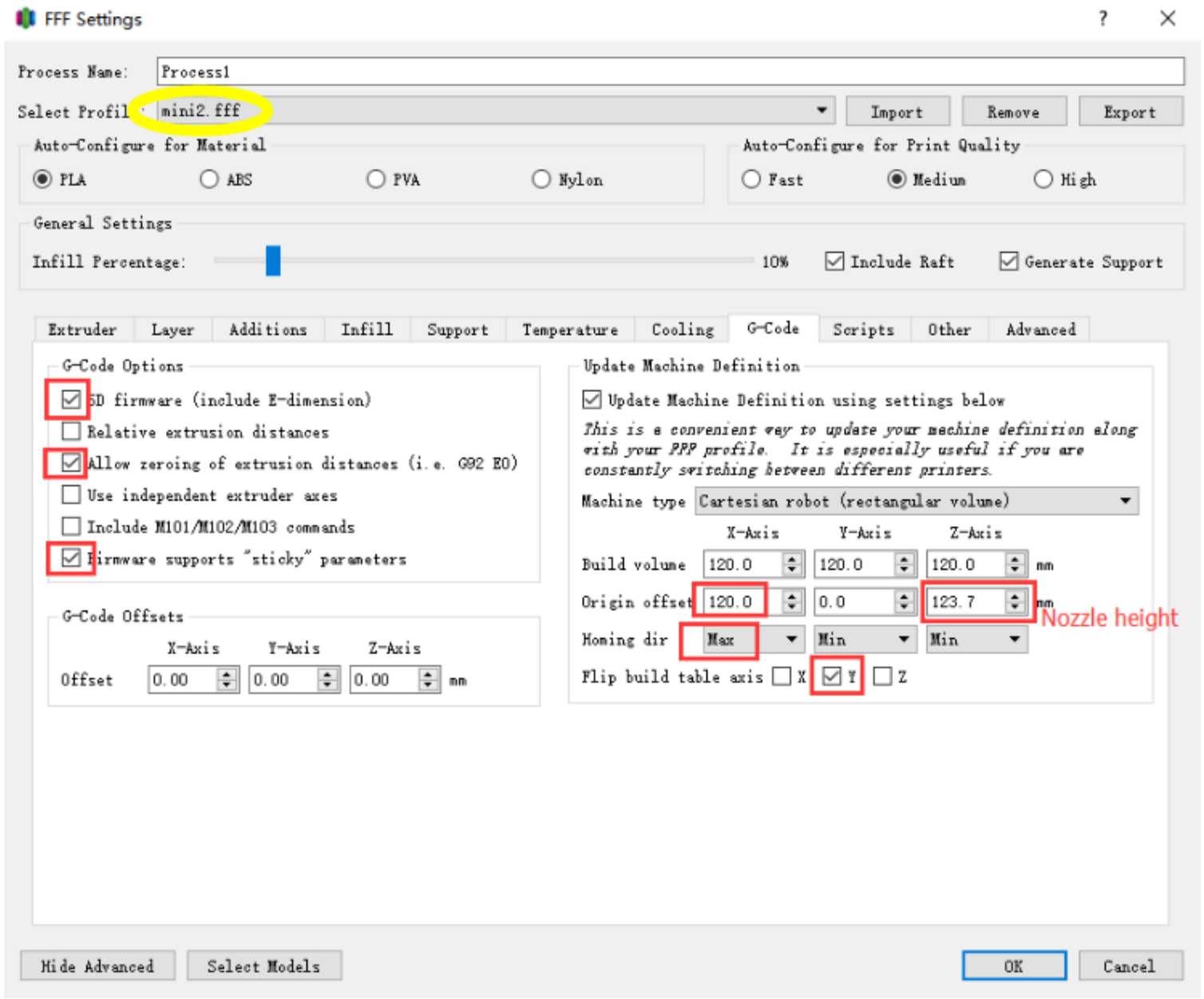


Step 5: Configure Printer

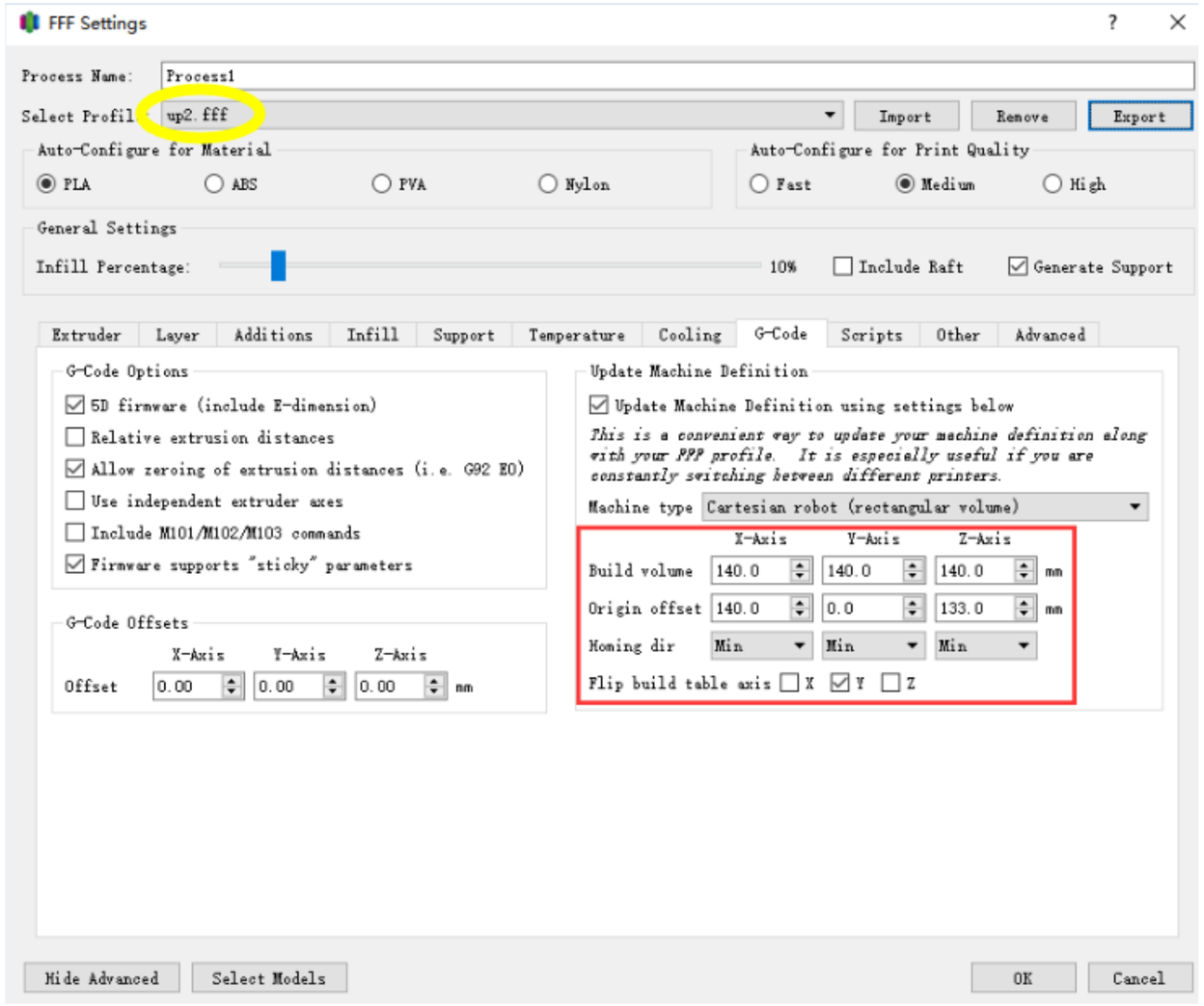
Use UP mini2 as an example: Go to G-code section and use the configuration shown in right.

Please note that the Origin Offset of Z axis should be equal to the nozzle height value measured from UP Studio, and you may still need to adjust this value to obtain the optimized first layer adhesion.

