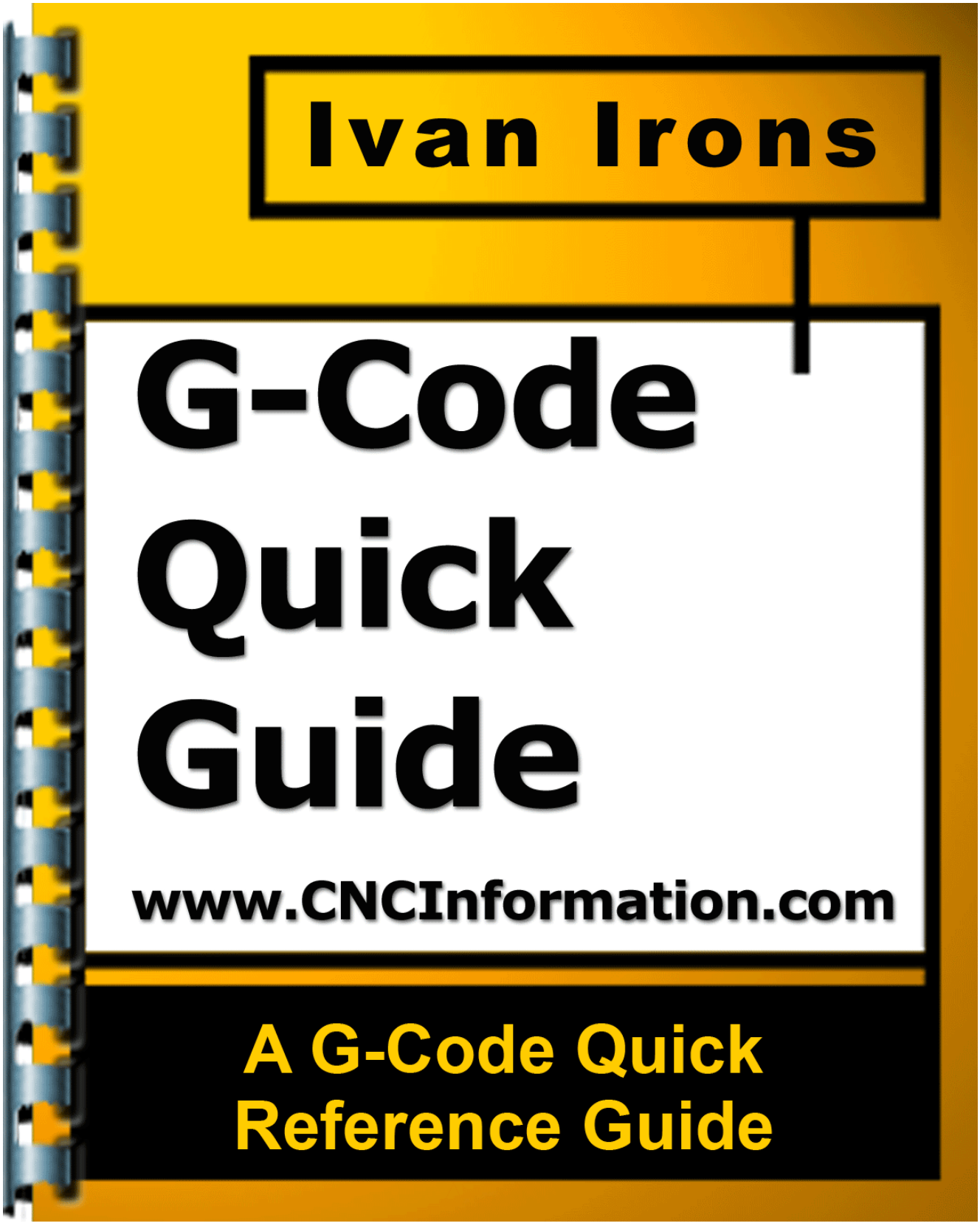
Ivan Irons
G-Code
Quick
Guide
www.CNCInformation.com
A G-Code Quick
Reference Guide

What is this guide?

A number of people ask me about G-Code on a consistent basis. It is one of those "unknown" things for people when they learn CNC. It is a computer language and that makes some people nervous. I wrote this guide to help people ease into g-code and to alleviate their anxiety about it.

What is G-Code?

G-Code programming is a very robust and simple language. G-codes are what CNC Machines run on. A CNC control program process the g-code line by line and send this information to the machine. The machine then takes this information and makes precise movements. Yes, it is as simple as that.

Where does G-Code come from?

That is simple enough, it come from your CAM program. When you process your design in your CAM program you pick out various machining parameters and inputs. You match it up with the CNC Machine the part will be run on and then Post-Process. You CAM program then spits out very precise G-Code for your machine and your part. If you make changes to your design, you will need to go through the CAM process again.

Post Processors

A post processor is a specific way you CAM Program will process your design. There are standard ones and custom ones. If a standard one doesn't work for you, most good CAM Software gives you the option to customize the post processor to fit your needs.

Canned Cycles

We won't be covering Canned Cycles here. There are G-Codes that do pertain to Canned Cycles. This guide is targeted toward beginner to intermediate CNC Learners.

The Big G-Code List

What comes next is a listing of what each g-code means. You can use this list if you are having a problem and are trying to decipher what is going on in your program. A note of caution…this is a generic list. Many manufacturers don't exactly adhere to this list and change the codes on a whim. You will need to be familiar with your own CAM Program, CNC Control Software and CNC Machine to determine if they have changed something. It is not something I can do for you.

G0 or G00 – Rapid Movement
The most rapid movement the CNC Machine can make to the next position. If moving in multiple axis, each axis will move as fast as they can independently of one another.

G1 or G01 – Linear Movement
A straight move with a speed defined by an “F.” [F=Feedrate] If moving in multiple axis, the machine will move in each axis until it reaches its defined position.

