

g-codesource.com  
User Orientation for  
Woodworking  
Applications

©2012 g-codesource.com

## Introduction

This document is for users of g-codesource.com g-code programs; to provide an orientation to the method for “portable” g-code programs.

g-code programs are more like a *file* sent to a printer than a *program*. It would be ideal if g-code programs could be moved from one machine to another without any modification, or the properties of different materials and cutting tools could automatically be compensated for, but the need for some program editing is normal. Programs are portable, but may need some tuning for optimal performance.

## g-code and your CNC machine

The starting point is to know the capability of your CNC mill or router. Capability ranges from axes travel limits, power, frame stability, to g-code command set the machinery can interpret. Every machine has a vocabulary of commands it accepts. g-codesource.com programs use a limited command vocabulary. The following table shows the maximum command vocabulary for woodworking programs.

G00	Set rapid motion mode
G01	Set fixed speed linear motion mode
G02	Clockwise circular or helical fixed speed motion mode
G03	Counter-clockwise circular or helical fixed speed motion mode
M0	Pause program execution
M2	End program
M3	Start clockwise spindle rotation
M5	Stop spindle rotation
M6	Tool change routine
F	Linear motion feed rate. The numbers after the F are the feed rate in the set units (inches or mm)/minute.
S	Spindle speed. The numbers after the S are the speed in rpm
T	Tool number. The numbers after the T are the tool number.

The following table shows settings that may be used in woodworking programs.

G49	Cancel any tool offset settings
G40	Cancel any cutter radius compensation
G17	Set the XY plane to the machine table
G80	Cancel any special motion modes
G50	Reset any scale factors to 1.0
G90	Set absolute distance mode
G20 or G21	Set inch unit mode or set mm unit mode

The woodworking programs may not use all of the listed commands, but they do not use other commands.

It is likely if you have a CNC mill or a full size CNC router, that the machine's vocabulary includes all of the listed commands plus other additional commands. Some manufacturers of CNC equipment, particularly tabletop machines, have truncated the command vocabulary. To complicate things, the literature provided with some of the machines may not explain what commands the machinery can process.

If any commands are missing, the most likely ones to be missing are:

M0 – pause

M6 and T – tool change

M3, M5 and S – spindle control

If you don't know the command capability of your machine, g-codesource.com has resources for assessing your machine.

## Cutting Tools

There is a broad selection of cutting tools available in many configurations and materials from many suppliers. Cutting tools are fairly expensive and they are consumables; they wear. The objective of tooling choices G-codesource.com uses is:

- to use a small number of standard tool sizes and types to allow fabricating parts without requiring a large library of or investment in cutting tools
- to use the largest tool practical to make the features of a part
- to use the tool that produces the best surface possible to limit the amount of finishing work needed after cutting a part.

If one were to optimize the fabrication of a part, there usually is a *best choice* for the cutting tools for the material and the CNC equipment used to get the fastest feed rate, but there often are many different cutters that will do the same job acceptably.

### ***2-flute versus 4-flute Cutters***

The choice of number of flutes usually depends on the material being cut and the CNC equipment. The choice affects feed rates, rotation speed, chip size, and chip removal rate. 2-flute cutters are generally better for softer materials and a high volume of chips. 4-flute cutters can cut faster but physically cannot move as large a volume of chips out of the way when cutting. 2- and 4-flute cutters frequently can be interchanged without program changes. Unless you are optimizing for high volume production, the choice often depends on the tools that happen to be on hand. 2-flute cutters are generally preferable for woodcutting applications.

### ***Router Bits versus Milling Cutters***

Router bits are optimized for wood cutting whereas one can find milling cutters optimized for cutting many different materials. In a pinch a router cutter can be used to cut metal. Most milling cutters work for cutting wood. Router bits usually have a straight, vertical cutting edge. Milling cutters often have a helical cutting edge like a drill bit to pull chips out. The lifting action of the helical cutting edge can cause wood surfaces to chip in some product designs and tool paths.

When cutting into a wood surface, sometimes the choice of the cutter type can depend on the chuck holding the cutter. Router chucks often hold one diameter shank, ¼ inch or ½ inch in the US, and the shanks often are smaller diameter than the cutter. Milling cutters shanks come in several sizes and either are the same diameter as the cutter or are a larger diameter.

