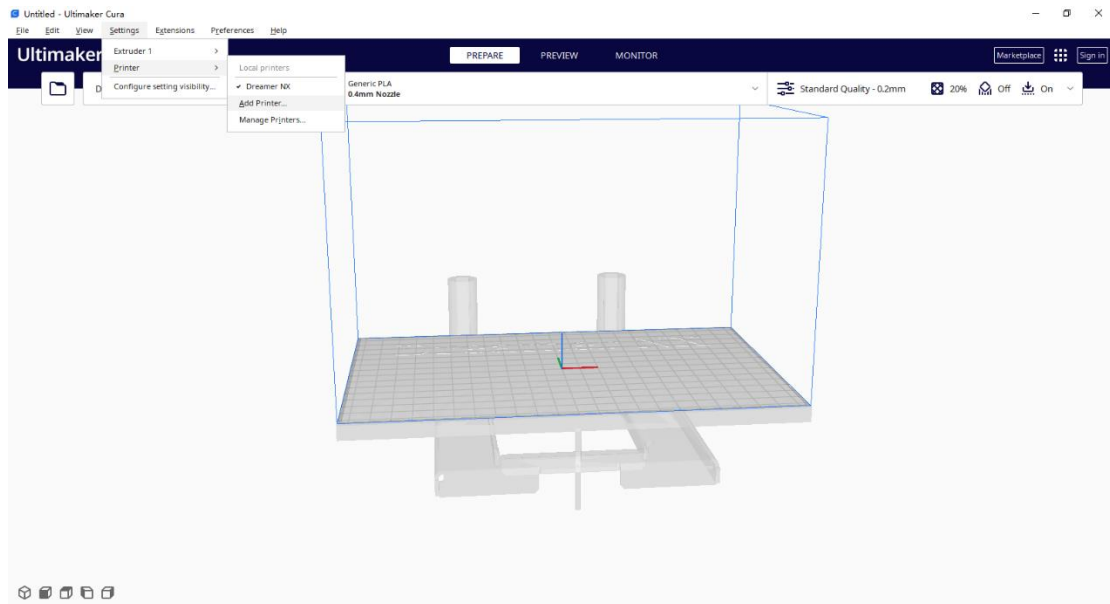


Cura Setup for Flashforge Creator 3/Creator 3 Pro

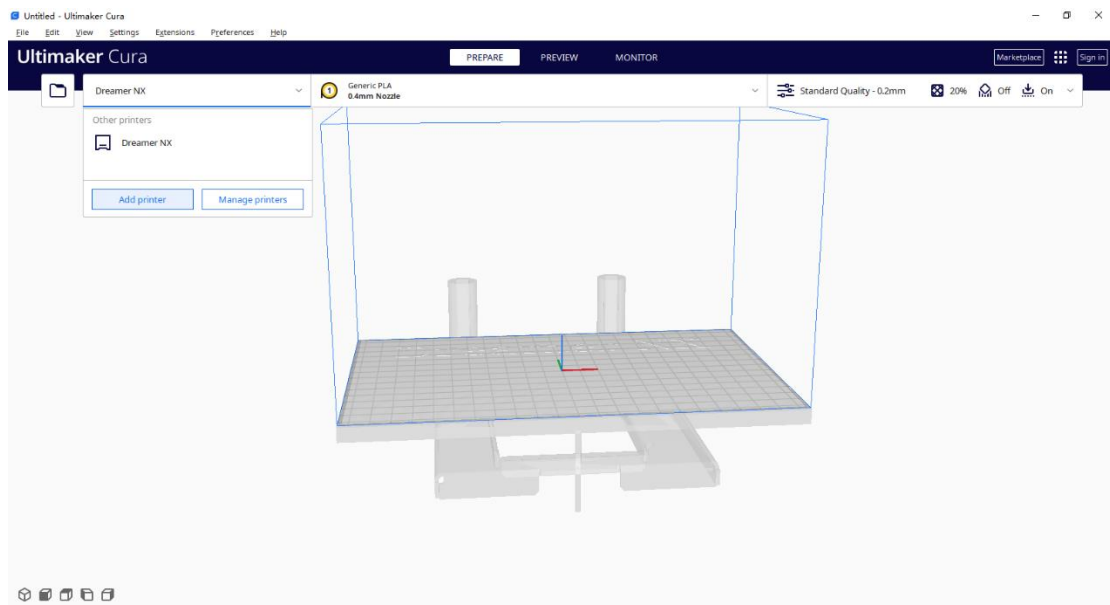
Step 1: Launch Cura

Step 2: Add Your 3D Printer

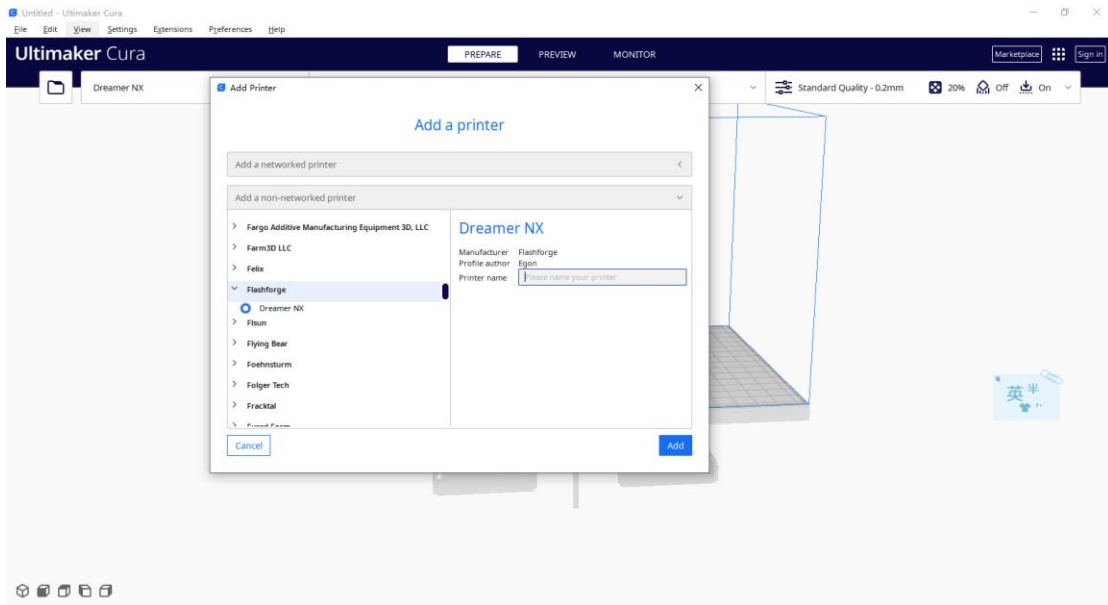
2.1 Option 1: Settings -> Printer -> Add Printer



Option 2: On the main interface, click the Printer section, then click *Add printer* in the dropdown.

