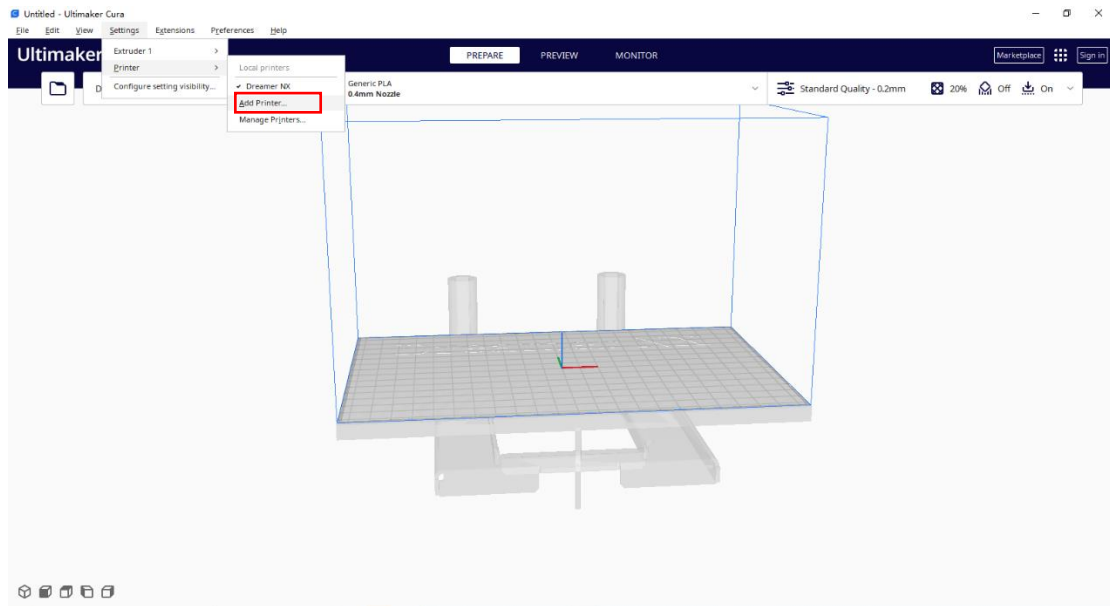# Cura Setup for Flashforge Creator Pro 2
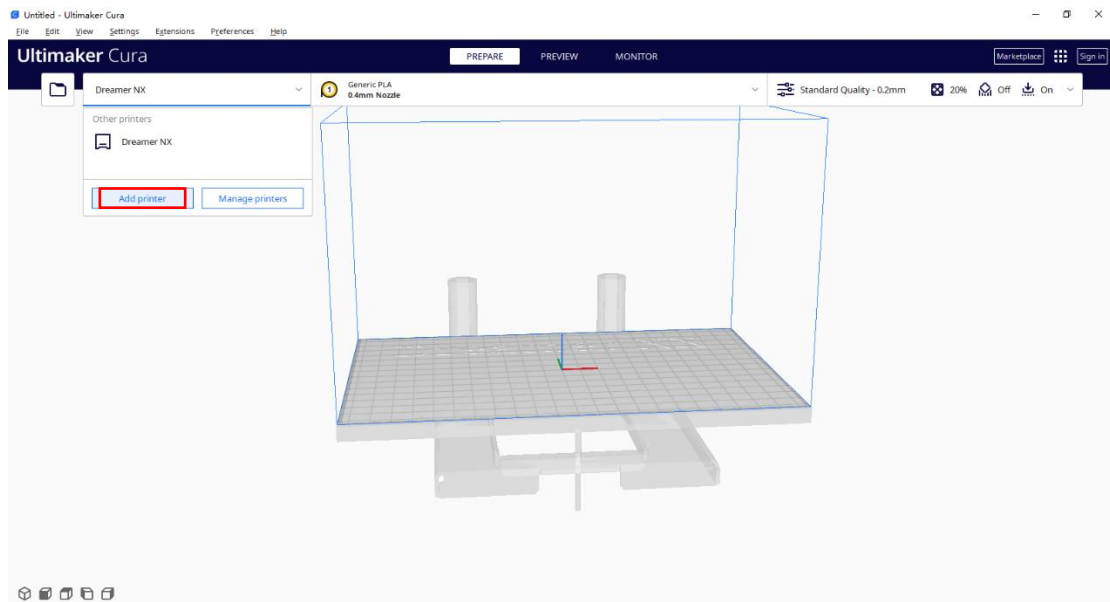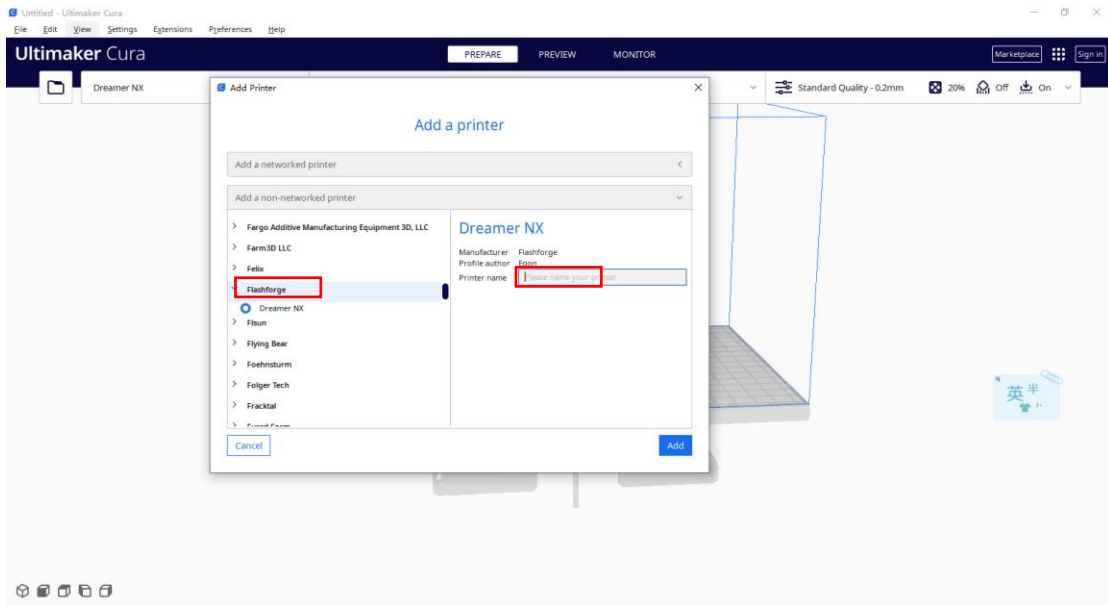
**Step 1: Launch Cura**

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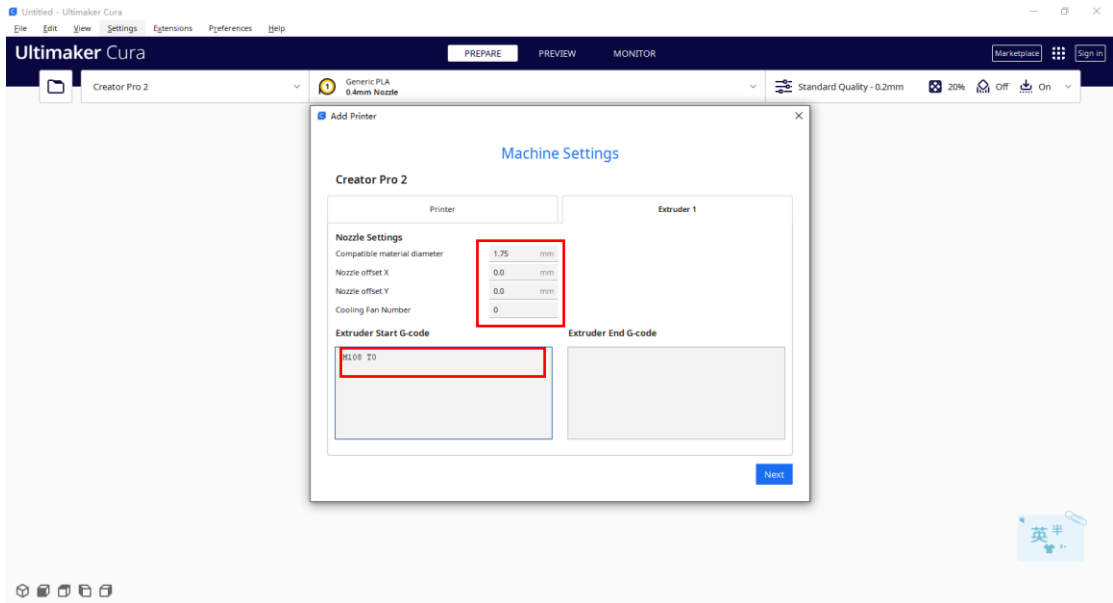
2.3 On the Machine Settings window, kindly fill in the **Printer** spec as suggested in the red. Switch to **Extruder 1** to confirm the extruder setting as suggested in the red. Then click *Next* to proceed.
