

G-ZERO Mill

Reference - Basics



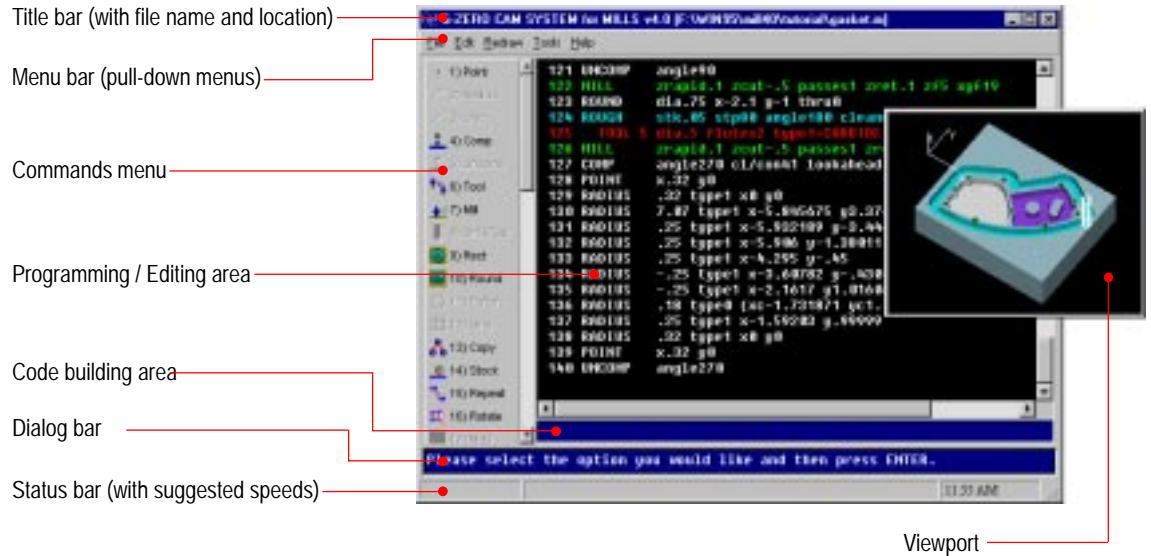
Contents

Screen Layout	5
Title bar	5
Programming / Editing area	5
Menu bar	6
Commands menu	6
Code building area	7
Dialog bar	7
Status bar	7
Viewport	8
Dialog Boxes (from Tools Options)	8
Preferences	9
Command Menu	9
Color and Font	9
G-code Text Editor	9
Metric / Inch System	9
Source Header - startsrc.org	9
File Management	10
Open	10
Incremental search - WinLauncher	10
Save	10
Save As	10
Backup	11
Merge Sources	11
Export Macro	11
Delete	11
Close	11
Print	11
Programming	12
Source Programming	12
Source Editing	13
G-code	14
Pop-Up Boxes	15
Edit Pop-up Boxes	15
Create Pop-up Boxes	15
Feeds and Speeds	16
Edit feeds and speeds	17
New material - Create new feeds and speeds file	17
Viewport	18
System	18
Clear Screen	19
Copy To ClipBoard	19
Print View	19
Zoom	19
Redraw	19
Rotate	20
SnapShot (with WinLauncher)	20

Solid Verification F6	21
Simulation	21
Tool display	21
Tool type	21
Solid View	22
Turbo Mode	22
Input Math	23
Operations and symbols	23
Hierarchy	24
Mainwin Calculator (Triangle)	24
Special Keys	25
Positioning	25
Programming	25
Tools	25
Viewport	25
Special Words	26
COLOR	26
ZTOP	26
% Commands	27
General	27
Tool change	27
Feedrate	27
Spindle speed	27
Coolant	28
Viewport display (F6 - Solid mode only)	28
Post	28
Math	28
Stepover	28
Rapid	28
Station	28
Wrap	29

Screen Layout

This section of the manual is intended to overview the components of G-ZERO Conversational CAM System for Mills main screen. Detailed information on specific features is covered in later sections.



Title bar

The title bar extends across the top of the application window, and displays the name of program as well as the name and path of the current source file. The three buttons located at the right end of the title bar allow you to minimize, maximize (or restore), and close the window. Closing the window will exit G-ZERO Mill.