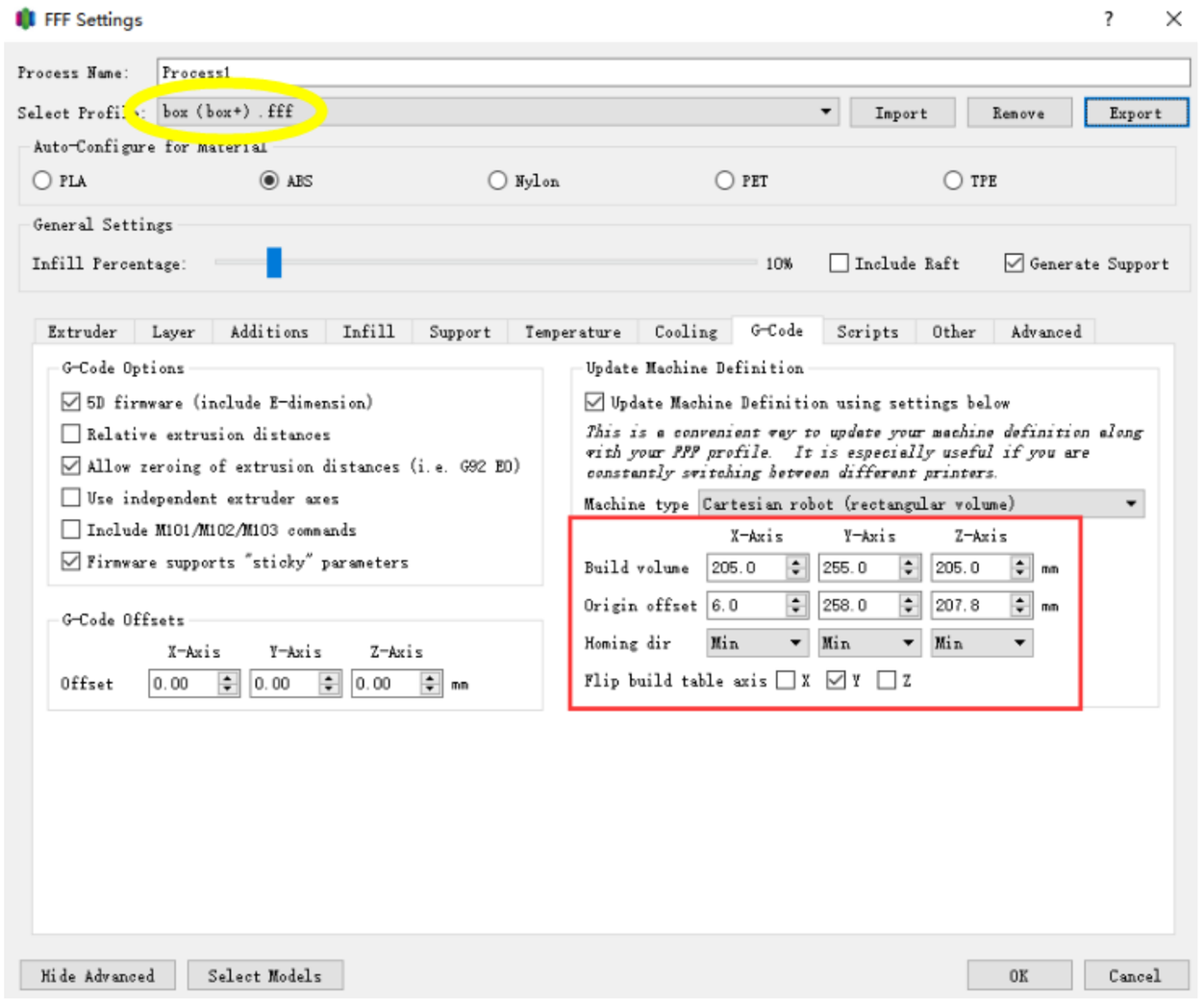
UP Plus 2 Configuration



UP Plus 2 Configuration