One thing to watch for with router cutters are that some are designed for hand router use and may not have cutting surfaces on the end face to allow plunge cuts. Cutters used for g-codesource.com CNC applications must work for both side and end cutting.

Unless the fabrication instructions specify using a specific type of cutter, use any type cutter suitable for your CNC machine.

### ***Carbide versus HSS (High Speed Steel) Cutters***

Carbide is a very hard cutting material that holds a sharp edge well, but unfortunately is brittle. Bump a large carbide cutter against something hard and a piece of the cutting edge may chip off. Bump a small carbide cutter and the whole tip may break off. Wood has a high silica content that tends to dull cutters quickly. In the absence of trauma, a carbide cutter will have a longer useable life than a HSS cutter. Router cutters with braised carbide inserts can take an amazing amount of abuse without damage. High speed steel is less fragile than carbide but will not hold a sharp edge as long. For cutting soft materials like wood or aluminum, carbide or HSS can be used interchangeably. The choice usually is economic since carbide can last longer but costs more while HSS doesn't stay sharp as long but generally costs less. There is no requirement to use either carbide or HSS cutters.

### ***Cutter Size***

When fabricating a part, the specified cutter size must be used to make a part with the intended finished form. Using a different sized cutter will result in different finished part geometry.

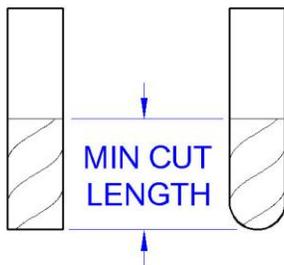
### ***Tool Table***

The table below shows the G-codesource.com standard tool library. The tool diameter is in its description. Note: a program may use only one cutter or a small number of cutters. It is not necessary to have the entire library of cutters listed below.

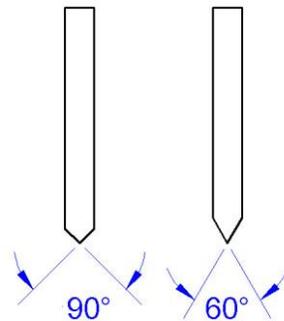
Inch Size Tool Number	Important! Use inch size cutters with inch formatted programs (G20)		mm Size Tool Number	Important! Use mm size cutters with mm formatted programs (G21)	
	Inch Sizes			mm Sizes	
	Tool Description	Minimum Cut Length (inches)		Tool Description	Minimum Cut Length (mm)
<i>Series 1</i>	<i>Flat Cutters</i>	–	<i>Series 5</i>	<i>Flat Cutters</i>	–
10	1/32 inch Flat	3/32	50	1 mm Flat	2
11	1/16 inch Flat	1/4	51	2 mm Flat	5
12	1/8 inch Flat	3/8	52	3 mm Flat	8
13	1/4 inch Flat	1/2	53	6 mm Flat	12
14	3/8 inch Flat	5/8	54	8 mm Flat	14
15	1/2 inch Flat	1	55	12 mm Flat	20

18	1/4 inch Flat Long	1-1/8	58	6 mm Flat	25
19	1/2 inch Flat Long	2	59	12 mm Flat	50
<i>Series 2</i>	<i>Ball Cutters</i>	–	<i>Series 6</i>	<i>Ball Cutters</i>	–
20	1/32 inch Ball	3/32	60	1 mm Ball	2
21	1/16 inch Ball	1/4	61	2 mm Ball	5
22	1/8 inch Ball	3/8	62	3 mm Ball	8
23	1/4 inch Ball	1/2	63	6 mm Ball	12
24	3/8 inch Ball	5/8	64	8 mm Ball	14
25	1/2 inch Ball	1	65	12 mm Ball	20
<i>Series 3</i>	<i>Angle/Engraving &amp; Specialty Cutters</i>	–	<i>Series 3</i>	<i>Angle/Engraving &amp; Specialty Cutters</i>	–
32	60° Engrave	Not Applicable	32	60° Engrave	Not Applicable
34	90° Engrave	Not Applicable	34	90° Engrave	Not Applicable
35	Fly cutter	Not Applicable	35	Fly cutter	Not Applicable
36	1 inch face cutter	Not Applicable	36	25mm face cutter	Not Applicable
<i>Series 4</i>	<i>Drills</i>	–	<i>Series 8</i>	<i>Drills</i>	–
40	Fractional size drills	Not Applicable	80	Decimal size drills	Not Applicable
41	Number series drills	Not Applicable			
42	Letter series drills	Not Applicable			