Place cursor on title bar and:

- ◆ double-click the left mouse button to maximize or restore the window
- ◆ click (left mouse button) and drag mouse to relocate the window (windows should not be maximized)
- ◆ right-click (click right mouse button) to display right mouse list box with these commands: Restore, Move, Size, Minimize, Maximize, and Close (Alt+F4)

Programming / Editing area

This is the largest section of the G-ZERO Mill application window that displays the contents of your current source file. (Note that you need to Save your file before the displayed source code is actually recorded in your source file).

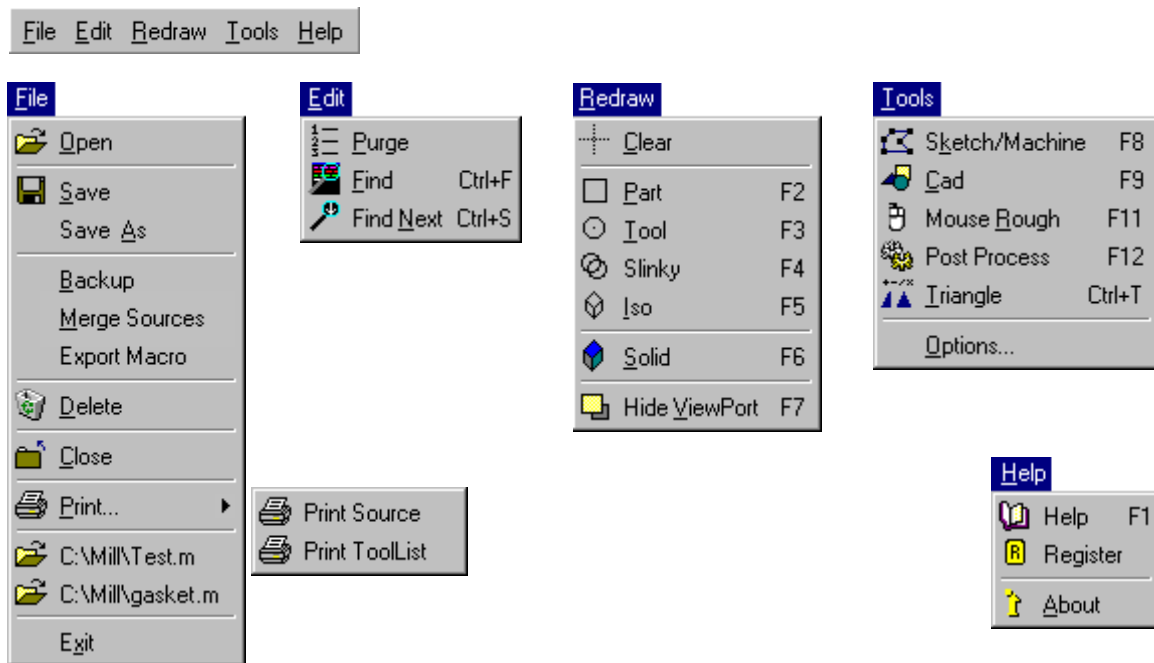
When the information on the programming area needs a larger display area, a vertical and/or horizontal scroll bar(s) will appear at the side and/or bottom of this section. Use your mouse to click the scroll arrows (at the end of the scroll bars) or click and drag the scroll box to move around the programming area and display the overflowed sections. Only one source file can be opened at a time.

The background color of the programming area as well as the font style and size can be adjusted in the Tools | Options | View window.

The Programming area is also the place where source code editing is done.

Menu bar

G-ZERO menu bar contains the following pull-down menus:



Use mouse to select an option from the menu bar. When the menu drops down, click on the command you want.

You can also use keyboard to make your selections. Press and hold the Alt key and key in the character that is underlined (e.g.: Alt+F+S for File | Save). Once the menu has dropped down, you can also use the keyboard arrow keys to make your selections and press the Enter key when your choice is highlighted.

A shortcut is available for commands most frequently used. Function keys or shortcuts are listed on the drop down menus.

Commands menu

G-ZERO Mill currently has 38 programming commands (#18 and #28 are not available) and they are listed on the left side of the application window. Use the scroll bars to display the rest of the commands.