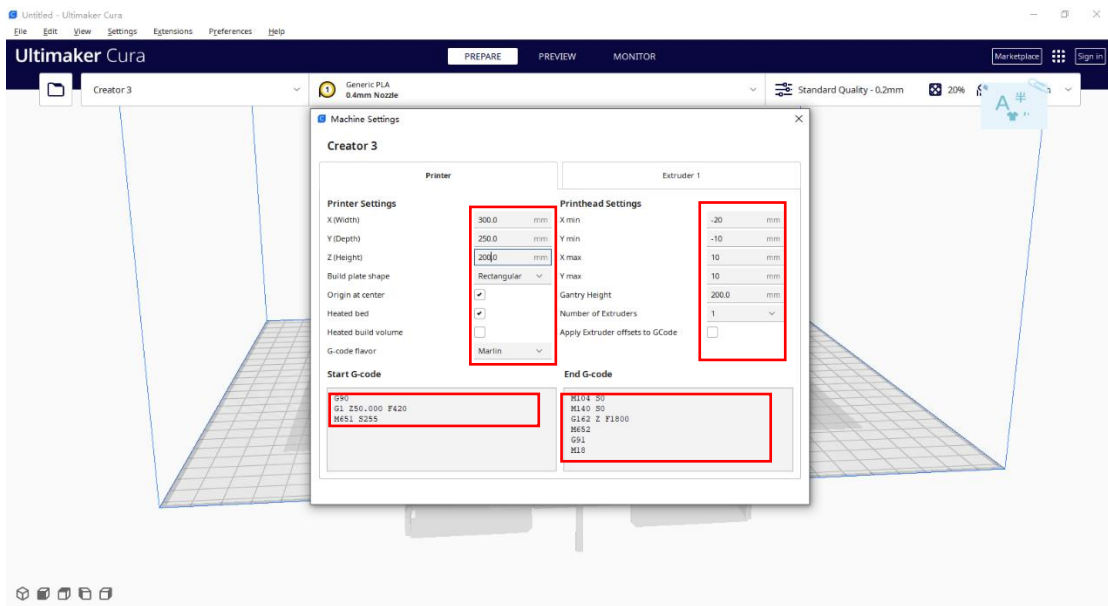


2.2 On the pop-up window, please find Flashforge in the left list, then fill in your Printer name on the right. Click *Add* to proceed.



2.3 On the Machine Settings window, kindly fill in the **Printer** spec as suggested in the red. Switch to **Extruder 1** to fill in the extruder setting as suggested in the red. Then click *Next* to proceed.

