

# Note on Gcodes + SD Card Printing

## What is Gcode?

Gcode are the actual 'print instructions' readable by 3D printers. For more info, you should read [3D Printing 101](#).

## Gcode Compatibility

### Short version

- Gcode **must have G28 and G29** at the beginning.
- Put the SD card into the controller and select the gcode file you want to print.
- If you use any of **gcodes supplied by IMADE3D, you need not worry**.

**Alert: Do NOT attempt to print anything before the Pre-Flight check, Z Probe height set up, and Nozzle fan height set up! You could damage your Jellybox.**

### Alert: gcode compatibility

- **Gcodes are printer-specific.** You cannot just put any gcode for any printer into Jellybox and expect good results. (You CAN put in gcode generated for another JellyBOX, but not a Printbot or Ultimaker gcode for example.)
- What's more, if you try to run a random gcode without inspection, *you could damage your Jellybox!*
- If you slice a model using **any IMADE3D slicing profile**, you're all set.
- If you use custom slicers and custom profiles, you **must include a "G28" and "G29"** code in the starting gcode settings of the slicer! Without this, Jellybox would not run the auto leveling procedure and could scar the bed. For more info, look at the guides on slicing and making custom profiles.
- If you know what you're doing, you can re-use many gcodes, but you may need to do a bit of text-editor magic first. If you don't know what that means... have someone show you (Filip may do a video on it.)

## SD Card Printing

### Short version

Put the SD card into the controller and select the gcode file you want to print. Done. :-)

### Long version

1. Put a (compatible) gcode file on an SD card
2. Insert the card into the reader on the left side of the LCD controller
3. Press the knob once, scroll all the way down and select
  - a. "Print from SD" -> your file
4. Done!
5. Jellybox should heat up to the printing temperature, perform auto bed leveling procedure, and start printing