To call (or use) a command, place your mouse cursor on top of the command icon or description you want, and click the left mouse button. You can also use the keyboard numbers (or 10-key pad) to call a command by typing the number that correspond to the command you want (shown before the name of the command). Example: keying in % has the same effect as using the mouse to click on the POINT command.

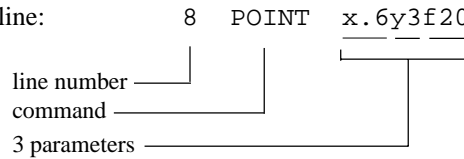
G-ZERO Mill normally displays the full command menu (icons and name); however, you have the option of displaying just the icons or no commands at all. The advantage of reducing the visibility of the commands menu is that it frees monitor space to increase the programming area. To adjust the size of the commands menu, place your mouse cursor on the right side of the commands scrolling bar until the cursor changes to a double arrow $\blacktriangleleft\blacktriangleright$; at this point, hold and drag it to the left or right to respectively reduce or increase the display of the commands menu. You can also, select (or unselect) *Full menus* from *Tools | Options | Views*.

G-ZERO disables (shows in gray) the commands that are not logically needed. The beauty of this feature is that it can guide new G-ZERO programmers in the proper sequence for programming.

Code building area

Each source code line is built in this area. Each time a programming command is called, a new line number and command name will appear on this bar. Commands parameters are also sequentially displayed here as the command is being formed. Once the entire command line is created, the source line will be transferred to the programming area.

Example of complete command line:



Dialog bar

This is the section G-ZERO uses to display the questions that will assist you to create a source line.

Example: When you first start a program, the dialog bar displays:

```
Please select the option you would like and then press ENTER.
```

When the POINT command is called, G-ZERO will immediately ask you the first question and waits for your answer to the first parameter value:

```
WHERE, IN X WOULD YOU LIKE TO POSITION THE TOOL? 0.6
```

You will notice that a default value is suggested by the program. This value is the last number entered to answer that specific question. If you accept that number, just press the ENTER key. If you need another number, just key in your new value.

After you enter the answer to the first question, the second question is displayed in the dialog bar:

```
WHERE, IN Y? 3
```

Again, the default value is displayed and G-ZERO waits for your reply. After you press ENTER, the last question will be asked:

```
MOVE TO THIS POINT AT WHAT FEEDRATE? (0=rapid) 20
```

After the last answer is keyed in, G-ZERO transfers the entire source line from the code building area to the programming area. G-ZERO will now wait for your next instruction:

```
Please select the option you would like and then press ENTER.
```

Status bar

The status bar extends across the bottom of the G-ZERO application window, and displays valuable information. Some of these information really help you decide for the most appropriate answer to the question displayed in the dialog bar. Examples of data displayed on the status bar are:

```
4.781, -2.041 . mouse cursor coordinates (only when mouse cursor is within the viewport display)
s60 S917 . . . . . s60 means 60 surface feet (G-ZERO calculates it using PartMaterial.S file based on
the tool type and tool size)
S917 means 917 rpm (G-ZERO calculates it using the surface feet and tool diam-
eter)
Zf=136.1 . . . . . Z feedrate (G-ZERO calculates it using the chipload listed in PartMaterial.S file
based on the tool type and tool size)
XYF=1.8 . . . . . XY feedrate (G-ZERO calculates it using the chipload listed in PartMaterial.S file
based on the tool type and tool size)
xff1.58=2.8 . . . recommended feedrate (base feedrate x feed factor)
1:52PM . . . . . current time (taken from computer clock)
```