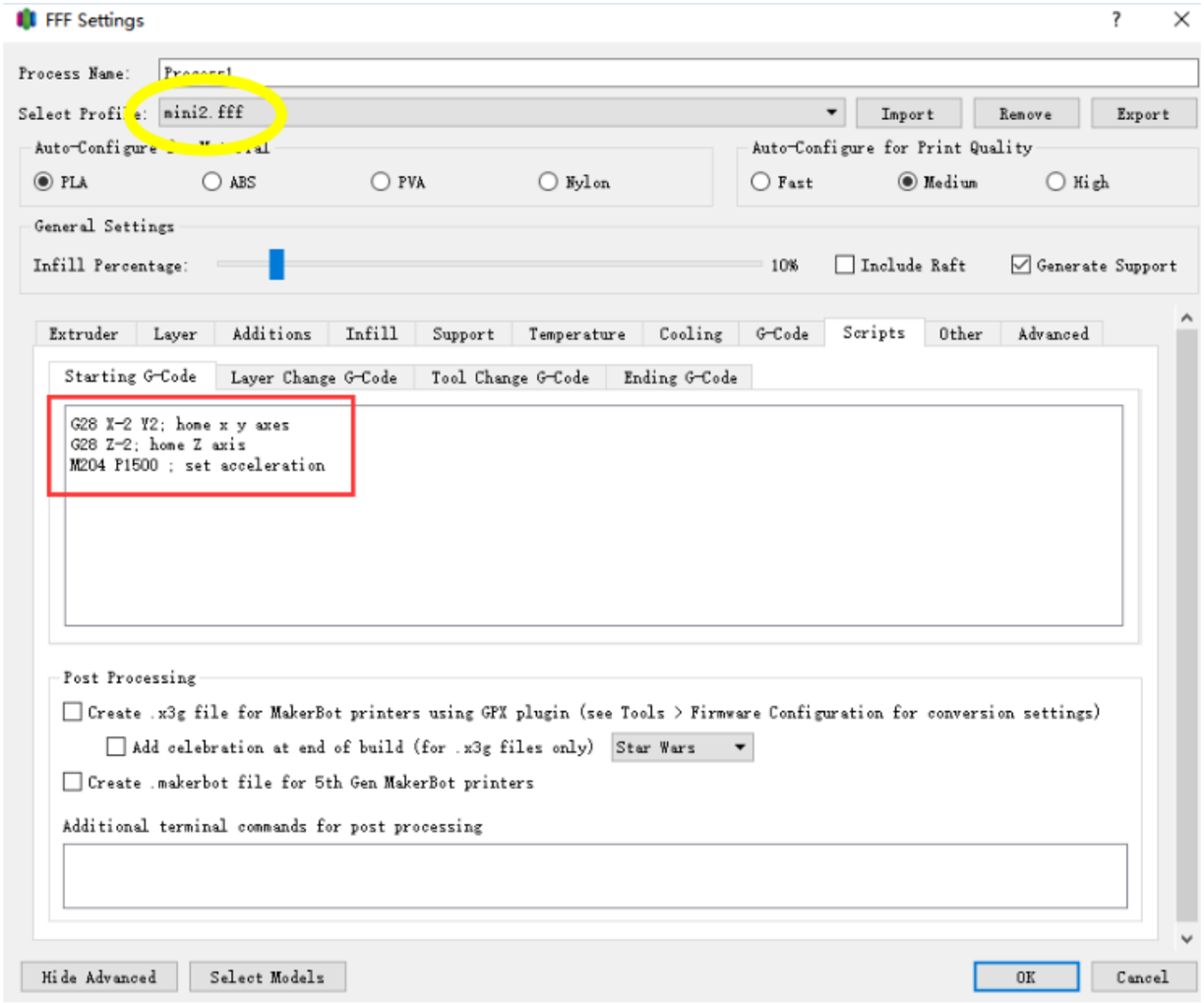


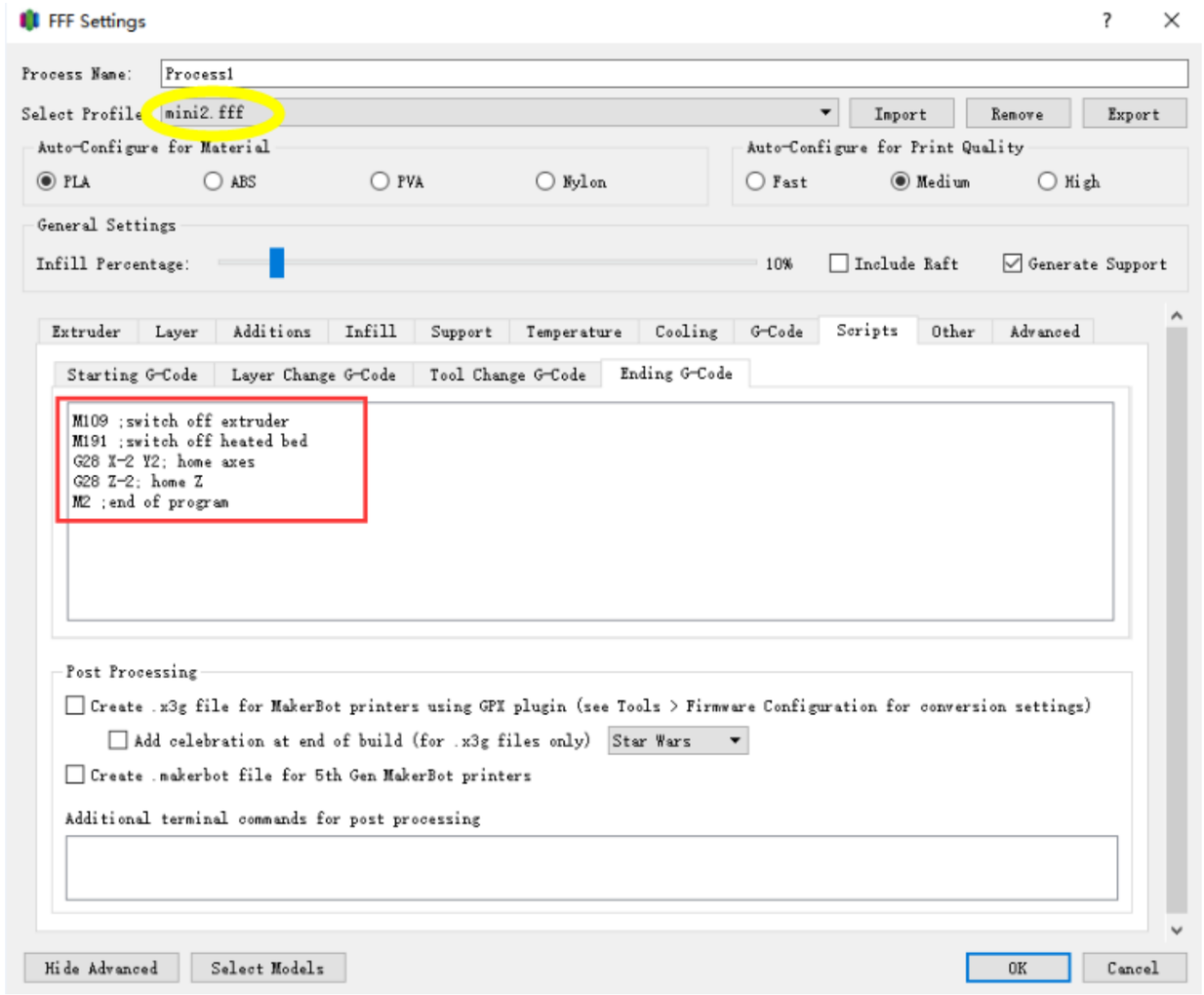
