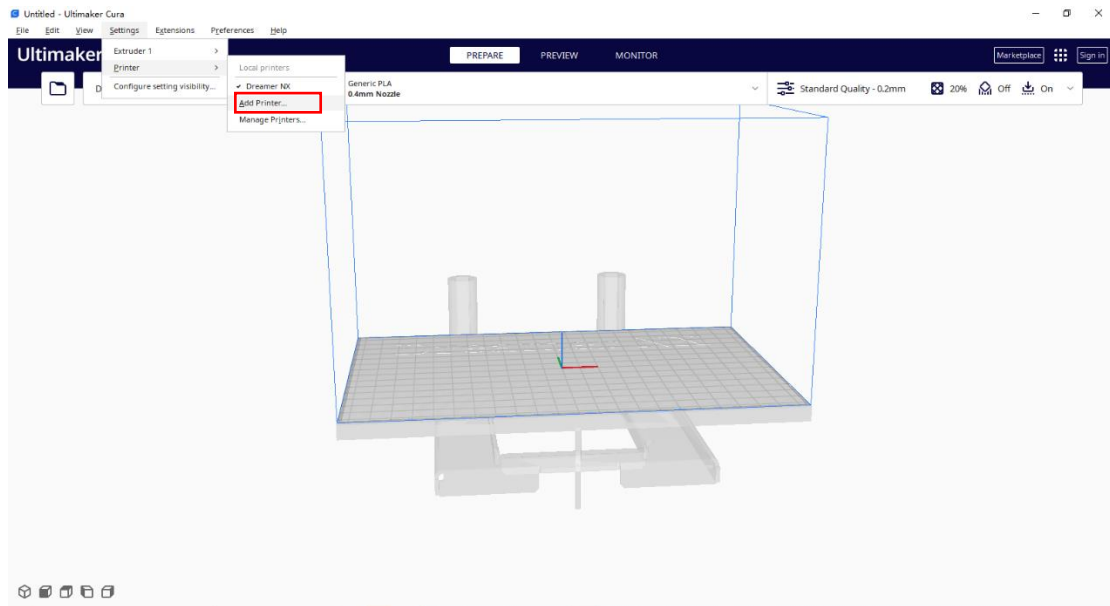# Cura Setup for Flashforge Dreamer
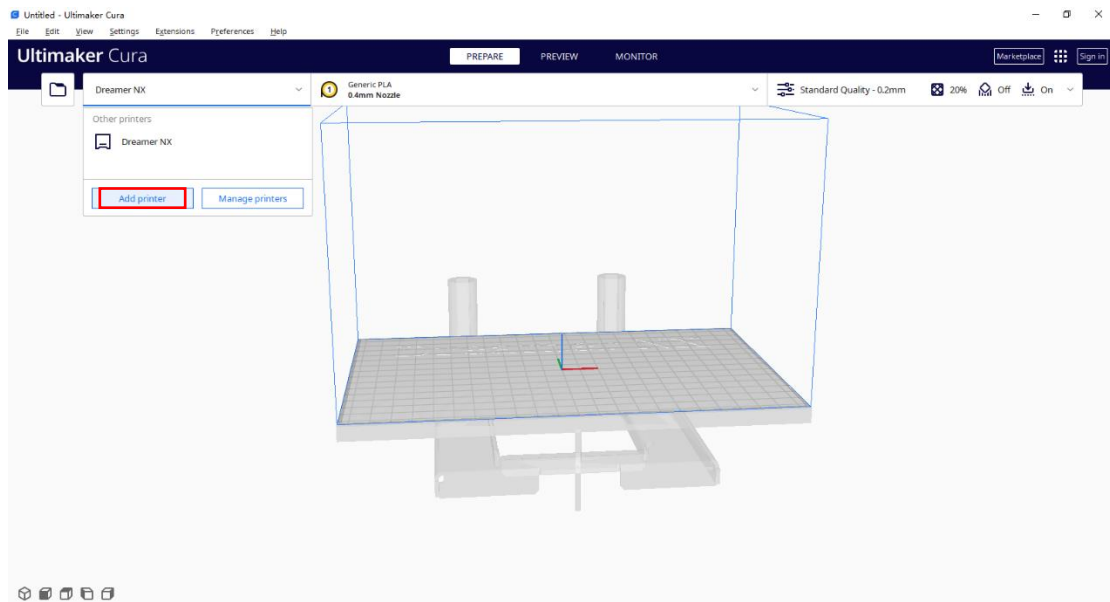
Step 1: Launch Cura
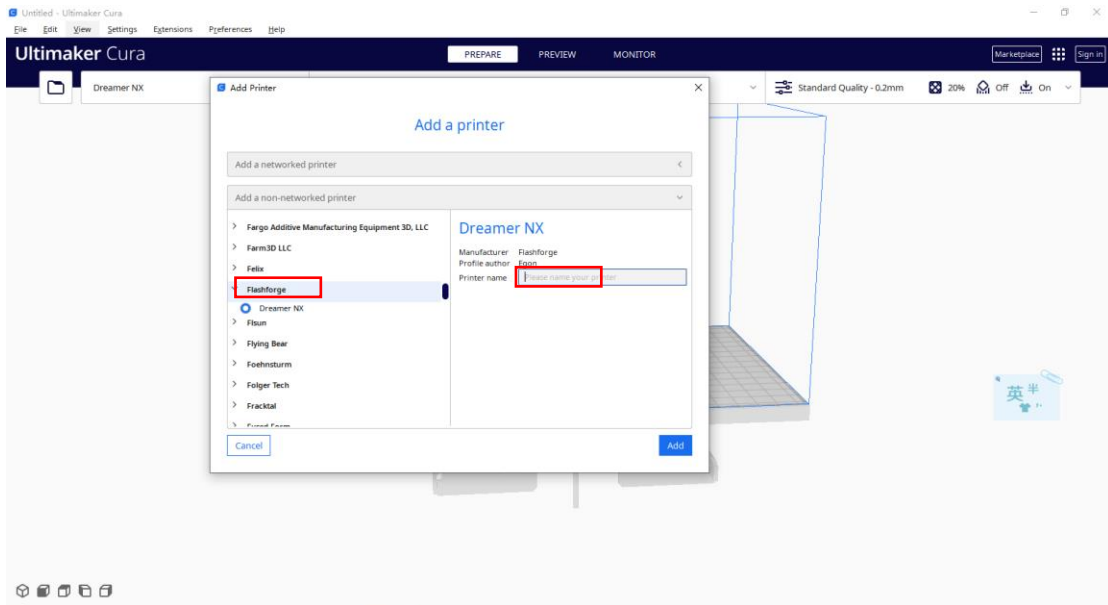
Step 2: Add Your 3D Printer