Viewport

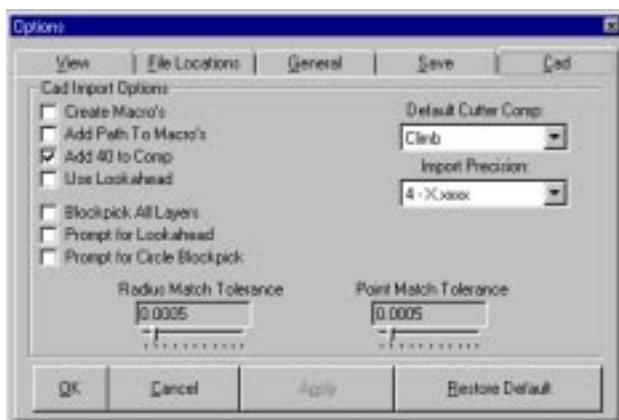
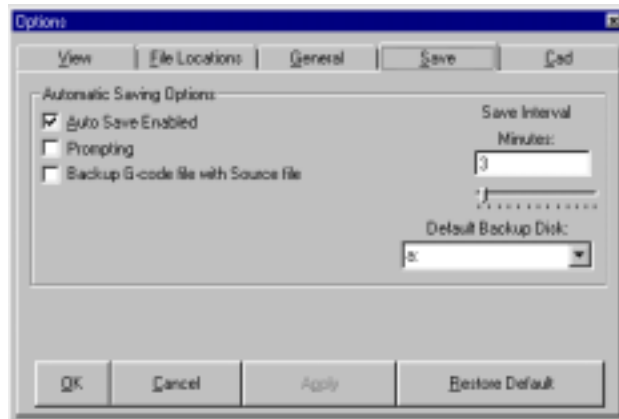
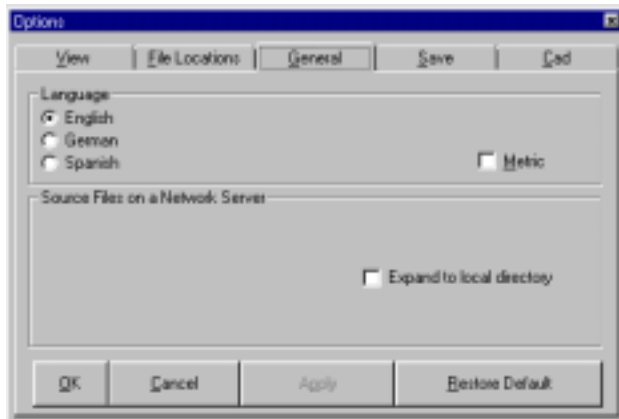
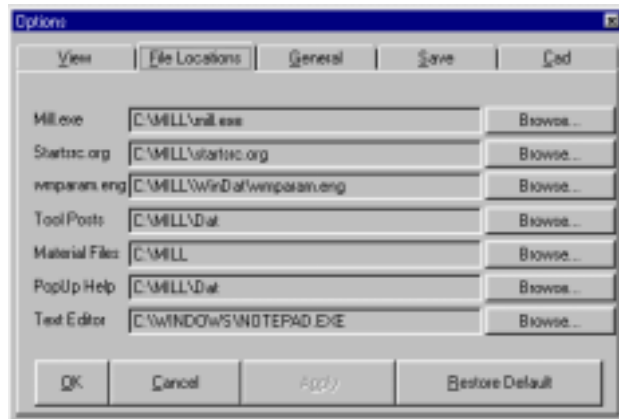
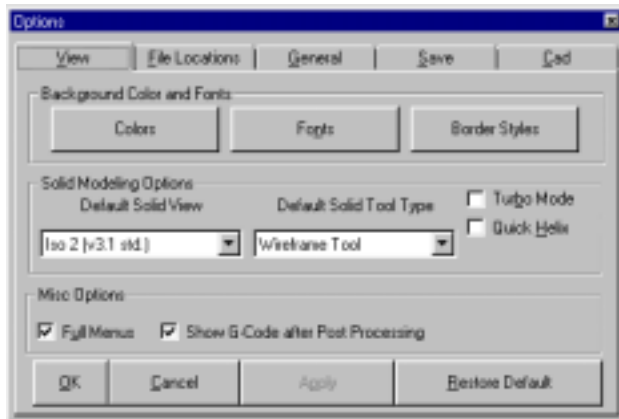
G-ZERO uses the viewport to display the graphical representation of the current source file in various formats (e.g.: solid view, wireframe, wireframe with tool path, etc.)

The viewport can also help you verify or debug a source program. Use your mouse to click on any section of the graphic displayed on it, and G-ZERO will find and highlight its corresponding source code in the programming area.

This viewport window can be resized and placed at any location on your computer screen, even attach it to the G-ZERO application window.

(See the Viewport section of this manual for more details.)

Dialog Boxes (from Tools | Options)



Preferences


Command Menu

You have the option of displaying full menus (icons and descriptions), or just icons. To switch from one option to the other, select (or unselect) the *Full Menus* check box in *Tools / Options / View* window.