UP BOX/UP BOX+ Configuration



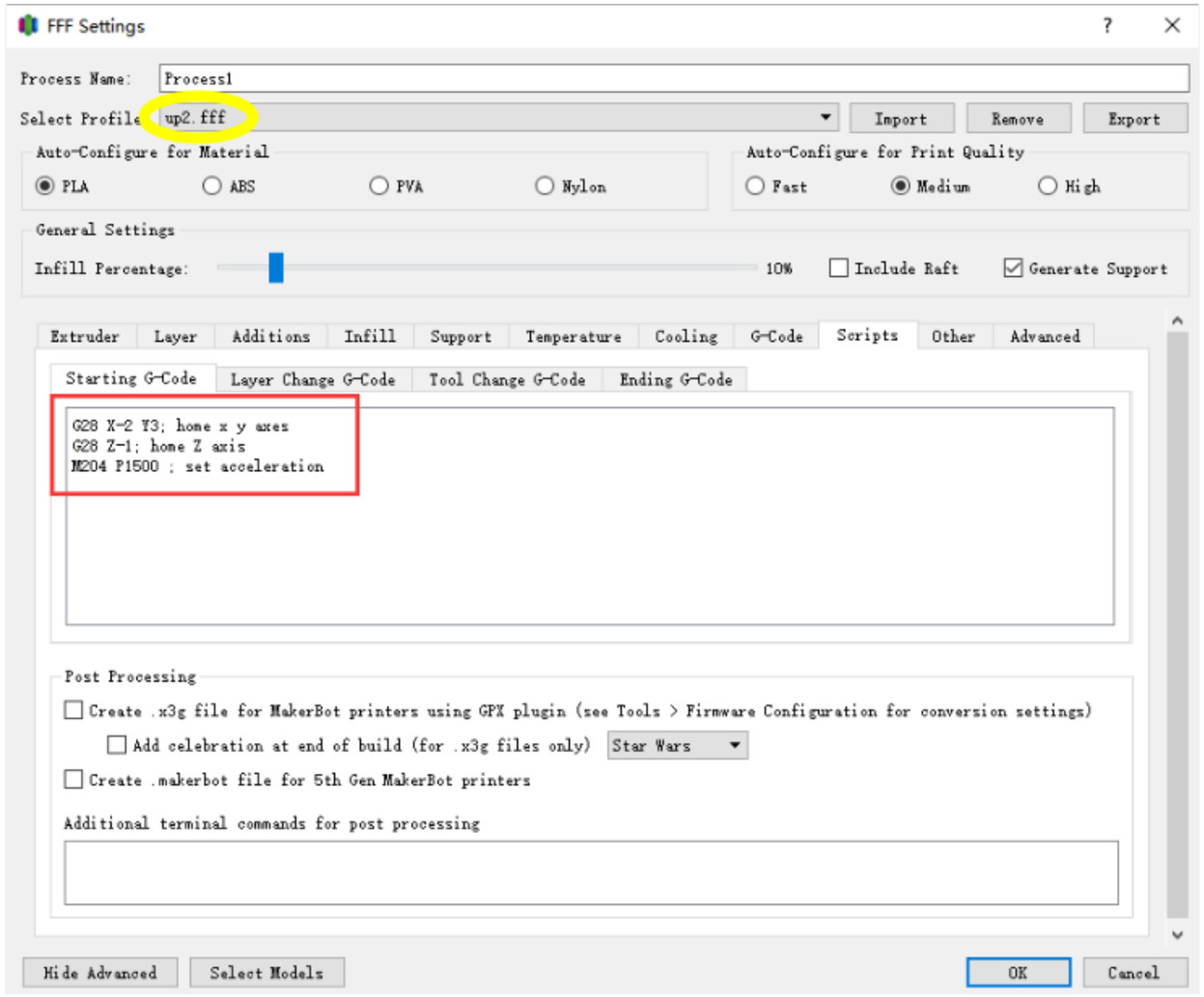
Step 6: Set Starting and Ending G-code