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 confirm the extruder setting as suggested in the red. Then click *Next* to proceed.
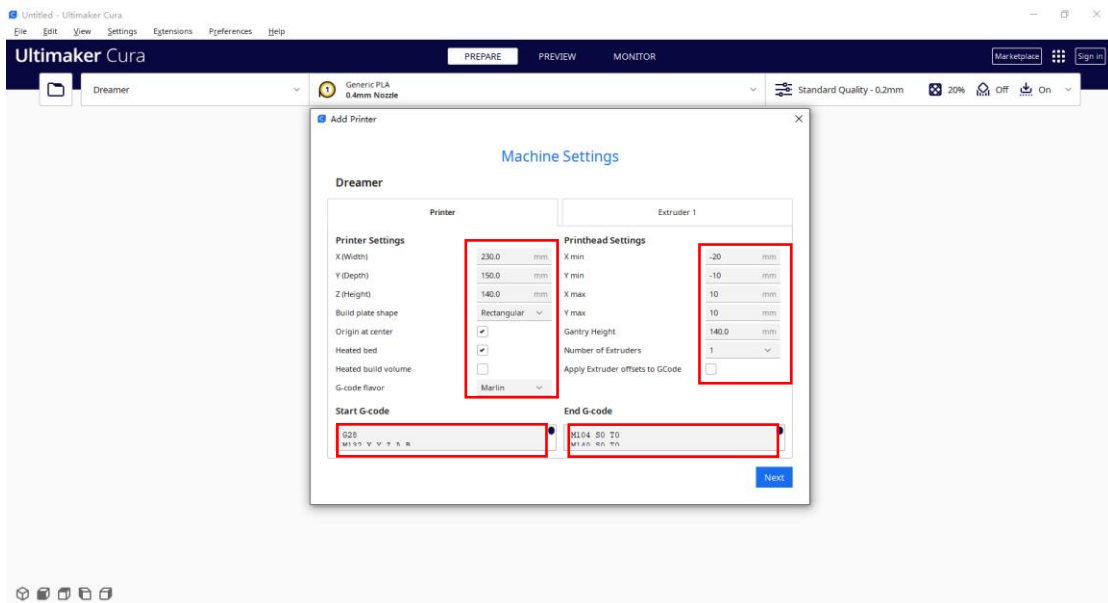
Printer Setting	Printhead Setting
X (Width) 230.0	X min -20
Y (Depth) 150.0	Y min -10
Z (Height) 140.0	X max 10
Build plate shape: Rectangular	Y max 10
Original at Center √	Gantry Height 140.0
Hotend bed √	Number of Extruders 1
G-code flavor Marlin	