Color and Font

You can customize the background color as well as the text font and size of your programming codes. To do this, make the desired changes in the *Background Color and Fonts* section in the *Tools / Options / View* window.

G-code Text Editor

G-ZERO uses the assigned Text Editor to open the G-code file right after it is created through the  20) POST command (F12).

Setup the G-code text editor you like in the *Tools / Options / File Location* window. The default editor is *Windows Notepad*.

Metric / Inch System

To program in Inches (output in inches):

- ♦ *Metric* check box (Tools | Options | General) needs to be unselected.
- ♦ **Ctrl+M** (input a millimeter value and let G-ZERO convert it to inches).
Example: key in **1** (mm) followed by **Ctrl+M** to have that number converted to **0.03937** (in)
- ♦ **Ctrl+Shift+M** (input an inch value and let G-ZERO convert it to millimeter).
Example: key in **1** (in) followed by **Ctrl+Shift+M** to have that number converted to **25.4** (mm).

To program in Millimeters (output in millimeters):

- ♦ *Metric* check box (Tools | Options | General) needs to be selected.
- ♦ **Ctrl+M** (input an inch value and let G-ZERO convert it to millimeters).
Example: key in **1** (in) followed by **Ctrl+M** to have that number converted to **25.4** (mm).
- ♦ **Ctrl+Shift+M** (input a millimeter value and let G-ZERO convert it to inches).
Example: key in **1** (mm) followed by **Ctrl+Shift+M** to have that number converted to **0.03937** (in).

Source Header - startsrc.org

If you would like a series of comment lines to pop up at the start of every new source program, create a text file named STARTSRC.ORG with the information you want (e.g.: part number, customer name, date, programmer name). Example:

1. Create text file startsrc.org

Use Notepad (Start | Programs | Accessories | Notepad) to store the information you like such as:

P/N:
CUST:
DATE:

Save this file to the Mill directory (e.g.: C:\MILL) using the file name STARTSRC.ORG (Make sure that the file name has extension ORG rather than TXT)

2. Register location of startsrc.org in G-ZERO

From G-ZERO's *Tools / Options / File Locations* use second Browse to locate the directory where you saved startsrc.org (e.g.: C:\MILL). Click *Apply* and *OK* to save settings and close Options window.

3. Start new source program / Use Source Header

Open new source program (File | Open and type new file name). Fill in comment lines if applicable.

File Management

Open

Start a new source file by typing a new filename in the *Source File to Open* window (File | Open). Since the file does not exist, G-ZERO will start a new source file.

Open an existing source file by selecting it from the *Source File to Open* window (File | Open). Make sure that the lower section of this window (file type) is set to display G-ZERO Mill source files (extensions .m)

Open an existing G-code file by selecting it from the *Source File to Open* window (File | Open). Make sure that the lower section of this window (file type) is set to display G-code files (extensions .t). G-code files are opened with the default text editor. To change the default text editor, go to Tools | Options | File Locations windows and use the Browse button to setup the link to the new text editor.



Incremental search - WinLauncher

Launching files through *WinLauncher* allows you to use the incremental search method: type the first letter(s) of your filename to narrow down your search by displaying only the filenames that meet your search requirements.

If you have taken a snapshot (*see Viewport section*) of your files, these pictures can help you search for the file you want to work on.



Save

Save your current source file by clicking the *Save* option from the *File* pull-down menu (File | Save). By default, G-ZERO Mill v4.0 automatically saves every three minutes. (This can be adjusted on the *Save* screen from *Tools / Options* menu.)

Save As

Save As creates a new file with the current information (duplicates the current file). If the current file has changes that were not saved, G-ZERO will prompt you to save the changes before closing the current file and starting the new file. Enter a new file name (you can change the drive/directory if you want).

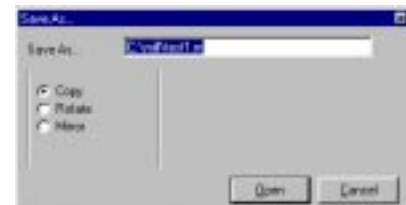
After entering the new file name, G-ZERO prompts you 3 options:

Copy. The new file has the exact same information as the current file.

