Standards for g-codesource.com Woodworking Programs

1/28/2012

This document is for developers of g-codesource.com g-code programs; defining standard practices in program structure, editing, and design considerations.

g-code Program Conventions

Editing a G-code Program

For an operator of a CNC machine, editing g-code programs is unavoidable. 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 powerful and free editor available at 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. Even word processing programs like Microsoft Word can be used to edit g-code programs, but they insert hidden characters for formatting when saving, 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.

Program Naming Conventions

G-code programs have a number of valid program extensions; e.g. .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. For consistency, use the .nc extension.

In a g-codesource.com project, 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 locator description like *Top, Bottom, Inside, Outside*, etc. gives a general sense of what portion of the part the program is used to manufacture.
4. A revision number. Programs occasionally must have multiple revisions to obtain the desired performance.

Units

Programs may be in inch or mm units. Inch unit programs must use inch sized cutters. Metric unit programs must use metric sized cutters.

G-code Commands and Settings

In the world of woodworking, there is a range in capabilities of CNC machines, from industrial machines with large g-code command vocabularies to tabletop machines with limited vocabularies. G-code programming has existed since the 1950's, well before computers were in common use. While there is standard for the language (ISO 6983), many proprietary G-code dialects were created to give individual controller manufacturers competitive advantages in the days when part drawings were made with

pencils and generation of g-code was difficult. Current CAM systems are great at generating complex coordinate motions for cutters and the advantage of different dialects has diminished. g-codesource.com programs for wood must be limited to a few g-code commands to make the programs as universally applicable as possible, but given the ability of current CAM systems to generate complex coordinate motions, these few commands can be used to make very complex part shapes.

The table below shows the commands that may be used in G-codesource.com program for wood applications.

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/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.

Even with this limited command vocabulary, there are machines that will not interpret all of these commands. g-codesource.com supplies applications to automatically modify programs for users whose machines do not process this entire command list. It is alright to have programs that do not use the entire vocabulary listed above.

In addition to the commands above, the following settings should be used once for initialization at the start of a program.

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

Capitalization
Although most machines are indifferent to capitalized or lower case letters, for consistency, use all capitalized letters in the code. Mixed cases can be used in comments.

Numerical Precision
All inch-unit programs should give coordinate values to the one-ten-thousand[th] value (0.0000). All mm-unit programs should give coordinate values to the one-thousandth value (0.000). Spindle rpm and feed rates are given to the one-hundred[th] value (0.00). Tool numbers are whole numbers between 0 and 99.

Drilling
Because of the router chuck designs, many CNC routers are unable to hold most drill bits. Since drill bits cannot be used on many machines, their use should be avoided where possible. Many holes can be cut using circular paths and flat cutters. Where small diameter holes or deep holes are necessary, the use of drills may be unavoidable. In cases where a drill is used in a program, the hole location should be spotted first with another cutter. With the hole spotted, those without the capability of drilling on their CNC machine may skip the drill routines in the program and drill the holes on a drill press after cutting the part, using the spotting to guide drill hole location.

Spindle rpm and Feed Rates
Parts generally can be made at broad range of feed rates and cutter rpm. Fabrication time can be optimized by specific settings of feed speeds and cutter rpm given knowledge of the specific operating conditions. Since it is not possible to optimize to all end-user operating conditions, making the g-code program friendly to end-user editing is more important than optimizing speeds. Limiting the number of feed speed and rpm variations limits the amount of editing necessary. The choices of rpm and feed rates are constrained by the CNC machine capability, and are affected by the tool type, tool condition, and wood type. Users will have to make rate adjustments and have the ultimate responsibility for optimization if that is important to them. The design of the program will determine how easy or difficult editing will be for the end user.

There are no universally right feed rates or spindle speeds. Use rates and speeds that work for your test cutting equipment. If a feed rate works for testing the programs, different working combinations are feasible.

Spindle speeds should only be set once at the tool change so a change at one location affects all cutting done with that tool.   End-users can insert additional rpm change commands where they like in a program.

Many CAM programs allow setting different feed rates for approaches, plunges, cuts and retracts. These functions allow optimizing a program for specific machines, cutters, and materials. But breaking the feed rates into different operations makes user editing of the programs more complicated. G00 should be used for rapid feed when not cutting. Plunge cuts and side cuts sometimes simply cannot be performed at the same speed. No more than two different feed rate settings should be used within the cutting of any feature.