**FLAT AND BALL CUTTERS**



**ANGLE/ ENGRAVING CUTTERS**



## ***Flat Cutters***

Flat cutters are also known as end mills or straight router bits. Care needs to be taken when using router bits to use the style that is end cutting.

## ***Ball Cutters***

Ball cutters are known as ball end mills or round nose router bits. The caution with router bits is that some round nose router bits are specific just for cutting beads and have a short cut length limiting their use for CNC applications.

## ***Angle/Engraving & Specialty Cutters &***

Angle/engraving cutters often are single point style – a rod with a pointed end where a half side of the point is removed. This type cutter has one cutting surface, so the cutter lifespan is limited. Two flute cutters generally have a longer lifespan. The diameter of the cutter is largely irrelevant. Cutters with a diameter equal to or greater than 1/16 inch (1.5 mm) are acceptable.

There are many specialty cutters that are optimized to do a single, specific job. Where possible, specialty cutters are avoided. They usually are expensive and spend much of their life in the tool drawer since they only can do that one job.

## ***Drills***

There are too many drill sizes to assign tool numbers to specific drill sizes. When drilling is necessary, fractional drill sizes will be used preferentially since they are more commonly available. The specific drill sizes used in a program are identified in the program documentation

## **Editing a G-code Program, Basic**

For an operator of a CNC machine, editing g-code programs is unavoidable. Cutter spindle rpm and feed rates are the features of the programs that most often need to be adjusted. The right rpm and feed rates depend on the CNC equipment, the cutter type, the cutter condition, and the material being cut. Often higher spindle rates allow higher feed rates. Wood typically cuts well at higher rpms, so CNC routers typically have spindle speeds in the 30,000 rpm range while some CNC milling machine will max out at spindle speeds in the low thousands rpm. As the spindle speed is adjusted, the feed rate correspondingly can be adjusted.

Only the user will know the actual running conditions. Higher speeds may be needed to prevent burning or shorten time to produce parts. Slower speeds may be needed to prevent overloading the axes drives, exceeding the stock clamping capability, or improving the surface finish of the parts. Programs have to be tested to determine what works for the individual running conditions. Fortunately, a broad range of running conditions often work fine.

## ***Program Names***

G-code programs have a number of valid program extensions; some examples are .apt, .cls, .fnc, .gc, .iso, .mpf, .nc, .tap, .txt, and many others. None of these extensions are significant to the creation and editing of the program. The extension choice will depend on your CNC machine setup. If your CNC machine is expecting a particular name extension, change the name of the program to the desired extension. Programs from g-codesource.com are supplied with the .nc extension.

## Other Fabrication Considerations

### **Testing**

Once you have a new program and a piece of fine hardwood in hand, do not run the program on the hardwood for the first run as that run may illuminate issues with the setup and program-machine interaction. Rather, have a trial run on a low cost material (mdf, plywood, Douglas fir dimensional lumber, etc.) to test the fixturing, speeds and tools. A trial run also allows a full understanding of the event sequence, tool changes, and compatibility issues. It is possible to run a program without cutting tools installed, in the air above the machine bed, but little useful information can be gained.