Rotate. The new file has all data from the current file rotated at a certain degree and/or shifted in the x axis and/or y axis. *Note: Open rotated file and use 26)UnEdit/Recalc to update G-ZERO auto-calculated values before posting.*

Mirror. The new file has all data from the current file mirrored based on the x or y axis depending on your selection. *Note: Open mirrored file and use 26)UnEdit/Recalc to update G-ZERO auto-calculated values before posting.*

The new file is going to become the new current file. (Check the new file name and/or location displayed on the upper left corner of your G-ZERO main window)



Backup

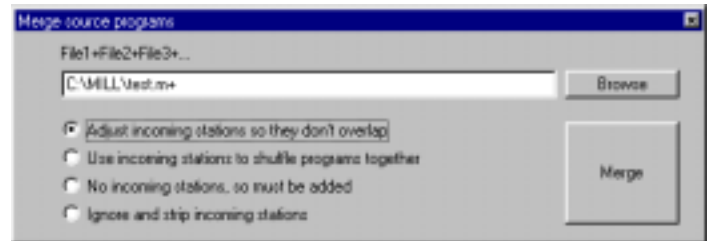
Backup the current file into a floppy disk by selecting the *Backup* option from the *File* pull-down menu (File | Backup). A backup file is an exact copy of the current file. The current file remains active. To change the default backup disk drive (A:), make the appropriate selection in the *Save* screen from *Tools / Options*.

Merge Sources

Creates a new file named *smerge.m* with the source of the current file plus one or more files of your choice. After the Merge source programs window appears, select one of the 4 options listed.

Note 1: You need to have one source file opened before invoking the Merge Sources command.

Note 2: After merging files, the *smerge.m* file becomes the active file. It's highly recommended that you rename this file to avoid overwriting the information the next time you perform another merge.

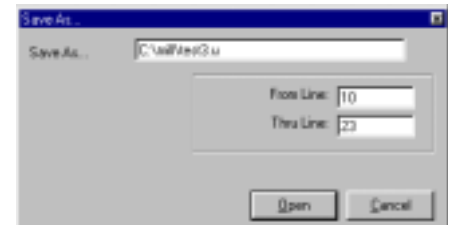


Export Macro

Create a macro file with a portion of the current source file by selecting the *Export Macro* option from the *File* pull-down menu (File | Export Macro).

After typing a new file name (for the macro file) in the *Save As* window, a new window pops up to allow you enter the sequence of source lines you want included in the new macro file.

Macro files are text files with default file extensions *.U*. They can be imported into a source file with the *#* commands.



Delete

Delete the current source file by selecting the *Delete* option from the *File* pull-down menu (File | Delete).

Close

Close the current source file by selecting the *Close* option from the *File* pull-down menu (File | Close). You can also click on the X button in the upper right corner of the application window (Title bar).

Print

Print entire source file


Send the current file directly to the default printer by selecting the *Print* option from the *File* pull-down menu (File | Print). To change the default printer, select *Start / Settings / Printers* from Windows taskbar (usually at the bottom of your screen), and select the *Set As Default* options in the right-click list of the printer you want to set as default.

Print from/to

To print only a section of your source code, use the PRINT command. G-ZERO will ask you to enter the first and last line numbers of the source codes you want to print.

Print tool list

To print only the lines that correspond to the TOOL command, select *File/Print/Print ToolList*.

Or, use the  21) Print command and print from line -1 to line 0.

Programming

Source Programming

G-ZERO is a Source Based Programming software that thinks the way a machinist thinks. You type in commands using exactly the same logic your CNC uses.

Programming Sequence

- First, start with the Material command (Mat'l). G-ZERO asks you several questions to guide you in providing the dimensions and type of material you are going to use. It's conversational!
- Second, program the Tool command. Again, G-ZERO will ask you several questions about the tool such as diameter, number of flutes, bull nose radius, and spindle speed. Whenever you need to answer a question about speed, G-ZERO will do some calculations and prompt you a suggested speed on the status bar.
- Third, program a cutting operation such as drilling (Drill&Tap), or milling (Mill). Once again, just answer the questions G-ZERO has for you about your cutting tool and depth of cut (Z value).
- Finally, program the geometry commands (e.g.: Point, Radius, Line, etc.) to tell your CNC the location of the cut. Just answer the questions about the X and Y values!

Repeat lines

Click source line number to instantly call the REPEAT command from that line.