When a feature requires several passes with retract and return motions with the program switching between G00 and G01 modes, each G01 callout should have a feed rate callout in the same line for CNC controllers that do not retain the previous feed rate setting.

Line Numbers
It is possible to have line numbers on g-code programs, but line numbering loses usefulness when a program is edited and numbers no longer are in sequence. Also many editors and CNC systems show their own line numbers and confusion can result when the line numbers embedded in a program do no match the actual line numbers. Line numbering is not used.

Blank Lines and Spaces
Blanks lines and spaces are ignored by CNC machines. Blank lines are not notably useful for editing as they do not provide search features nor make sections stand out visually when scanning through a programming at higher scrolling speeds. Do not use blank lines in programs.

Spaces can make an individual line appear more readable as they create visual breaks between each element within a line. Spaces are used between each command in a line.

Comments
Comments are essential program elements. Their importance is not to explain how the program functions but to identify the program name, identify copyright information and identify the start of every program element (more later on program elements). There are no comments to identify the end of program elements. Comments are contained between parentheses. Many CNC machines display program comments, but often display only a limited number of characters. Comments should convey information with as few words as possible, since only the first few words may be visible.

Repeat Linear Motion Commands
The G00 and G01 commands should set a fixed rate linear motion mode that the CNC machine should stay in until another mode is set. The commands do not need to be repeated.

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 when fabricating a small number of parts, the effort of optimizing can be more than the benefit realized.

2-flute versus 4-flute Cutters
The choice depends on 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. While 2-flute cutters are generally preferred for wood applications, the user most likely will use the tools they have on hand.   Design and test with the cutter type you prefer.

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.

Other than having both cutting faces, design and test with the cutter type you prefer. If the program requires a router cutter to minimize chipping, state the requirement in the fabrication instructions.

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 cutting tip may break off. Wood has a high silica content that tends to dull cutters fairly fast. 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 and is not as durable at holding a sharp edge. For cutting soft materials, carbide or HSS can be used interchangeably. The choice usually is economic since carbide can last longer but costs more while HSS doesn't last as long but generally costs less. There is no requirement to use either carbide or HSS cutters.

Tool Table

The following table shows the g-codesource.com standard tool library. The tool diameter is in its description.

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)
Series 1	*Flat Cutters*	*–*	*Series 5*	*Flat Cutters*	*–*
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
Series 2	*Ball Cutters*	*–*	*Series 6*	*Ball Cutters*	*–*
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
Series 3	*Angle/Engraving & Specialty Cutters*	*–*	*Series 3*	*Angle/Engraving & Specialty Cutters*	*–*
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
Series 4	*Drills*	*–*	*Series 8*	*Drills*	*–*
40	Fractional size drills	Not Applicable	80	Decimal size drills	Not Applicable
41	Number series drills	Not Applicable			
42	Letter series drills	Not Applicable			
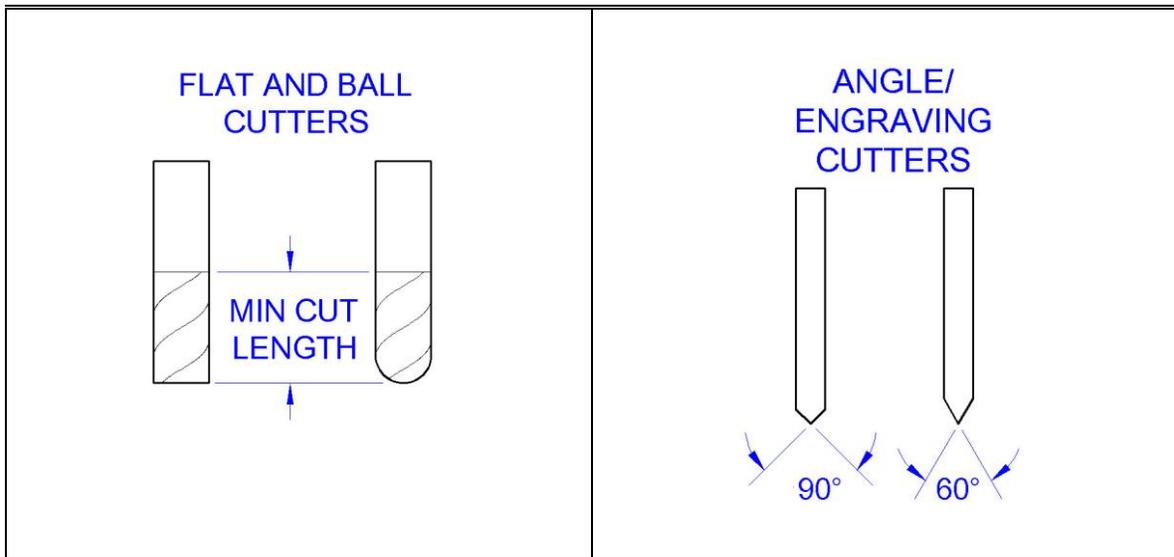
Flat Cutters

