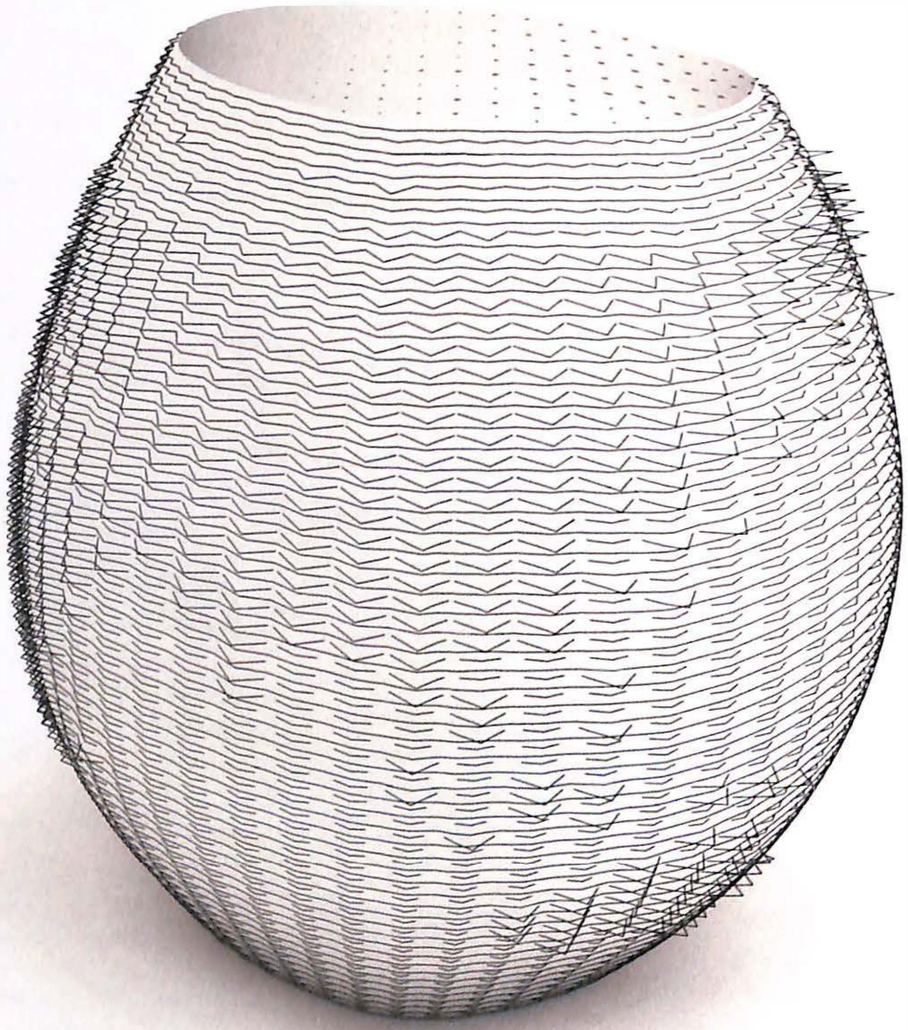
*.3dm & *.gh *files*



PART IV
Drawing 3D printing

The first step to control the detailed movement of a 3D printer is getting the curves for the paths. Then, the curves will transform into a g-code that can be understood by the machine.

The aim of this chapter is to help Grasshopper® users to control the g-code for a 3D printer directly from Grasshopper®, without leaving its environment and without plug-ins. There are some amazing plug-ins available, but their use could constrain the results and also become obsolete or useless over time if developers do not maintain them up continuously. Controlling the code opens a huge amount of possibilities for 3D printing as well as design possibilities.

No knowledge of coding language is necessary but expert users that have knowledge of Python, C#, C++, VisualBasic etc. can use this to maximise their model's possibilities and their Grasshopper® definitions.

According to hardware, 3D printer manufacturers worldwide are continuously bringing new models to the market. We will focus on WASP technology, specifically on a WASP DELTA 2040, a 3 axes delta 3D printer. The customization possibilities that this type of machine offer depend mainly on the path, the speed and the flow.

The **path** is the curve that must be followed by the 3D printer's nozzle. It is the base for any design. You can find some paradigmatic study cases in Part V and their explanation in Grasshopper®.

The **feed rate** (F) or **speed**. It can be a constant or a variable along the design, producing different results. Changes in the speed could create quite impressive effects when printing translucent clay.

The **extrusion rate** (E) or **flow** or amount of extruded material could be proportional for the distances between points or not. A controlled overflow of material can create interesting textures. Extrusion is calculated by the proportion between the material that enters in the extruder and the material that exists according to the nozzle diameter.