- Example:
- Click on line number 22
 - G-ZERO calls the command REPEAT:
REPEAT from22 thru
 - Click on line number 27
 - G-ZERO completes programming the command:
REPEAT from22 thru27

Collapse/Expand lines

Ctrl+LeftArrow. Collapses source program hiding all lines except for the Material and Tools commands. Line numbers will be preceded by a double arrow symbol **»**. Note: If source lines from some Tools were previously made visible, they are not going to be collapsed.

Ctrl+RightArrow. Expands the entire source program to its fully visible condition.

Click on double arrow **».** While in collapsed condition, use your mouse to click on a double arrow **»** to display all lines from that Tool. Visible lines (while in collapsed condition) are shown with a dot **•** in front of their line numbers. Press any Redraw function key (F2 - F6) to view a graphical representation of only source lines shown.

Click on dot **•.** While in collapsed condition, use your mouse to click on any dot symbol **•** to collapse that Tool back down again and hide its source lines.

```
1 M03 S1000
2 G00 X0 Y0 Z0
3 M08
4 G01 Z-0.1 F0.05
5 G01 X1.25 Y0
6 G01 X1.25 Y1
7 G01 X0.875 Y-2.9872
8 G01 X1.875 Y-2.9872
9 M09
10 M30
```

```
1 M03 S1000
2 G00 X0 Y0 Z0
3 M08
4 G01 Z-0.1 F0.05
5 G01 X1.25 Y0
6 G01 X1.25 Y1
7 G01 X0.875 Y-2.9872
8 G01 X1.875 Y-2.9872
9 M09
10 M30
```

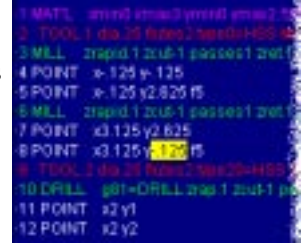
```
1 M03 S1000
2 G00 X0 Y0 Z0
3 M08
4 G01 Z-0.1 F0.05
5 G01 X1.25 Y0
6 G01 X1.25 Y1
7 G01 X0.875 Y-2.9872
8 G01 X1.875 Y-2.9872
9 M09
10 M30
```

Source Editing

Change a value

To change a value in a source program, use your mouse to **click** the value you want to change, **type** in a new value to replace the one that is highlighted. The background of the editing area turns to blue when you are in editing mode; press the **Enter** key to save your changes and exit editing mode. Press **Escape** to leave it unchanged.

Note: To edit a *comment*, just click on the comment to access the editing mode. Use **Backspace** and **Arrow keys** to assist your editing. Press the **Enter** key when done, or **Escape** to abandon editing.



```
1 MATE  arrd1 arrd2 arrd3 arrd4 arrd5  
2 POINT 1 dia 25 holes1 passes1  
3 MILL 2rad1 2cut1 passes1  
4 POINT  x-1.25 y-1.25  
5 POINT  x-1.25 y2.825 R5  
6 MILL 2rad1 2cut1 passes1  
7 POINT  x3.125 y2.825  
8 POINT  x3.125 y2.825 R5  
9 POINT 1 dia 25 holes1 passes1  
10 DRILL g01+DRILL rad1 cut1 p  
11 POINT  x2 y1  
12 POINT  x2 y2
```

Change sign

To change the sign of a value, use your mouse to **click** (left mouse button) the value to which you want to change the sign, **right-click** (right mouse button) the just highlighted value to change its value from positive (+) to negative (-), or negative to positive. Press the **Enter** key to exit editing mode.

POINT to RADIUS type2

Press and hold the **Alt** key and use your mouse to **click** on a POINT command to convert it to a RADIUS type 2 (corner). This is a nice help to clean up CAD data.

RADIUS type2 to POINT

Press and hold the **Alt** key and use your mouse to **click** on a RADIUS type 2 command to convert it to a sharp POINT.

LINE (unknown angle 0 known angle)

Press and hold the **Alt** key and use your mouse to **click** on a LINE command with **unknown angle** to convert it to a LINE command with **known angle**.