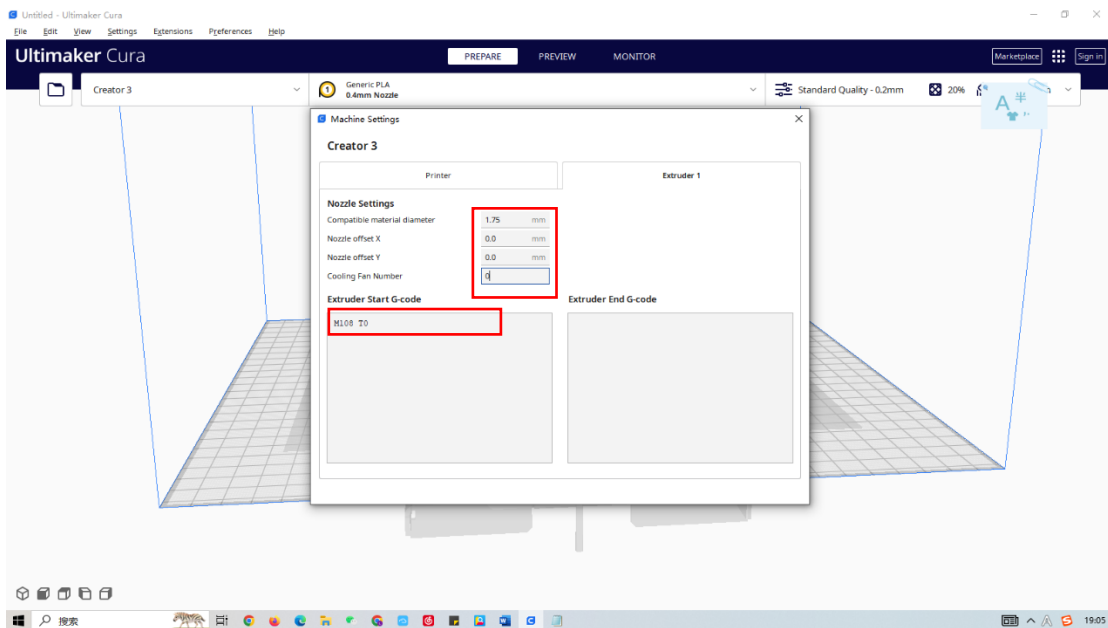
Note: Cura currently does not support slicing files for dual extruder printers. The **Extruder 1** on Machine Settings window refers to the right extruder.



Printer Setting	Printhead Setting
X (Width) 300.0	X min -20
Y (Depth) 250.0	Y min -10
Z (Height) 200.0	X max 10
Build plate shape: Rectangular	Y max 10
Original at Center ✓	Gantry Height 200.0

Heated bed ✓ G-code flavor Marlin	Number of Extruders 1
---	------------------------------

Start G-code: G90 G1 Z50.000 F420 M651 S255	End G-code: M104 S0 M140 S0 G162 Z F1800 M652 G91 M18
---	---

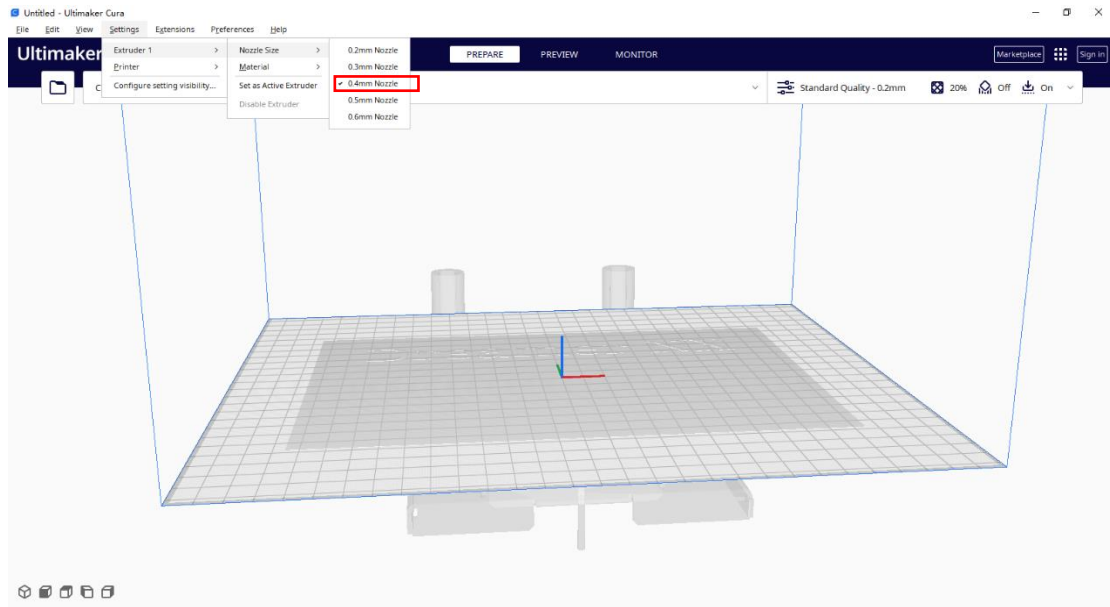


Nozzle Settings
Compatible material diameter 1.75
Nozzle offset X 0.0
Nozzle offset Y 0.0
Cooling fan number 0

Extruder Start G-code
M108 T0

Step 3: Select Nozzle Size

Once the printer is added, you may go to Settings -> Extruder 1 -> Nozzle Size to select your printer's nozzle size as 0.4mm.



Step 4: You may now create new project and start slicing

Hope this helps. If you need any assistance on slicing or printing, Flashforge's dedicated support team is only an email away. Feel free to drop us a line at support@flashforge.com .