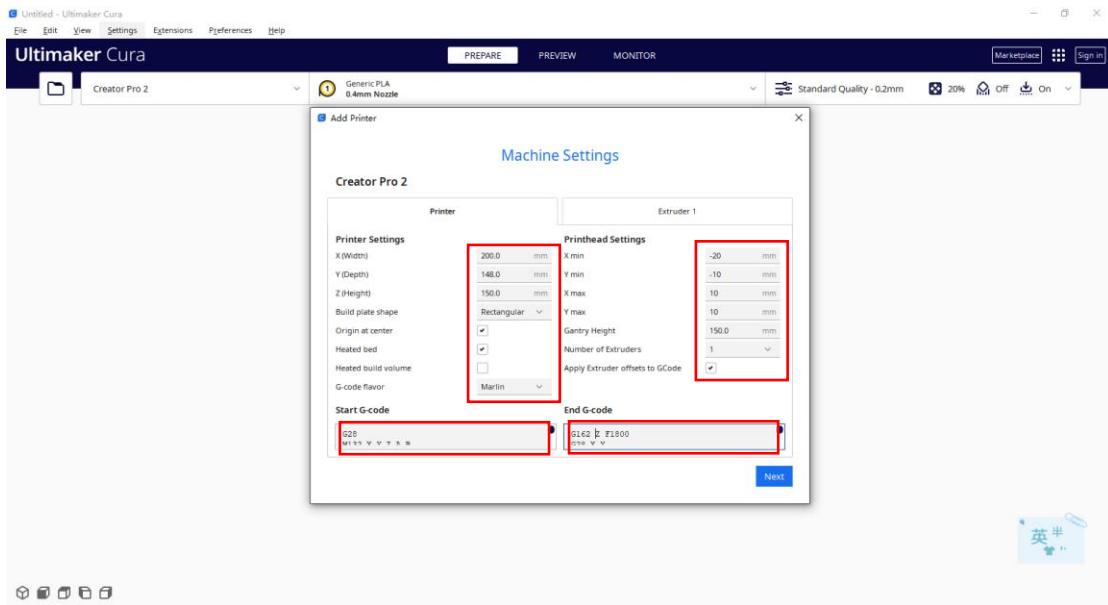


| Printer Setting | Printhead Setting |
|---|---|
| X (Width) 280.0 | X min -20 |
| Y (Depth) 148.0 | Y min -10 |
| Z (Height) 150.0 | X max 10 |
| Build plate shape: Rectangular | Y max 10 |
| Original at Center  √ | Gantry Height 150.0 |
| Hotend bed  √ | Number of Extruders 1 |
| G-code flavor Marlin | Apply Extruder offsets to GCode  √ |

| 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 |
| | 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 Creator Pro 2'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.
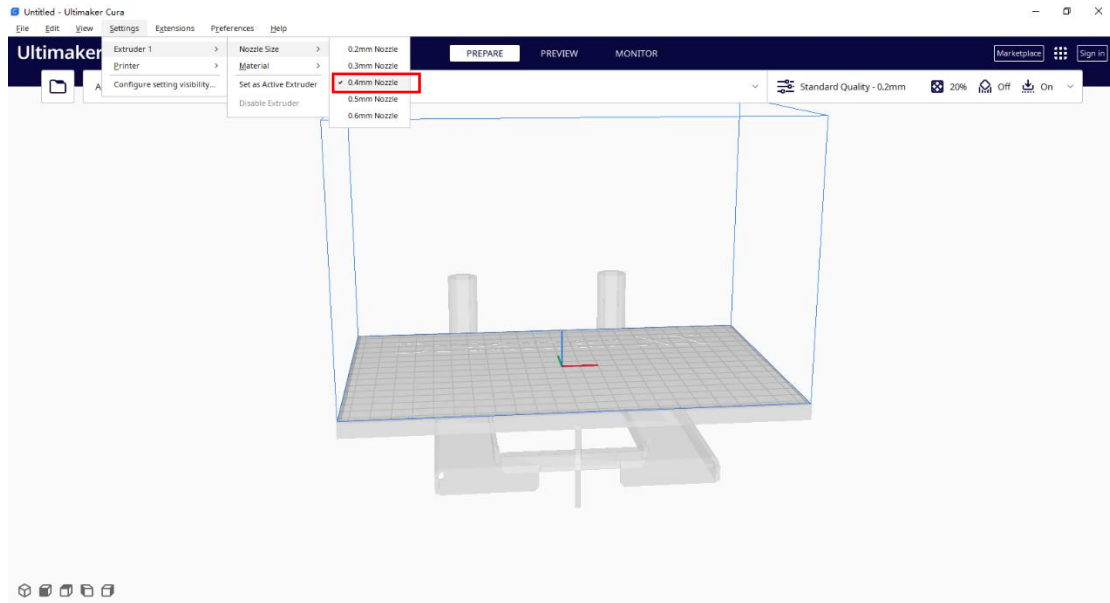
**Step 4: You may now create new project and start slicing**

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 .