G-Code

The g-code is a text file that includes the orders that tell the machine how to move and the extruder how much to extrude, as the combination of movement and extrusion is fundamental for 3D printing.

It is the most extensive language used by C.N.C. automated machines as milling machines or 3D printers. It is an alphanumeric language that contains orders made of the combination of a letter and a number (e.g. G28). This code contains orders with all the letters of the English alphabet (A-Z) but it has evolved so that the letter G is chosen to describe the controlled movements of the machine. So, orders with 'G' as G0 or G1 will be the most common in this type of code. That is why it is known as g-code.

G-code can describe hundreds of orders for a machine. But 3D printers do not use all the possible orders as some of them have been developed to work with other type of machines. E.g. G97 allows to control the spindle speed on a milling machine which is unnecessary for a 3D printer.

The g-code will be read by the machine's firmware. Firmware is a software installed on the memory chip of the 3d printer. The most common firmwares are Marlin, Repetier, Reprap, Prusa, etc. As some machines use variations of these, the first thing to establish is to find out what firmware our machine uses as they do not read the g-code in the same way. There are some orders that make no effect on some firmware and have to be changed by another.

As our machine uses Marlin as firmware, the reader can find below a list with the orders you need to know to take control over the g-code of a WASP 3D printer. Also google 'Marlin g-code' and the 'Reprap project' to find a complete list of g-code commands as well as a bounty of useful information for different firmware.

G-code common commands list:

; Text after semicolon will make no effect on the code. Very useful for explanations and comments.

E Extrusion amount in mm (E10 means 'extrude 10mm').

F Feed rate in mm/m (F1800 means speed of 1800mm per 1 minute).

G0 Rapid positioning. G0 must be followed by one or more or the following: an E value, F value or X,Y or Z value (G0 F3000 X10, means move 10mm in X axis at 3000mm/m). In case of no F value, the movement will be done at the max speed.

G1 Linear movement. G1 must be followed by one or more or the following: an E value, F value or X,Y or Z value (G1 F1800 X10, means move 10mm in X axis at 1800mm/m). G1 needs a F value. Both G0 and G1 describe a movement. In fact they could be used in the same way. Depending on the firmware installed in the 3D printer, there could be slight differences. Usually G0 is used to describe a movement in the most efficient way according to speed for travels, and G1 is a movement at a certain defined speed for extrusion. Also, this could be used just to differentiate a movement with extrusion from another without it. Always test beforehand for critical jobs.

G4 Wait or dwell. It must be followed by the time we want the machine to stop. It can be described by P and the amount of milliseconds or S and the number of seconds. G4 P500 means wait for 500 milliseconds, so half a second. G4 S0.5 means wait for half a second too. Parameters such as extruder temperature or fan speed should remain the same.

G20 Set units to Inches.

G21 Set units to Millimetres (one of the two G20 or G21 has to be chosen, but not both).

G28 Go home to all axes if none are given, or move home to only the specified axes. (G28 will move all axes home, G28 X Y only X and Y axis).

G90 Absolute coordinates. After G90 XYZ coordinates are absolute to the origin of the machine. This is usually the firmware's default.

G91 Relative coordinates. After G91 XYZ coordinates are relative to the last position. This means that the last coordinate is the new zero for the next one.

M82 Set extruder to absolute coordinates. Equivalent to G90, but for the extruder.

M83 Set extruder to relative coordinates. Equivalent to G91, but for the extruder.

G92 Overwrite the position of an axis to zero or to the specified value. No physical motion will occur. (G92 E will reset E to zero. G92 X50 will reset the current machine's X coordinate to 50. If no values are specified, all axes will be set to zero).

If you work with thermoplastic material, some extra orders need to be controlled:

M104 Start heating the extruder. The temperature is one of the most important parameters when fusing thermoplastic material. It has to be followed by the letter S and the temperature in degrees (M104 S210 sets the extruder temperature to 210°Celsius).

M106 Set fan ON. It could be followed by S and a parameter from 0 (stop) to 255 (max. power).

M107 Set fan OFF. You can also use M106 S0 instead, but it depends on the firmware availability.

M109 Wait until the extruder is heated to run other commands (M109 S210 means wait till extruder temperature is 210°).

M117 It is used to display a message in the LCD screen (M117 *Waiting*).

M140 Heat the build plate. Followed by temperature. (M140 S60 heat build plate to 60°Celsius).

M190 Heat build plate and wait until the build plate is heated to run other commands (M190 S60 means wait till build plate's temperature is 60°Celsius).