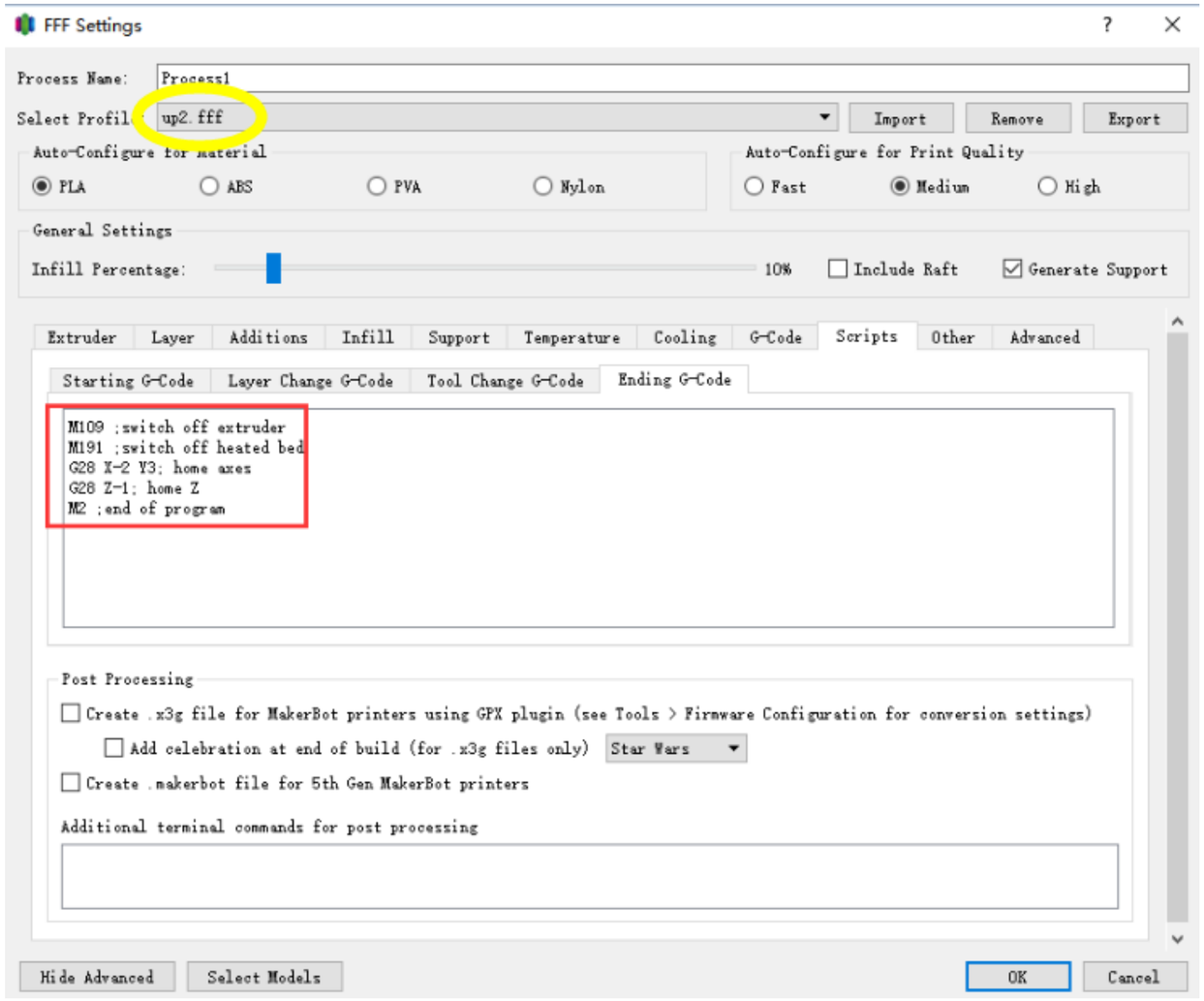
Starting and Ending G-code for UP mini 2:



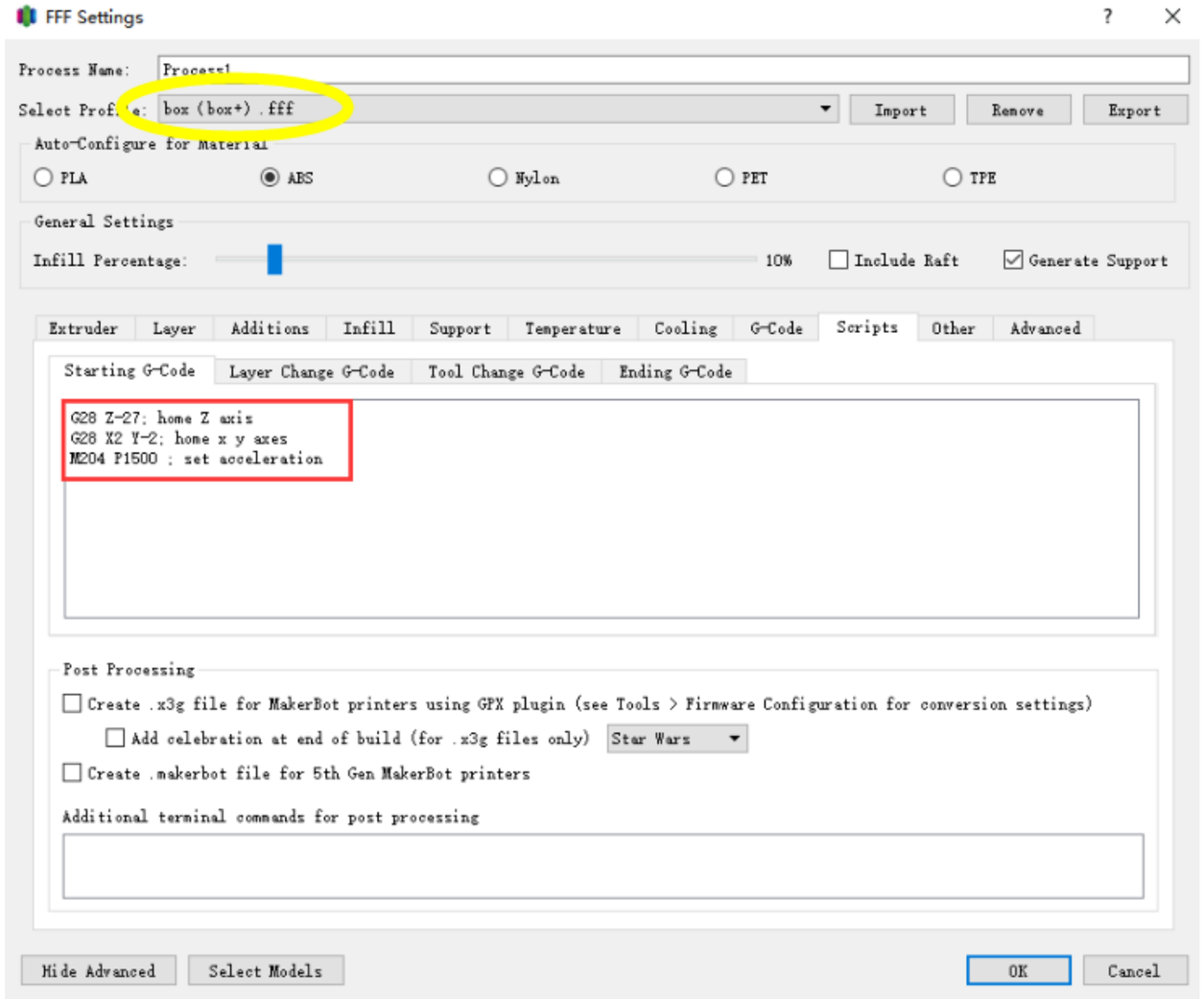


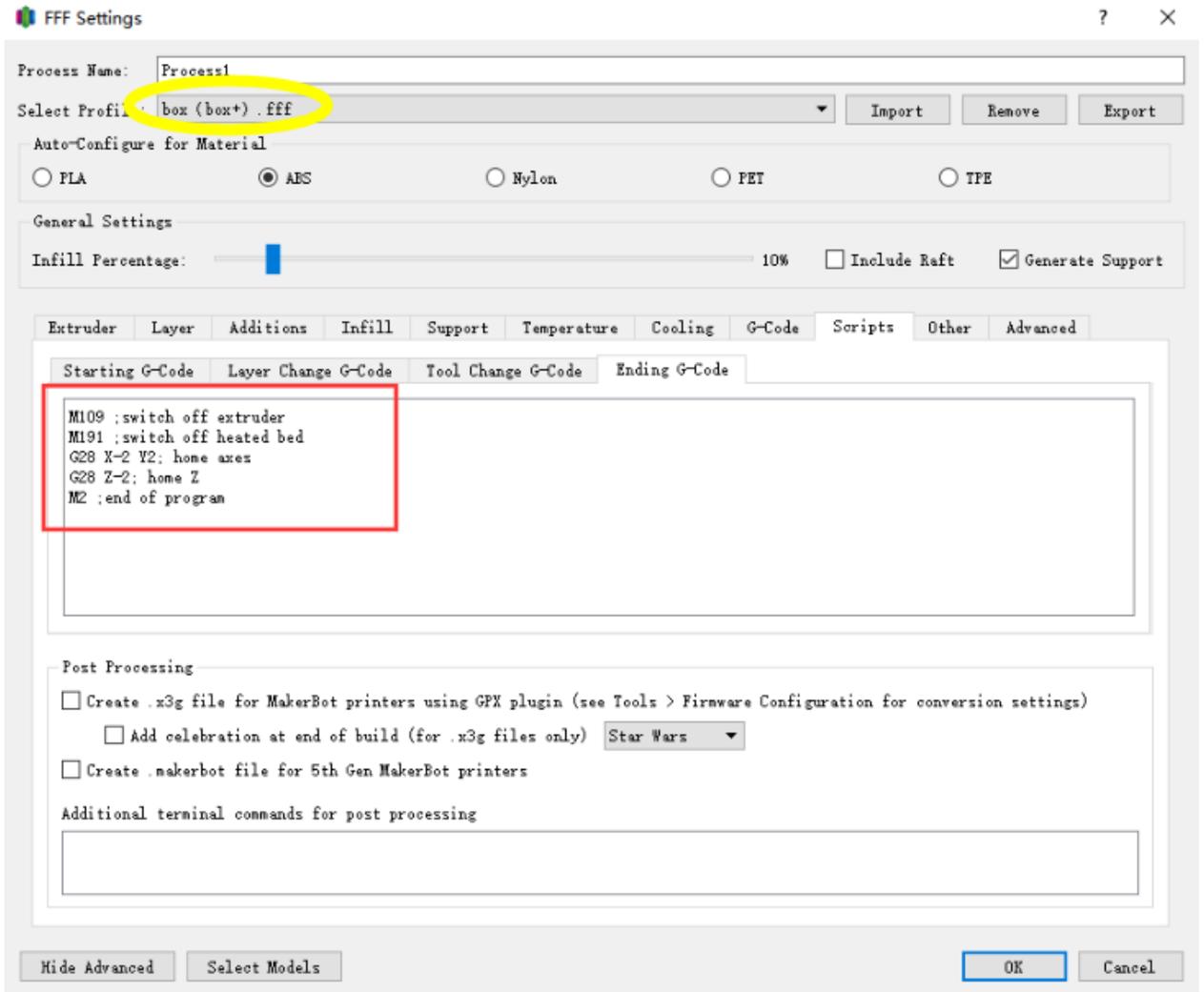
Starting and Ending G-code for UP Plus 2:



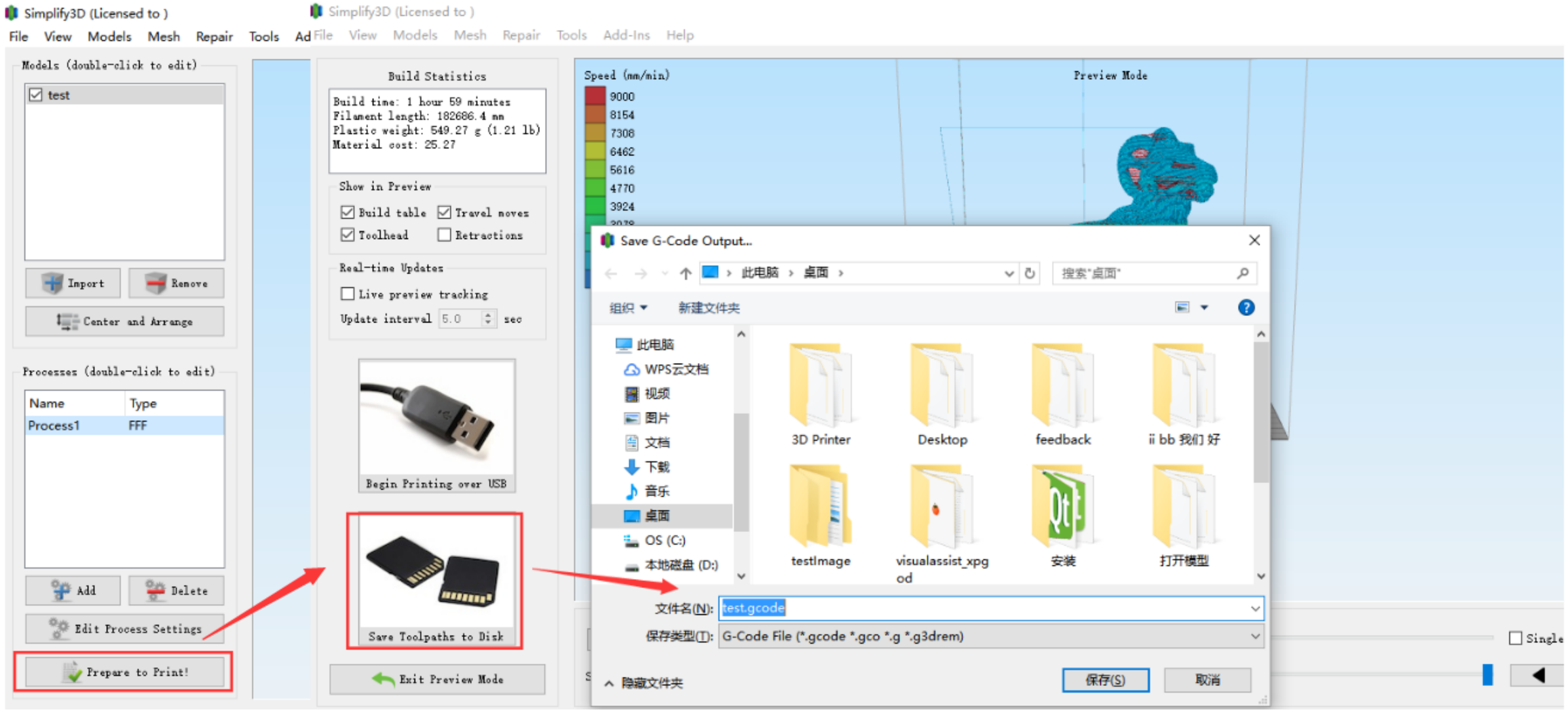


Starting and Ending G-code for UP BOX/UP BOX+:





Step 7: Generate G-Code



Step 8: Add G-code into UP Studio