Flat cutters are also known as end mills or straight router bits. Care needs to be taken when using router bits to only 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 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 are acceptable.

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

Specialty Cutters

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

Program Structure and Comments

G-code programs are simple in terms of the variety of commands used, but can be extensive when defining complex parts. It is essential that g-code programs have a structured format and have

appropriate comments to make it easier for users to adapt a program to different CNC equipment, different tooling, and different materials.

Programs can be divided into one of the five following program 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 should be one Initialization and one Set-up Check at the start of a program, and each program should have 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.

Initialization
A typical program initialization looks like this:

```
(Initialize)
G00 G49 G40 G17 G80 G50 G90
G20
```

Initialization occurs only at the start of a program, just after title and copyright identification. The section starts with a comment identifying what it is. This initialization has the commands listed below. If the commands are not applicable to the CNC machine, they can either be left in the program if the machine ignores them, or they can be edited out of the program.

G00	Set rapid motion mode
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	Set inch unit mode (or G21, Set mm unit mode)

Set-up Check
Doing a Set-up Check is a way to prevent wasting time and materials from a simple set-up error. A set-up check can look like this:

```
(Set-up check)
X5.7500 Y0.0000
G01 Z0.0000 F40.0
Y-3.0000
G00 Z0.1000
X5.7500 Y0.0000
```

```
M5
M0
M3
G00 Z0.1000
```

The section starts with a comment identifying the section. This routine does a single cut across the middle of the stock from X=5.75 and Y=0, to Y =-3 at Z=0, then moves the cutting tool out of the way, stops the spindle and pauses the program to allow measuring. The motions and the checks in a Set-up Check are specific to each program. This Set-up Check is for a simple surfacing program so checking the height is the only thing that is needed. One often useful thing to check is to have a cutter move around the periphery of part cut outline to verify clamp clearance.

You can't tell from the program what checks to perform to verify the set-up. The documentation that comes with each program explains how to check to verify.

This section introduced the following commands:

G01	Set fixed speed linear motion mode
F	Linear motion feed rate. The numbers after the F are the feed rate in inches/minute when in inch mode.
M5	Stop spindle rotation
M0	Pause program execution
M3	Start clockwise spindle rotation

Programs should be written with spindle control commands as it is easier for customers to remove unneeded commands from a program than to generate missing commands.

Tool Change

Tool Change routines on different CNC machines can range from non-existent with no spindle control to fully automated no-touch tool changes. Programs are written for use with machines that recognize tool changes as it is easier for users to edit out incompatible sections than to write code for missing sections. A sample Tool Change looks like this:

```
(Tool 14, 1/2 inch flat)
G00 Z3.000
X5.75000 Y-4.0000
M6 T14
M3 S4500
G00 Z0.1000
```

This routine starts with a comment that lists the tool number and a concise description of the tool being installed. For machines that have a tool change position command, users can substitute that for the X, Y, and first Z motions. M6 calls the tool change routine and identifies the tool number. After the tool is changed, and the spindle is started with the M3 command and rotation speed for the tool is set. Finally, the tool is returned to a clearance position.

Spindle speeds (rpm) can be set or changed anywhere in a program. In G-codesource.com programs, spindle speeds are set once in the tool change routine and not modified until the next tool change. This

supports straightforward editing of spindle speeds as the same speed is used for the duration of use of the tool.