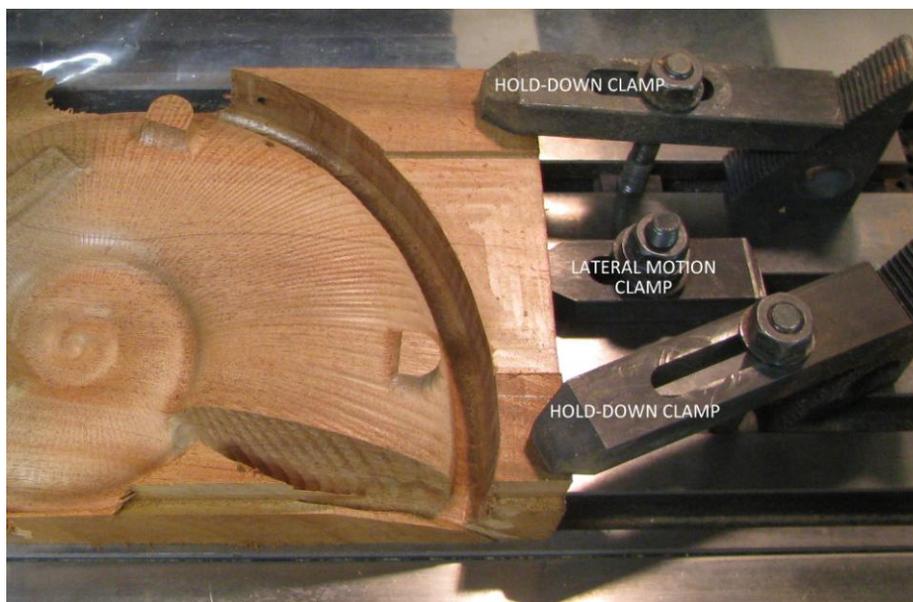
### **Wood Grain Orientation**

Wood is a highly oriented material. Significantly different results can occur when the stock is placed in different orientations. In G-codesource.com programs, the grain direction runs in the X axis direction. Cutting passes most often are performed in the X travel direction.

### **Clamping Stock**

Stock on a machine bed can move two ways, vertically up from the bed and laterally on the bed. Given the opportunity to shift, the stock will shift. Clamps that are set to do one function well usually do just the one thing well. Clamps that hold the stock down can be ineffective at preventing lateral motion. Clamps that prevent lateral motion may not prevent vertical motion.

The program documentation will identify locations clear from cutting for hold-down clamps but adequate clamping has to be determined by the end user. In the photograph below, three hold-down style clamps are shown, but only the two clamps on the stock corner are used for holding the stock down. The center clamp is bolted directly to the bed with its nose pressing against the edge of the stock to prevent lateral motion.



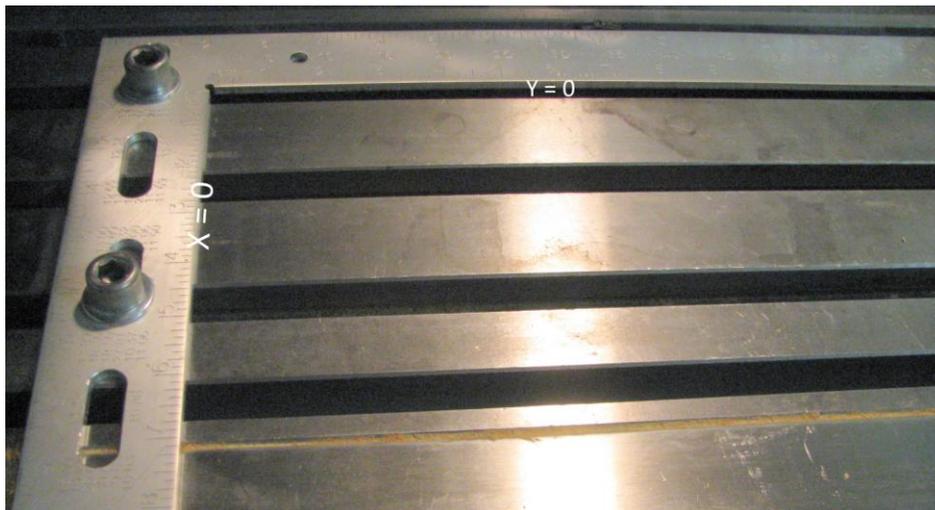
Since the stock grain orientation typically is in the X axis direction, and the majority of cut passes are in the X axis direction, Lateral Motion Clamps most often are needed to prevent motion in the X direction.

One of the outcomes that can be determined from a test run of a program is the effectiveness of the clamping method.

### ***Fixturing and X & Y Axis Zeroing***

There are many methods for setting X and Y axis zero. The X and Y zero settings along with stock clamping are critical in parts where two sides are shaped on the CNC machine and both sides have to be in registration with each other. Stock sizes are expected to not be exact, making registration more difficult.

