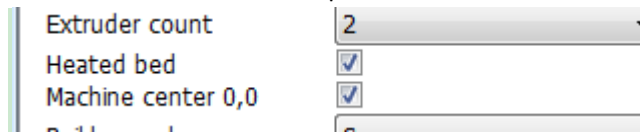


## Printing is not in the center of the bed

If you are using cura, the bed center 0,0 should be marked in machine settings as our manual said. You can also check the photo below.



If you are using repetier host , please set the printer configuration as our manual said. Then your prints will be in the center.

If you are using simplify 3d or cura, the printing is on the left side of the bed. This usually happens after either canceling a print and restarting, or after a print finished and start another extruder printing without powering off the machine.

In our repetier host slicing settings (Repetier host will not have this problem), we have the gcode below in start part (only for E1 printing):

**T0**

**M302**

**G92 E0**

**G28**

**G1 Y0 F300 E0**

**G92 E0**

This gcode function is to set the first extruder coordinate system, and the problem will be solved.

You can add this **below G28 in cura**, or **below G28 Z0 in simplify 3d** . **If your printing is for the second extruder E2, the gcode should be added as below (The above gcode is for E1):**

**T0**

**M302**

**G92 E0**

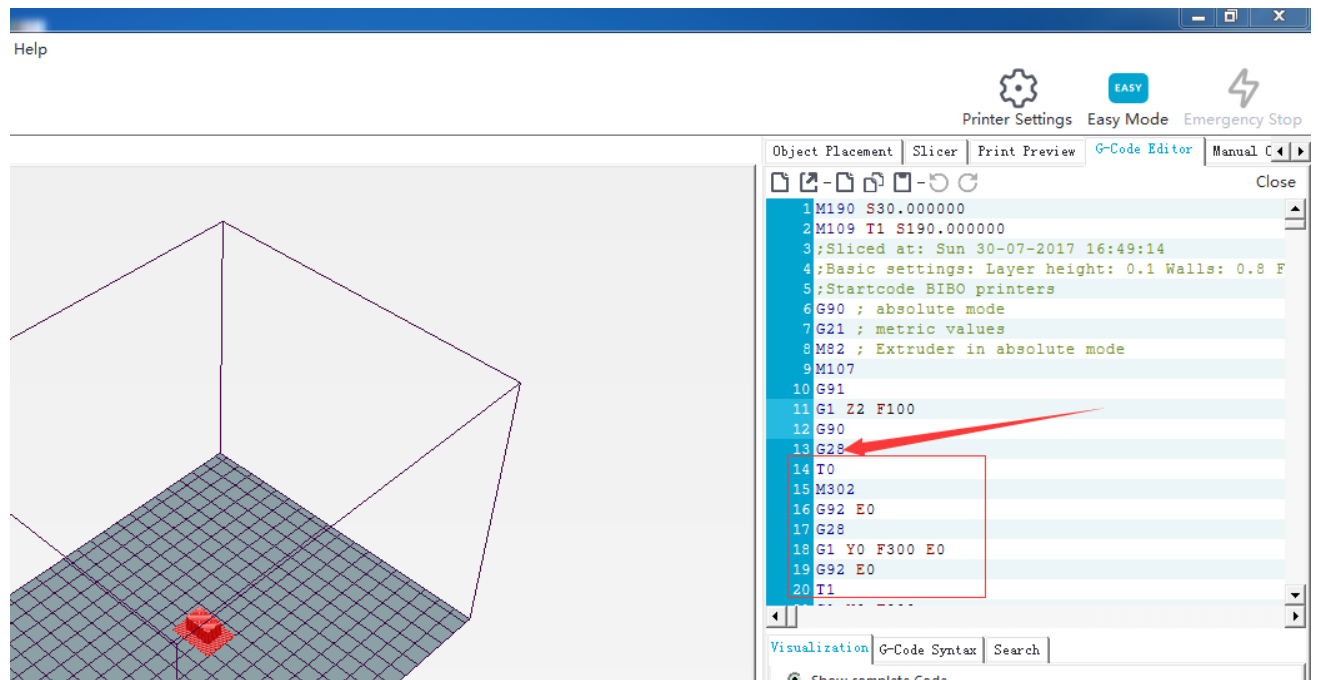
**G28**

**G1 Y0 F300 E0**

**G92 E0**

**T1**

For example, we add the above gcode for E2 printing which sliced by Cura and Simplify 3D . Please check the photo below:



You can use repetier host to open the gcode file and add the gcode as the photo above.