This section introduced the following commands

M6	Tool change routine
T	Tool number. The numbers after the T are the tool number.
S	Spindle speed. The numbers after the S are the speed in rpm

Feature Cutting
Often there will be a tool change followed by several feature cutting sections, then another tool change followed by several more feature cutting sections. A *Feature* is a part geometry or group of part geometries that are in some way related. A feature can be 3 lines or thousands of lines long. The program designer ultimately decides how to break the part geometry into features. The example below shows a feature

```
(Clear right corner)
X9.3713 Y-2.7434
G01 Z-0.1875 F30.0
X9.6263 Y-2.6006
X9.6899 Y-2.5611
X9.7493 Y-2.5156
X9.8037 Y-2.4643
X9.8287 Y-2.4365
X9.8521 Y-2.4074
X9.8738 Y-2.3769
X9.8937 Y-2.3451
X9.9118 Y-2.3123
X9.9279 Y-2.2785
X9.9554 Y-2.2090
X9.9797 Y-2.1382
G00 Z0.1000
```

Each Feature starts with a comment to describe the feature that is cut. Since comment words should be used sparingly, it is not necessary to use verbs like "cut" or to describe the type of cutting action like "parallel profiling" or "horizontal profiling." The text only needs to identify the affected geometric feature.

There are two required lines in a g-codesource.com Feature:
1. The first command line of the feature after the comment is an X and a Y coordinate starting point for the feature. Every feature starts with both an X and a Y setting.
2. The last command line of the feature is a Z axis clearance setting; in this case Z = 0.1000. This sets the tool in a ready position for the next feature. Every feature ends with a Z clearance setting.

With these two simple elements in place any feature can be:
1. removed from a program without having to check or modify the subsequent feature,
2. the order of features can be rearranged within a program as long as the feature remains with the same cutting tool, and
3. Features can be edited without concern that the edit may alter a subsequent feature.

In between the first and last element, various commands and coordinates are given to cut the described feature.

In some cases where a number of independent cuts are bunched together or where a profile cut is repeated multiple times to reach full depth, it can be helpful to subdivide a feature into *Passes*.

```
(Free Right side 12Pass*)
X7.2500 Y-1.6250
G01 Z-0.1875 F30.0
X8.2500
G00 Z0.1000
(*Pass 2)
X7.2500 Y-1.6875
G01 Z-0.1875 F30.0
X8.3750
G00 Z0.1000
(*Pass 3)
X7.1875 Y-1.7500
G01 Z-0.1875 F30.0
X8.6674
X9.4412 Y-2.0973
X10.1465 Y-0.1585
G00 Z0.1000
.  .  .

(*Pass 10)
X6.7753 Y-2.6273
G01 Z-0.1875 30.0
X7.2253 Y-2.0000
G00 Z0.1000
(*Pass 11)
X6.8516 Y-2.6820
G01 Z-0.1875 F30.0
X7.4352 Y-1.8955
G00 Z0.1000
(*Pass 12)
X9.6233 Y-2.6019
G01 Z-0.1875 F30.0
X10.4840 Y-0.2236
G00 Z0.1000
```

In this case, the feature has twelve passes. The starting comment gives the count of 12 passes for this feature and visually links the asterisk character to passes. Then each Pass starts with a comment consisting of an asterisk and identifying the pass number. Like a Feature, each Pass starts with an X and Y coordinate set and ends by setting Z in a clearance position.

Ending
Program ends should be simple and clean, putting the tool retracted and out of the way, turning off the spindle, and ending the program execution. A program end can look like this.

```
(End)
G00 Z3.0000
X0.0000 Y0.0000
M5
M2
```

Like all other program elements, this element starts with a simple comment. The comment acts as a closing comment for the previous feature. M5 stops the spindle. M2 ends the program.

Program Design

There are several do[s], don't[s], and try not to[s] in creating CNC applications for wood. There is more to generating a functioning g-code program than drawing a part on a CAD system and having a CAM program automatically generate the code. The following has some of the considerations that should go into a program for g-codesource.com.

Surface Finish

There are many different surface finish qualities obtainable when cutting parts. Often there is compromise between speed of part production and surface finish. The g-codesource.com programs were developed to produce the best surface finish. Often when doing a series of passes, the best surface finish is obtained cutting only in one direction although that is slower than cutting in a back-and-forth motion. It is fairly easy to measure machining time (your CNC machine may even do this) and usually is more nebulous measuring time spent sanding and finishing parts. The chosen philosophy of g-codesource.com programs is to have the machine do as much of the work as possible and minimize what the person has to do.

