

Understanding Gcode Commands

as used for
Image Engraving

February 2015

John Champlain and Jeff Woodcock

Introduction

Reading and understanding gcodes is helpful for trouble-shooting cnc engraving processes, whether using rotary spindles, lasers, or other engraving methods. We hope the following helps new users to use their machines in a more effective manner.

This paper will use laser engraving as its example, although it will also be applicable to rotary spindle engraving. It assumes the reader has an operational cnc laser machine and is familiar with its controls.

Please note: this paper is not intended to be a complete tutorial on gcode and all of its nuances you may encounter in cnc operations. It is limited to those gcode commands used for simple raster engraving usage.

Image Engraving

Engraving shaded images is usually done with a method called **RASTER**¹ engraving, although some simple line drawings might be more easily engraved with a **VECTOR**² engraving method. The gcode commands used for raster engraving are simpler to read and understand, and fewer in number, than the gcode commands used for vector engraving, although some commands are common to both engraving methods.

Understanding Gcode Commands

Gcode programs are simple text files, easily loaded into any popular text editor on a computer. Programs are made up of a series of lines and are read by a cnc controller program (ex: GRBL, Mach3, Linuxcnc, etc.) in sequence from the top down. For examples used in this paper, Microsoft's Notepad editor is used.

Gcode Program Structure

Each gcode line can be either executable (readable) or non-executable (non-readable). Executable lines are those that the controller program will read and execute, while non-executable lines are ignored by the controller and are used for documentation and descriptive purposes. Non-executable lines always begin with a special separator character to tell the controller to skip over them and proceed to the next line of the file.

-
- 1 **Raster** engraving is a method whereby a laser is moved back and forth (scanned) across the material being engraved as the laser burns each pixel (point) on the line with its power being modulated proportionally to the shade of the pixel. The laser is incremented to a new line at the end of each scanned line.
 - 2 **Vector** engraving is a method whereby a laser will follow a continuous line drawn in an image from the beginning of the line to its end as the laser burns the material at a fixed power level. Vector engraving allows a laser to follow curves and change directions similar to drawing with a pen on paper, but it is very limited in reproducing changing shades of an image.

Examples of the most common separator characters used are:

% comment next,
and
(comment here).

Separator characters may also be included at the end of an executable line for documentation, and all following characters on the line will be ignored by the controller.

Gcode lines may or may not be numbered, depending upon the CAD/CAM program that generated the gcode program. When lines are numbered, the first character will be a "N" followed by a number. The number will usually be incremented at each line by 10, but may also be incremented by any other amount as well. The use of line numbering is optional, and is usually only used to make the gcode more easily readable, although it can be used by some gcode programs for various control reasons.

Gcode Program Example:

N10 (this comment line will be ignored)
N20 (this comment line will be ignored, but the next line will be executed)
N20 G01 X1.0 Y1.0 Z-0.5
N30 M30 (this command line will be executed, but all after the "(" will be ignored)

Gcode Commands

This is a list of the gcode commands and character letters most commonly used in gcode programs for raster engraving images. We are not including other gcode commands that are used to set up specific machine operating parameters and which are not normally included within a gcode program.

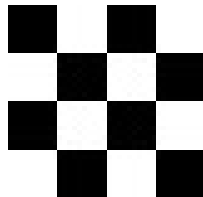
G00, move an axis at maximum preset feed rate from one point to another. (modal³)
G01, move an axis at a preset feed rate from one point to another. (modal)
G28, last command in a gcode program – sends X/Y axes to machine Home position.
G20, sets machine to run in Imperial mode (inches). (modal)
G21, sets machine to run in Metric mode (mm). (modal)
F, set a feed rate for moves. (modal)
M03, spindle/laser on. (modal)
M05, spindle/laser off. (modal)
M30, End gcode program and rewind to program beginning (may not be valid with GRBL).

N, program line number
S, spindle speed rpm/laser power level. (modal)
X, right/left axis
Y, forward/backward axis.
Z, up/down axis.
A, usually denotes a rotary axis, but can also be a linear axis slaved to the Y axis.
B / C, these can be either rotational or linear axis designators.

³ Please note that some commands are marked as modal, i.e. they remain in effect until changed by another command. Other commands only are active at the time they are being executed.

Reading a Gcode Program

Space does not permit the inclusion of a complete gcode program for raster engraving an image. Typical programs can have from hundreds of thousands to million of lines. So for this section we will use only the beginning and ending of a gcode program for engraving a small image. We have used our PicLaser Lite program to generate the gcode for a test image comprised of 16 alternating black and white squares, each of which is 10 pixels square in size. The total image size is 40 pixels by 40 pixels. A pixel is another unitless value, and must be assigned a size in PicLaser Lite in the Pixel Resolution setting.



Test Image

We will assume we are using a laser diode for engraving, controlled by PWM signals, and moving horizontally back and forth across the image, beginning with the X0/Y0 position located at the bottom left of the image.

Line numbers have been added manually to assist in explaining what each line of the gcode program does. Line numbers are not typically included in raster engraving gcode programs in order to minimize the total file size loaded into a computer.

We will also use Imperial units for this explanation, because that is the unit system which we are more comfortable working with. Gcode commands themselves actually use unitless values, and are only assigned a unit value by the inclusion of a G20 or G21 at the beginning of the gcode program.