G2 or G02 – Interpolation Clockwise
A circular movement in 2-axis in a clockwise motion. Will create an arc to a specified radius defined by an R or I/J combination.

G2 or G03 - Interpolation Counter Clockwise
A circular movement in 2-axis in a counter-clockwise motion. Will create an arc to a specified radius defined by an R or I/J combination.

G4 or G04 – Dwell
Machine will dwell once reached position to a user defined time, noted by a “P”

G9 or G09 – Exact Stop/Exact Position
Machine will not traverse to next line of code until it locates exactly to specified position.

G17 - XY plane selection

G18 - ZX plane selection

G19 - YZ plane selection

G20 - Machine in inch

G21 - Machine in MM

G28 - Return to Reference Position
Normally machine home.

G30 - Return to 2nd reference position

G40 - Cutter Compensation Cancel

G41 - Cutter Compensation Left

G42 - Cutter Compensation Right

G43 - Tool Length Compensation +

G44 - Tool Length Compensation -

G54-G59 - Work Coordinate Systems
User defined, XYZ

G68 - Coordinate Rotation

G69 - Coordinate Rotation Cancel

G73 - Peck Drilling Cycle

G76 - Boring Cycle

G80 - Cancel Canned Cycle

G81 - Drilling Cycle

G82 - Drilling Cycle

G83 - Pecking Cycle

G84 - Tapping Cycle

G85 - Boring Cycle

G86 - Boring Cycle

G87 - Back Boring Cycle

G90 - Absolute Command

G91 - Incremental Command

G92 - Programming of Absolute Zero

G94 - Feed Per Minute

G95 - Feed Per Revolution

G98 - Return to initial point in canned cycle

G99 - Return to R point in canned cycle

The Future of G-Code

The future of g-code is in jeopardy. Kind of…but don't hold your breath. There are changes going on to try to create a new standard. A new standard where you can go straight from CAD or CAM right to the CNC machine.

This is basically an attempt to try to eliminate a step in the CNC Process. Others have tried to update or eliminate g-code before. So far, its simplicity has won out. It has survived. Is it worth learning a little about it today? Yes.

2" Square G-Code Example

The numbers on the left of the G-Code are the sequence numbers of the code and the order it will be executed in. I have added notes after different actions to let you know what that line of code means. My note starts with a "-" then continues. A "-" means nothing in G-Code.

N0000 (Filename: 2INSquare.txt) – Name of File
N0010 (Post processor: Plasma.post) – Name of Post Processor Used
N0020 (Date: 8/1/2007) - Date G-Code was generated
N0030 G20 (Units: Inches) – Measurement Units Used
N0040 G53 - Machine co-ordinate system
N0050 G90 – Absolute Programming
N0060 G40 – Tool radius compensation off
N0070 F1 – Feedrate of 1" per min
N0090 (Part: Square) – name of the part
N0100 (Process: Plasma, DEFAULT, Plasma, 0.01 in kerf) – Tool to be used
N0110 G00 – Go to Home
N0120 X0.1113 Y-0.0904 – Move to this exact X, Y position
N0130 G04 P2000 – Pause for 2 seconds, this is in milliseconds
N0140 M03 – Fire the plasma torch
N0150 G04 P500 - Pause for .5 seconds, this is in milliseconds
N0160 G03 X0.0000 Y-0.0050 I-0.1113 J-0.0298 F40.0 – move to this X, Y position in an arcing movement at a feedrate of 40 inches per min
N0180 G01 Y2.0000 – Travel in the Y+ direction for 2 inches to X0, Y2
N0200 G01 X2.0000 – Travel in the X+ direction for 2 inches to X2, Y2
N0220 G01 Y0.0000 – Travel in the Y- direction for 2 inches to X2, Y0
N0240 G01 X0.0000 – Travel in the X- direction for 2 inches to X0, Y0
N0250 G03 X-0.1113 Y-0.0904 I0.0000 J-0.1152 – move to this X, Y position in an arcing movement
N0260 M05 – Turn off the plasma torch
N0270 G04 P2000 – Pause for 2 seconds, this is in milliseconds
N0280 G00 – Move to Home
N0300 M30 – End Program

Do I need to be a G-Code Expert?

In a word, no. You will probably need to know how to quickly scan your G-Code if you are having problems during your machining simulation. Other then that G-Code and a CNC program are throwaway programs for the most part.

What are M-Codes?

M-Codes are other codes that are throughout your CNC Program. Think of them like the lesser cousin to G-Codes. **M-Codes are usually very specific to the machine.** For example, “M03” might stand for “Torch Off” on a CNC Plasma Cutter. It also might stand for “Spindle Turn Clockwise” on a CNC Mill or CNC Router.

M-Codes

M0 - program stop
M1 - optional program stop
M2 - program end
M3 - turn spindle clockwise
M4 - turn spindle counterclockwise
M5 - stop spindle turning
M6 - tool change
M7 - mist coolant on
M8 - flood coolant on
M9 - mist and flood coolant off
M26 - enable automatic b-axis clamping
M27 - disable automatic b-axis clamping
M30 - program end, pallet shuttle, and reset
M48 - enable speed and feed overrides
M49 - disable speed and feed overrides
M60 - pallet shuttle and program stop

Disclaimer

This is not a class on learning g-code. It is a quick guide of the different g-codes. In other words, this guide is for reference only. You should always prove out your process, program and cnc machine before you ever begin machining.

In order to learn the entire CNC Process, you will need to spend time. Time learning about the whole process, not just one part. G-Code is just one part of a whole process. If you are focused on only one part you are destined to have trouble.

This guide is assumed to be correct, but is subject to change at any time. This guide is for reference only. Your safety is your responsibility. Use you head.