**GCODE Primer for Laser Cutter Control**

*General Information*

Gcode is a very simple code designed to control 3D printers. We have an adapted version of a GCODE interpreter courtesy of Tim Schmidt from the Lansing Makers Network (<https://github.com/lansing-makers-network/buildlog-lasercutter-marlin>) that has been augmented to better control laser based machines like our laser cutter.

The flow of Gcode is fairly simple. Commands are given line by line with appropriate arguments. For example, a typical move command would look like this:

“G0 X1 Y2 Z3 F1000”

The initial command G0 specifies the function G0 to be applied. The following arguments are comprised of an initial letter to describe the type of argument, and a numerical value to quantify the argument. For example, the letters X, Y, and Z denote coordinates in the X, Y, and Z directions respectively. The numbers following X, Y, and Z show the coordinate value to move towards. The F value is a different argument, which specifies the movement speed. So 1000 quantifies how fast to move for this variable.

The Gcode RepRap wiki (<http://reprap.org/wiki/G-code>) gives a fairly comprehensive list of Gcode commands and appropriate descriptions. There are different Gcode interpreters including our interpreter (Marlin).

Interpreter firmware/Gcode syntax has been extensively used by the RepRap community to simplify the ability to control 3D printing movement. It is important to note though, that Gcode simply encodes for firing of electrical pulses in a predefined manner. So, even though Gcode was not designed for controlling a laser cutter, commands can be co-opted for unorthodox uses if the controller board is correctly wired to the laser cutter.

Below is a list of Gcode commands specific to our laser cutter/Marlin firmware setup with accompanying arguments and descriptions.

*Important Commands for Controlling our Firmware*

* + M649
    - Set Laser Fire Settings
    - Arguments
      * S
        + Laser Intensity
        + Values are 0 to 100 (float type)
        + Units are % laser power
      * L
        + Duration of laser pulse
        + Values are integers
        + Units are microseconds (μs)
      * P
        + Pulses per millimeter
        + Values are float type
        + Resolution discussed below
      * B
        + “Beam type”
        + 0 for continuous fire
        + 1 for Pulsed mode (note: specifying “P” should automatically change to B1 mode)
        + 2 for raster mode
      * D
        + Diagnostics
        + Logical 0 or 1 for whether to output diagnostics
  + M3
    - Turn laser on with current laser settings
    - Laser settings can be specified with an M3 command, but if a setting is not specified, the previously used value will be used.
  + M5
    - \*\*\*\*\*Turn laser off
  + G0
    - Move without laser on
    - Behaves identically with old marlin Gcode
  + G1
    - Move with laser on
    - Uses previously stored laser settings
    - Can take arguments from M649 to customize laser settings
  + M114
    - Obtain current coordinates
    - Enter directly to pronterface
    - Can be input while printing without disrupting the current print
  + G92
    - Set Coordinates
    - Takes X, Y, and Z arguments
  + G90
    - Set absolute coordinates
    - G0/G1 commands with X, Y, and Z will specify absolute coordinate locations
    - i.e. if you enter G0 X1 Y2 Z3 twice, the first command will move to (1,2,3) and the second command will not move (because you are already at the specified locations)
  + G91
    - Set relative coordinates
    - G0/G1 commands with X,Y, and Z will specify a relative movement direction
    - i.e. G0 X1 Y2 Z3 twice will move in a <1,2,3> vector direction twice to a final destination of (2, 4, 6) relative to the initial position
  + M84
    - Turn motors off