Z Axis Retraction

While the G-codesource.com programs generally are not optimized for minimal time, one time-wasting feature in CNC programs is time spent retracting and returning the cutter to cutting position. In g-codesource.com programs, retraction typically is held to a small distance, 0.1000 inches above the part, to minimize wasted travel time.

Wood Grain Orientation

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

Cut Direction

Wood is a natural material and has some challenges in working that don't exist in metal work. One undesirable thing that wood can do when cutting is chip. The method of cutting affects chipping. The two possibilities for cutting are Climb Cutting and Conventional Cutting.
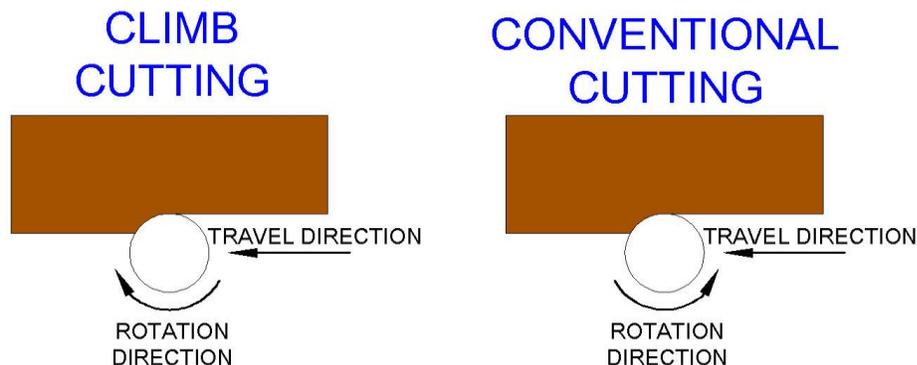
Conventional cutting can cause chipping in wood as the rotation of the cutter lifts the edge of the material away. Climb cutting presses the edge of the material into the material itself and usually results in less chipping.

Clamping Stock

There is no universal way to clamp stock to the machine bed. Securing can be done with hold-down clamps, a vise, position stops, vacuum chucks, hot melt glue, even double stick tape (for parts with light cuts). Part of the developer's job is define a workable clamping method. There are many different machine bed styles so users will have to adapt the method defined by the developer to their machine.

Stock on a machine bed can move two ways, vertically up from the bed and laterally on the bed. Given the opportunity to move, the stock will move. Clamps that are set to do one function well usually do just the one thing well. A hold-down clamp will prevent stock from moving vertically but can ineffective at preventing lateral motion. A clamp that prevents lateral motion may not prevent vertical motion.
If hold-down clamps are used, the program documentation should identify locations free of cutting for hold-down clamps. 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.
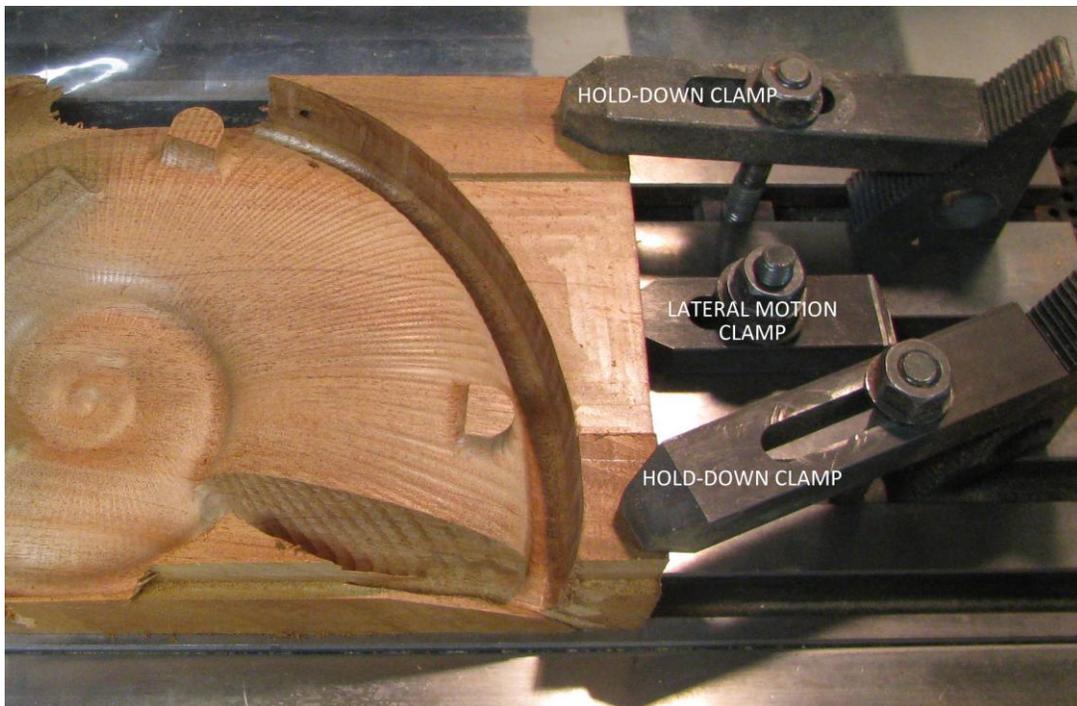
Since the stock grain orientation typically is in the X axis direction, and the majority of cut passes are in the X axis direction, Lateral Motion Clamps more often are needed to prevent motion in the X direction. If Y direction lateral motion clamps are needed, the program documentation will state so.

