

G-code Manual

General

Keep an eye on G-Code Console Tab. If you change extrusion or speed via slider, the command for it will appear in there. If you send a command with a long output, the response will be displayed on the screen for only a couple of seconds. It will, however, be logged in the console. Also, do not be afraid of Macros - to use it, make new ones and read existing ones to see exactly what's going on. While writing your own, do everyone a favor and comment each line on what it does! Everything after ";" in a line is considered a comment.

Everytime the BOD2 machine is turned on or after a reset, it loads the default parameters on start-up. The default configuration can always be found and / or edited in the "config.g" file on the printer.

"config.g" can be found here: **Duet Interface => Settings => System Editor**.

DO NOT MESS AROUND IN THERE! All experimental settings should be configured through the gcode console, as a reboot will recover any mistakes made.

Commit changes only if necessary, i.e.: if you just installed a printer and need to change print area size, or invert direction of one of the axes.

Gcode is not case sensitive. It can be a command on its own or needs to be inputted with an additional variable. I.e.: "M122" will return firmware diagnostics. But it can also be sent as "M122 P100" for summary of recent moves.

Informative G-code base specifically for Duet at: <https://duet3d.dozuki.com/Wiki/Gcode>

Setup

G90 ;Set to Absolute Positioning. All coordinates from now on are absolute relative to the origin of the machine.

G91 ;Set to Relative Positioning. All coordinates from now on are relative to the last position.

// Its good to know how to switch between the two. However COBOD is almost always using Absolute Positioning.

G92 ;Origo Related

// One of the commands we use the most - changing/setting machine origin. I.e.: "G92 Z0" will set current Z coordinate as new Origo (Z=0).

// Lets say we need to print on pallets, thus to set Z=0 we move nozzle down until we almost touch the pallet and call "G92 Z0"

// This command can be also used with multiple parameters at once: "G92 Z0 Y200 X255". Useful if adjustments needs to be made

// while trying to align while printing on top of previous day print.

G0/G1 ; Move Rapid(ignore Feed rate) / Move Controlled

// "Go to coordinate" in linear motion. I.e.: "G1 X2000 Y4200" will move from current coordinate to X=2000 and Y=4200.

// Before making the printer move anywhere, make sure there is nothing to collide with in its path.

Control/Adjustment

M220 Sn ; Speed override factor

M221 Sn ; Extrusion override factor

//Just 'M221' reports the current extrusion factor

// With Duet2 interface we can control print speed and extrusion values via slider interface. However, we are somewhat limited to

// the min/max values of those sliders (for now). If we have to adjust those values outside the slider given min/max, we use the commands above.

// I.e.: "M220 S350" will change current print speed to 350mm/s. Similarly, "M221 S115" will change current extrusion rate to value of "115"

// IMPORTANT! If changing value of extrusion, make sure you double check value before submitting - Extra digit will stop printer from moving

// and will empty half of the hopper on the spot, before you manage to hit "Pause".

M201 ;Set Max Acceleration

M203 ;Set Maximum Feed Rate

// Our Defaults:

M201 X300 Y300 Z100 E1500 U5000 ; Set accelerations (mm/s²)

M203 X15000 Y15000 Z2500 U7500 E50000 ; Set maximum speeds (mm/min)

// 1 Meter per Second edition:

M201 X400 Y400 Z100 E1500 U5000 ; Set accelerations (mm/s²)

M203 X50000 Y50000 Z2500 U7500 E50000 ; Set maximum speeds (mm/min)

Information

M408 Sn ; n=0,1,2,3,4,5

// Depending on "n" value, it will give all kinds of information related to current print (file name, speeds, extrusion, Layer, time, etc),

// machine parameters, firmware, etc. Try "M408 S3".

Zprobing:

The built in Z-Probe can be used to calibrate the printers Z height precisely to the ground. Read and understand the entire process before attempting to calibrate the Zprobe.