PicLaser Lite - Ver.1.1.03 - Registered to: JOHN CHAMPLAIN

Select File Load File Hide Settings End Program Help

Original Image

| | |
|--------------------|-----------------|
| File Name | Test1.bmp |
| File Size | 2.6 KB |
| File Width | 40 Pixels |
| File Height | 40 Pixels |
| Horiz. Resolution | 100 Pixels/Inch |
| Vert. Resolution | 100 Pixels/Inch |
| Image Width | 0.400 inch |
| Image Height | 0.400 inch |
| Total Image Pixels | 1,600 |

Engraved Image

| | |
|----------------------|-----------------------|
| Width - inch | .400 |
| Height - inch | .400 |
| Num. Gcode Lines | |
| Gcode File Size - KB | |
| Gcode File Directory | C:\Users\John\Desktop |

Click Labels or Values to Change

| | | | |
|--|---|---------------------------------|---------------------------------|
| Feed Rate | 050 | inch/min. | <input type="checkbox"/> Metric |
| Pixel Resolution | 0.0100 | inch | Start Comments (|
| Max. Laser Value | 255.0000 | inch | End Comments) |
| Min. Laser Value | .0000 | inch | |
| Laser Off Command | M05 | | |
| Laser Control Command | S | | |
| <input type="checkbox"/> Left - 45° | | | |
| <input type="checkbox"/> Vertical | <input checked="" type="checkbox"/> Engrave Outline | No. Passes | 1 |
| <input checked="" type="checkbox"/> Horizontal | <input checked="" type="radio"/> GRBL | <input type="radio"/> Std_Gcode | |
| File Extension | .nc | | |
| File Directory | C:\Users\John\Desktop | | |
| Save Settings | Machine Setup Codes | | |

PicLaser Lite Settings Used

Simple Editing

Let's suppose you have a gcode program for raster engraving an image, and you want to test it with either a faster or slower feed rate, and not change any of the other commands or values. Rather than reload PicLaser Lite and perhaps have to reset all the settings to the same values as in the original gcode, you can easily and quickly make a simple change to the feed rate value with a text editor (Notepad for example).

- 1) Open the file with Notepad.
- 2) Locate the feed rate line you want to change (**hint: the line with Fx – where x is some number**).
- 3) Replace the F value with your new feed rate value.

Example:

```
( File: Test1.bmp )
( 02-02-2015 at 17:31:55 )
( PicLaser Lite ver.1.1.03 )
( GRBL Gcode )
( FeedRate = 050 ) <<< change this value to match the new feed rate set below to document your change.
( Pixel Resolution = 0.0100 )
( Max. Laser Value 255.0000 )
( Min. Laser Value .0000 )
( Engraving Angle = Horizontal )
```

```
G20
F50 <<< change the 50 to 60 to increase feed rate by 10ipm, and to 40 to decrease by 10ipm.
M05
G01 X0.00 Y0.00
```

- 4) Save the file and reload into your sender program and test.

Note: If you have more than one feed rate setting in your file, then it is likely that the last Fx just before the G01 will be the correct one to change. Some trial and error may be needed until you become more comfortable with gcode editing.

Comments

- 1) G00, G01, M03, etc. are often seen abbreviated in some gcode programs as G0, G1, and M3. There is no difference in their control actions whether abbreviated or not
- 2) Standard gcode programs for machines using a Z axis and rotary spindle frequently end with a M30 command. Some “flavors” of GRBL do not accept this command.
- 3) Axis values always denote a positive direction move, unless their values are preceded by a “-” sign. Ex: X1 (positive move) – X-1(negative move).
- 4) Different gcode generation programs and controllers may need a gcode program file to use a specific file extension other than .txt or .nc, which are the two most common file extensions in use. Regardless, all gcode program files should be capable of loading into a plain text editing program, such as Notepad. Care must be used if opening and saving gcode files with one of the popular, more fully featured text editor programs, such as Microsoft Word, that gcode programs are saved as plain text files, with no “fancy” formatting or “hidden” characters, which can cause some strange, and hard to find, errors during execution.
- 5) Gcode viewing programs are of extremely limited use with image gcode files, usually only displaying a solid color rectangle.

- 6) When more than one axis movement follows a G01 command, all axes will begin moving at the same time, and will finish at the same time. When more than one axis movement is preceded by a G00 command, each axis will begin at the same time, but the shorter distance moves will finish first.
- 7) G00 commands do not utilize axis acceleration and deceleration settings; G01 commands do. Consequently, G00 commands can frequently be a cause of lost steps (position loss) when/if the maximum feed rate is too high in the machine setup instructions.
- 8) Vector engraving gcode programs can use other more complicated commands not discussed in this paper, but we do not need them for raster engraving images.

Additional Reading Sources

There is a lot more about gcode programming that is not covered in this brief paper. Interested readers are encouraged to continue their learning with online research.

- 1) <http://www.cnccookbook.com/CCNCGCodeCourse.htm>
- 2) <http://en.wikipedia.org/wiki/G-code>
- 3) http://www.shapeoko.com/wiki/index.php/G-Code#G-code_supported_by_Grbl