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

Stock Size and Z Axis Zeroing

In metalworking applications, the blank starting stock is likely to be precisely sized. In woodworking, the material generally is not an exact size and often has warped or irregular surfaces. The program documentation lists minimum and maximum length, width, and height dimensions for the stock. The program also may incorporate features to accommodate a wider range of dimensions.

Most shops with CNC equipment have a table saw where width and length can be cut with fair precision. When cutting stock to size on a table saw for example, the final stock measurement can be accurate within 1/32 inch (0.03125 inches), however CNC machines generally are accurate within 0.002 inches or less, greater than an order of magnitude more accurate than obtained on a table saw. The CNC machine generally brings a new level of accuracy to woodworking operation. Only shops equipped with a planer, will be able to set stock height with fair precision.

There are ways to configure projects to get repeatable results using stock that might not be repeatable size. One way to get a regular part height on a two-sided part made from an oversized height stock is to set Z=0 at the material height for the first operation, and then set Z=0 to a relative to the CNC machine bed for the operation on the opposite side when the material is flipped over. Program documentation will describe the Z zeroing method applicable to the part.

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.
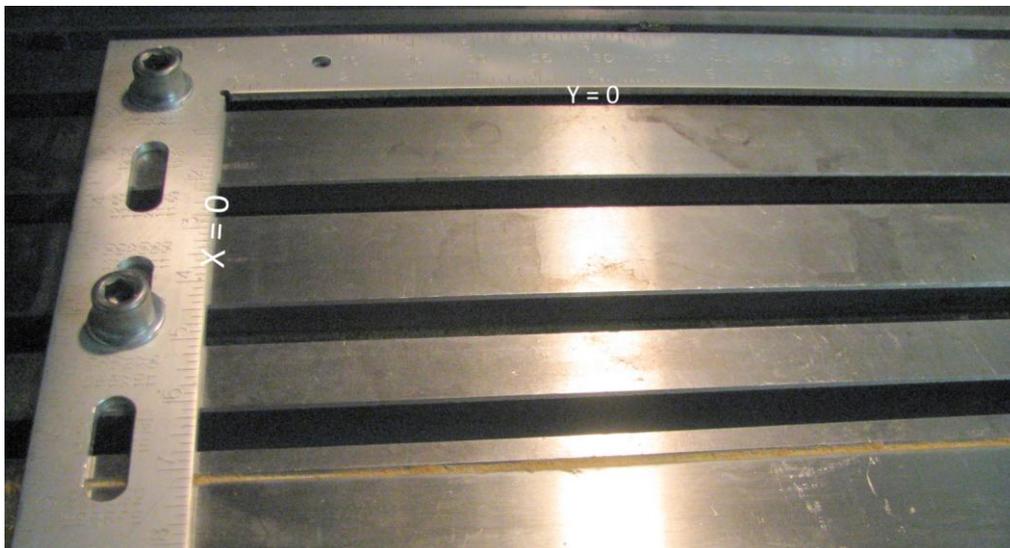
The Table Square serves two purposes. It holds X = 0 and Y = 0 position from part to part for fast changes when doing multiple parts in series and it acts as a lateral motion clamp in two directions. The square shown above was adapted from a #1240 framing square from Empire Level Company, which is a 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 the CNC machine's accuracy, is set against the Table Square for good front/back registration.