Setup of Zprobe:

BEFORE ATTEMPTING TO USE THE ZPROBE, MANUALLY MOVE THE PRINthead TOWARDS THE GROUND IN SMALL STEPS AND MAKE SURE THE ZPROBE VALUE IN THE WEB INTERFACE CHANGES TO 1000 BEFORE THE NOZZLE TOUCHES THE GROUND. PREFERABLY THERE IS STILL A 10CM BUFFER BETWEEN THE TIP OF THE NOZZLE AND THE GROUND WHEN THE VALUE SWITCHES

A) Calibrating the Z trigger height

1. Find a relatively leveled surface that spans from the most frontal part of the nozzle to behind the end of the Printhead. Make sure it is even with the rest of the printing area. Use a spirit level to ensure the surface evenness.
2. Disable the axis limits by sending `M564 S0`. THIS WILL ALLOW THE PRINTER TO MOVE OUTSIDE THE DEFINED PRINTER VOLUME. DO NOT MOVE IN LARGE STEPS. USE WITH CAUTION.
3. Move the nozzle towards the ground in small steps until the nozzle is about 1mm from the ground. Manually set this position as Z0 by sending `G92 Z0`
4. Enable the axis limits by sending `M564 S1`.
5. Move the Printhead manually upwards until the Z probe value in the Webinterface reads "1000".
6. Move 100mm further up.
7. Start a Zprobing move by sending `G30 S-1`. This will move the printhead towards the ground and stop moving when the Zprobe is triggered. Write down the reported Z height.
8. Repeat from step 6 two or three times and calculate the average reported Z height.
9. Save the measured Z trigger height to the G31 command in the *config.g* file:

```
...  
G31 X0 Y200 ZNNN  
...
```

B) Using the Z Probe

1. Zprobe at the current position
 - a. `G30`
2. Zprobe at a specific location
 - a. `G30 XNN YNN`

C) Setup of the complete bed compensation

1. Find the minimum and maximum X and Y coordinates of the area to be measured and subtract 200mm from the maximum Y coordinate.
2. Decide the probing interval. We recommend values between 500mm and 1000mm
3. Set the calibration are with `M557 XMIN:MAX YMIN:MAX SINTERVALL`
4. Alternative set the calibration with `M557 XMIN:MAX YMIN:MAX PXX:YY` (where XX:YY represents the total number of reference points in each axis)
5. The `M376 HNN` command is used to taper off bed compensation over a specified height. We recommend a H value of 500.

D) Use the complete bed compensation

The Printer has to be homed in all axes before using the bed compensation

1. Clear the previously loaded height map
 - a. `G29 S2`
2. Probe the bed in the area defined by M557 and save it with a specific
 - a. `G29 S1 P"FILENAME"` 
3. Load a previous bed compensation file (visible in the *system files* tab)
 - a. `G29 S3 P"FILENAME"`

E) Setup of the advanced bed compensation

It is possible to probe the bed on a specified set of X and Y coordinates. This is especially helpful when printing on an odd-shaped foundation.

1. Collect a list of points to be probed. To do so, manually move the printhead to various positions along the brim of the foundation as well as the center area and write down the X and Y coordinates. Keep in mind that the Zprobe is offset from the actual nozzle.
We recommend a grid spacing of 500mm to 1000mm
2. Find the *bed.g* file in the *system files* tab and open it.
3. Delete all contents of the *bed.g* file
4. Begin the file with the following commands:

```
M561 ; clear any bed transformations
G28 ; home the printer
```
5. Insert the previously collected probing points into individual `G30` commands. Increase the *P* parameter for each new point. Keep the `Z-99999` value, as it indicated the firmware a probing move. Any other value for this will be interpreted as a manually set Z offset at this point.

```
G30 P0 XNNN YNNN Z-99999
G30 P1 XNNN YNNN Z-99999
...
G30 P23 XNNN YNNN Z-99999
G30 P24 XNNN YNNN Z-99999
```

F) Use the advanced bed compensation

1. To use the previously created custom calibration set, call a `G32 S3` command