Start G-code：	End G-code：

G28	M104 S0 T0
M132 X Y Z A B	M140 S0 T0
G1 Z50.000 F420	G162 Z F1800
G161 X Y F3300	G28 X Y
M7 T0	M132 X Y A B
M6 T0	M652
M651 S255	G91
M907 X100 Y100 Z40 A80 B20	M18
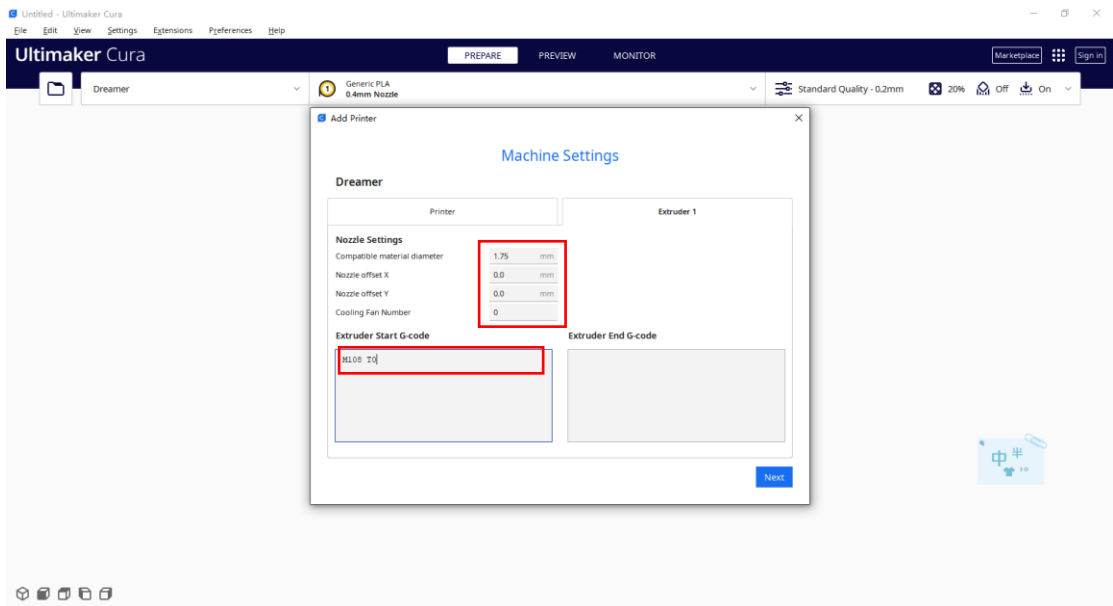
Nozzle Settings
Compatible material diameter 1.75
Nozzle offset X 0.0
Nozzle offset Y 0.0
Cooling fan number 0

Extruder Start G-code
M108 T0

Note: Cura sliced files are only available for printing with Dreamer's right extruder.

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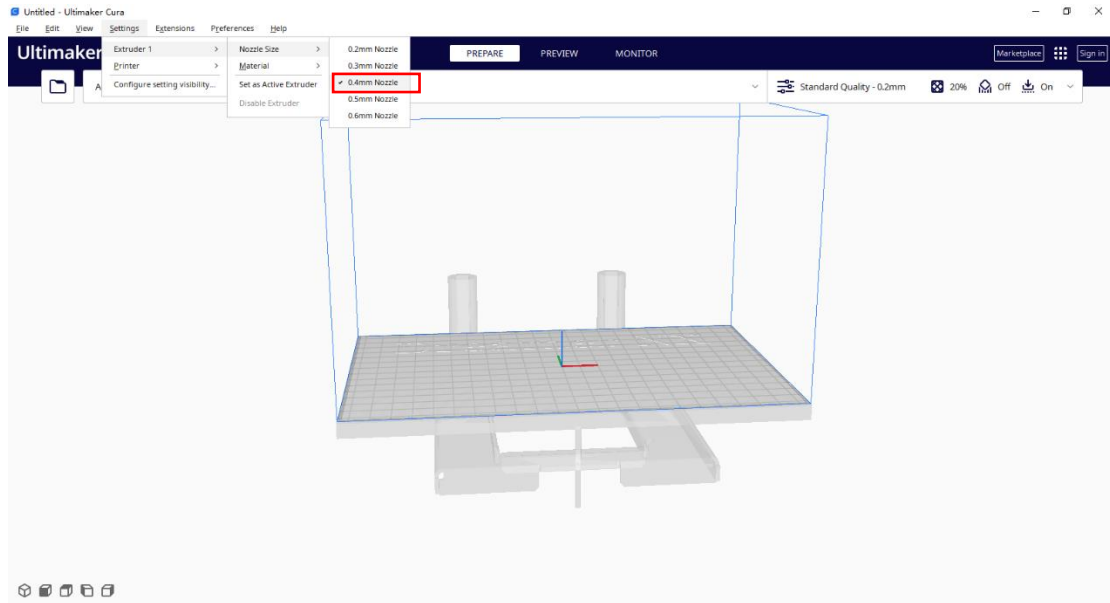
Step 4: You may now create new project and start slicing

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 .