Restarting an interrupted program
One occasional occurrence is the need to interrupt the execution of a program and to be able to restart the program at a convenient point. There is no universally applicable way of doing this but the developer should suggest methods for restarting a program in the program documentation. Often a convenient place for restarting an interrupted program is at the previous tool change.

Testing

Every program should be rigorously tested by being running multiple times using different materials. MDF can be used to test geometry but is not a suitable material for assuring that the program will work with wood, if wood is an intended final material. Douglas Fir (usually available as dimension construction lumber) often is a good test material because of its low cost, availability, and tendency to

chip. If a program works with Douglas Fir, it will work with many other woods. In addition to verifying the final part produced, testing also should verify that the cutters stay within their intended range of travel, that the clamping method can hold the part in place, that the fixturing and zero positioning works, and that the program documentation can be followed. It is a best practice to have a tester who wasn't involved in the development of the program run the program on a different CNC machine. This confirms that the program is portable and that the documentation can be followed.

Program Documentation

Sales Information
The g-codesource.com philosophy is that the photographs are the key sales tool. The customer is going use the data to inform whether the operating range matches the available equipment capabilities. The high quality photographs are the best marketing tool for the design.

The sales data provides the following information:
- The tool travel ranges in x, y, and Z axes
- The stock size
- The unit system (inch or mm)
- The cutting tools used
- The number of sides cut or set-ups
- The cutting time per side (at as-provided or test feed rates)
- The complexity rating of the project (see below)
- The author/developer
- The cost (you the developer set the cost for the program)

Five part photographs should be provided with each program.
- One photograph of a completed part with finish in the environment in which it is used.
- One photograph of the part in a raw state as it taken off the CNC machine.
- One photograph of the part in a semi-processed state sanded but without finish applied
- The remaining two photographs are your option.

The backdrops used in photographs should contrast from the part so the part clearly stands out but should not distract attention from the part. The initial customer exposure to the part is a thumbnail size photograph so the parts should be prominent in the photos.

The complexity rating of the part is based on four project attributes:
1. The number of sides of the stock that are cut (for example if the top and the bottom of the stock are cut, the number of sides is 2)
2. The number of unique cutting tools used in the programs (for example if two cutters are used on the top side, and two cutters are used on the bottom side, but one of the cutters is the same for both top and bottom, the number of cutters is 3)
3. The number of programs that must be run to complete the project (for example if two programs are used, one for the top side and one for the bottom side, the number of programs is 2)
4. The final number of parts produced by the project (for example, if the project produces a separate box and a lid, the number of parts is 2)

Count and sum the four project attribute values. Use the following table to normalize the results on a scale of 1 to 10.

Attribute Count	Complexity Rating
4	1
5	2
6	3
7	4
8	5
9 to 10	6
11 to 12	7
13 to 14	8
15 to 16	9
17 or greater	10

User Instructions
The User Instructions provide the information needed to fabricate the part from the g-code program, The potential customers can download the user Instructions before purchasing the program to help decide if they can make the part and if they want to purchase the program,, The user instructions also should be considered a sales tool. The document should include photographs and illustrations beneficial to understanding the fixturing, clamping, and fabrication of the part.

The documentation must include:
- Project resources list describing:
  - The knowledge/skill sets needed to fabricate the part
  - The CNC machine capability the program is written for (the g-code commands employed and the cutting travel range
  - A list of cutting tools needed
  - A list of the program names for the g-code programs needed
  - A list of any other tools or equipment needed to fabricate the part
  - Material or stock requirements
- A safety warning
- Recommendations for testing the program
- A list of the major fabrication steps including at least:
  - Material/stock preparation
  - Fixturing and clamping
  - Instructions for each included program including a program flow table
  - Instructions for finishing the machined part
- Legal disclaimer

A good starting point for creating the user documentation for a program is to copy the format from an existing sample of user documentation. It is standardization, not plagiarism!

— **END** —