A way to take advantage of the accuracy of the CNC equipment is through a fixturing method using a Table Square. A Table Square is a (any) flat, "L" shaped square mounted to the CNC machine bed with edges at  $X = 0$  and  $Y = 0$ . The following photo shows a Table Square installed on a machine bed.



The Table Square serves two purposes. It holds  $X = 0$  and  $Y = 0$  position from part to part for fast changes when doing multiple parts in series and it acts as a lateral motion clamp in two directions. The square shown above was adapted from a #1240 framing square from Empire Level Company, which is a 3/16 inch thick aluminum L square. One leg of the square was cut shorter and slots were cut for attaching the square to the bed.

The Table Square can be used with a CNC machine for precise registration when making parts with sides that have to be in registration to each other. On the first side processed, a reference edge is cut into the stock. When the stock is flipped over to process the opposite side, the reference edge is butted to the straight-edge to precisely register the two sides relative to each other. Some g-codesource.com programs employ this method.

The following photo shows the concept using a table square and a reference cut on a test machining run of a pistol grip. A machinist's square (a plain L square made of 3/16 inch thick steel) is used as the Table Square and clamped to the table with its inside edges corresponding to  $X = 0$  and  $Y = 0$ . The program for

cutting the first side (Up in photograph), cuts the reference edge. When the stock is flipped to cut the opposite side, the reference edge, which was cut to the CNC machine's accuracy, is set against the Table Square for good front/back registration.



### ***Restarting an interrupted run***

On occasion it is necessary to interrupt the running of a program (tool slipping or break, stock shifting, etc.). The most convenient place to restart a program is at the most recent tool change prior to stopping the program. Your user instructions should have information to help find the best places to restart an interrupted run.

## **Editing a G-code Program, Advanced**

It isn't necessary to do advanced program editing, but for those who like to dive deeper into things and are knowledgeable about g-code, advanced editing allows customizing your parts and using/reusing modules of code. With a structured program and appropriate comments, it is not difficult to find one's way around a program that someone else wrote, and make appropriate edits.

Program files are ASCII plain text files, and as such they can be edited with any basic editing program like *Notepad* (which comes with every Microsoft operating system), or Notepad++, a more powerful and free editor available at [www.notepad-plus-plus.org](http://www.notepad-plus-plus.org). There are a number of other free editors and also editors specifically made for editing g-code programs which have features to aid in navigating g-code programs. While it has the potential to make problems, word processing programs like Microsoft Word can be used to edit g-code programs. Word processing programs insert characters for formatting that are hidden from view but read by g-code interpreters, therefore the use of a word processing program is not recommended. If used, choose the "save as" function and select "plain text" as the file type to strip out hidden characters.

There may be multiple programs used to produce a part; one program for one side, another for the opposite side, one for surfacing between operations, etc. The file name has the following elements:

1. The program name starts with a short description of the item the program is used to make
2. The operational sequence the program is used in is given. When there are multiple programs to make a part, there always is a *First Op* and a *Last Op*. If there are more than two operations there may be programs identified as *Second Op*, *Third Op*, and so forth.
3. A geometry locator description like *Top*, *Bottom*, *Inside*, *Outside*, etc. gives a general sense of what portion of the part the program is used to manufacture.

Programs have five elements:

1. Initialization – to clear any pre-existing settings in the CNC machine memory, and to make settings that are required for the program
2. Set-up Check – a routine to verify that the material is positioned correctly and that the axes are zeroed correctly
3. Tool Changes
4. Feature Cutting
5. Ending

There is one Initialization and one Set-up Check at the start of a program, and one Ending. There can be multiple Tool Changes and Feature Cutting sections in a program. Between Tool Changes there can be multiple Feature Cutting sections.

*Comments* are text inserted into a g-code program to identify program elements. Comments are contained between parentheses. Every feature is an independent module. A feature can be removed by deleting the text from the parenthesis for the comment at the start of the feature to the parenthesis for the comment at the start of the next feature. For some features there is a dependency on subsequent features. Removing a roughing feature may make a subsequent finishing feature not work. However, features like engraving can be removed and sometimes replaced with other features to customize a part.

---

**END**