Many other orders could be found in a g-code file and depending on the 3D printer's firmware, they could differ from one to another. For further interest, the reader can find more information on the RepRap project's wiki, by looking up 'g-code'.

The g-code for a 3 axis 3D printer is made of three parts:

Start protocol (get ready)

Core with instructions (print our design)

End protocol (finish)

The start and the end protocols are a set of g-code routines or instructions for the machine to get ready and to finish printing. They include information as 'go home in all axis' or, when finished, 'go back to origin'.

Each machine has a protocol. We will focus on the Delta WASP 2040 3D printer protocols, but many thermoplastics 3 axis machines have similar instructions. If you own a 3d printer and you are not so sure about your machine protocol, you can just open an already made *.gcode extension file in notepad app (for Windows) or textedit in Mac and copy the start and end protocols of your printer.

Find an example of the start and end g-code protocols for Delta WASP 2040 as well as an explanation of the 'core' on the right-hand page:

Start protocol

Start g-code instructions explanation:

; -- START GCODE --	Text after semicolon makes no effect on the code
M82	Set extruder to absolute coordinates
G28	Home all axes
G92 E0.0000	Overwrite extruder position to zero
G1 E-4.0000 F6000	Retract the extruder -4mm at a speed of 6000mm/m
; -- end of START GCODE –	

Core with instructions

The core of the g-code is made of a sequence of movements describing our design. The main order for a controlled movement will have this structure:

G1 F1800 **X**10 **Y**10 **E**5

G1 is the order in g-code to describe a controlled movement.

F is the feed rate or speed in mm/minute. If it is constant and it does not change, this parameter is only necessary at the first line of the code.

X and **Y** are the coordinates to move to.

E is the length in mm for the extruded material.

End protocol

End g-code instructions explanation:

; -- END GCODE --	Text after semicolon makes no effect on the code
G92 E0.0000	Overwrite extruder position to zero
G1 E-4.0000 F6000	Retract the extruder -4mm at a speed of 6000mm/m
G28	Home all axes
; -- end of END GCODE –	

Examples starting at 0,0,0 coordinate:

Move 10mm in X direction from 0,0,0 extruding 1mm

G1 X10 Y0 E1

Move 10mm in Y direction from 0,0,0 extruding 1mm

G1 X0 Y10 E1

When 3D printing by layers, all the points of the same layer have the same Z value. This can be repetitive and could increase the g-code length, adding the Z component to every single line of code. Software that creates g-code by layers, changes the Z coordinate by stopping the movement, moving to the next level and working in the next Z value again only with X,Y coordinates. To stop and dwell, the order G0 is commonly used:

G1 F1800 X10 Y0 E10

G1 X10 Y10 E20

G1 X0 Y10 E30

G1 X0 Y0 E40

G0 X0 Y0

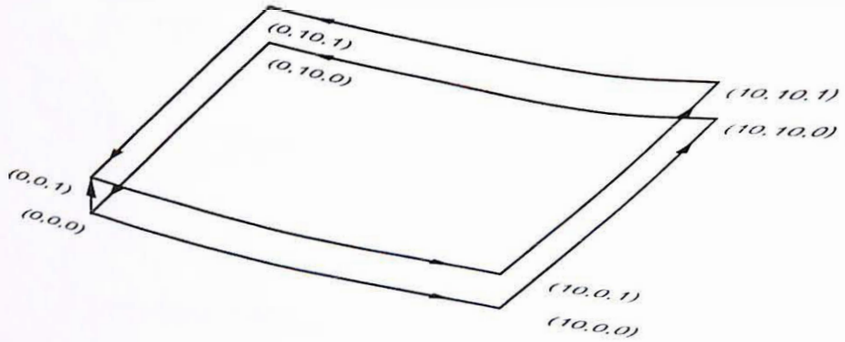
G0 X0 Y0 Z1

G1 X10 Y0 E50

G1 X10 Y10 E60

G1 X0 Y10 E70

G1 X0 Y0 E80



This is the core's g-code to draw two layers of a square of 10 mm edge length, at a constant speed of 1800 mm/minute and a constant flow of 1mm. of material per mm. of length. The layer height is 1mm.

Now, to check out if you got it, try to make your own g-code by hand to draw a square of 20 mm. side. If you own a 3D printer, use a g-code file as reference for the Start and End protocols. In the next chapter we will learn how to launch it.

; -- START GCODE –

.

.

.

.

; -- end of START GCODE –

.

.

.

.

.

.

.

.

; -- END GCODE –

.

.

.

.

; -- end of END GCODE –