RADIUS (type0 unknown radius 0 type1 known center)

Press and hold the **Alt** key and use your mouse to **click** on a RADIUS type 0 command (**unknown radius**) to convert it to a RADIUS type 1 command (**known center**). Note that there is no “undo” for this procedure, but you can always manually change type “1” to type “0” (the x and y values in parenthesis will be ignored by G-ZERO for all RADIUS type0)

Search source line using Viewport

Click on any wireframe or solid parts in the viewport (walls or floors) and G-ZERO will highlight the related command line in the source program. If you see that the viewport shows a graphic that doesn't seem correct, click on the questionable element to locate the line code that needs to be changed.

Purge file

Most “deleted” lines in G-ZERO are actually just hidden, and not permanently deleted. The Purge option (from the *Edit* pull-down menu) permanently removes all such lines from the source program. After a file is purged, all line numbers will be automatically re-sequenced.

Find - Find Next

Find. Use *Ctrl+F* or *Edit / Find* to locate the first value (or strings of values/characters) within your active source code.

Find Next. Use *Ctrl+S* or *Edit / Find Next* to locate the next appearance of the value (or strings or values/characters) previously searched using *Edit / Find*.

G-code

After programming your source, G-ZERO can regenerate your entire part program in precisely the language your CNC understands. *Rapid Output* customizes your post processors so your time is spent making good parts, not editing tedious G-code.

Generate G-code

You only need two simple steps to generate G-code:

- ◆ **F12.** Press F12 (or *Tools / Post Process*, or *20) Post*) to open the Tool Post Selection window. (Note that post processors are customized and sold separately)
- ◆ **Post.** Select a tool post (for example: FADAL) and watch how G-ZERO creates the right G-code for the machine selected. G-ZERO also saves the G-code file (extension .t) and displays it on your screen using Notepad (or your default G-code Text Editor)

To change your default G-code Text Editor, click on the *Tools / Options* menu, and select the “File Location” dialog box. On the “Text Editor” section, assign the program you would like to use as the default text editor.

Open G-code file

Select *File / Open* from the G-ZERO application window. In the “Source File to Open” window, make sure to display “G-code (*.T*)” type of file in the “Files of type” option at the bottom of the window.

Cancel the “Sent to CNC...” window if you don’t want to send it now; G-code will be opened with the default text editor (e.g.: *Windows Notepad*)

Post Processor Perfection

If you ever need to edit your G-code, DON’T DO IT! What you need is to update your post processor, and we can do it for you when you send us (via e-mail support@g-zero.com, fax 510-475-0483):

1. Print out of the incorrect G-code with clear instructions (e.g.: cross out codes you don’t need, or write down those you need to add)
2. Source program (.m file) that generated the incorrect G-code and post (.p file)

Pop-Up Boxes

Pop-up boxes add extra information to your computer screen to assist you answering G-ZERO prompts. For example: the 6th question for the command 17) MAT'L asks "What is the part made of?" This question appears along with a pop-up box that lists the numbers to key in that correspond to specific types of materials.

Information from pop-up boxes are stored in text files in the MILL\DAT directory. The default pop-up boxes are:

File name	Command	Question #	Sample contents
17.6	17) MAT'L	6	Type of materials (e.g.: 303 STAINLESS)
6.4	6) TOOL	4	Type of tools (e.g.: CARBIDE MILL)
8.1	8) DRILL	1	Drill Cycles (e.g.: C'DRILL)
81.4	8) DRILL	4 for 81=C'DRILL	Standard number of pecks <i>or</i> peck distances
82.4	8) DRILL	4 for 82=C'SINK	Countersink diameters <i>or</i> seconds of dwell
83.4	8) DRILL	4 for 83=PECK	Number of pecks <i>or</i> depth per peck
84.4	8) DRILL	4 for 84=TAP	Threads per inch
15.3	15) REPEAT	3	Reminder of value substitution option

Edit Pop-up Boxes

If the standard pop-up boxes do not contain exactly what you need, the information can be changed. For example, if you have always used a countersink tool in slot 3, you may want to keep it that way and change the pop-up box. To edit the information in a pop-up box, use a text editor such as Windows Notepad.

Example 1: You want to modify the pop-up box that appears when you are answering the 4th question from command 6) TOOL. Use Windows Notepad to open the file C:\MILL\DAT\6.4 and make your changes.

Note: if you are using a word processing, make sure you always save the files as TEXT ONLY.

Example 2: You want to delete the pop-up box that appears when answering the 3rd question of command 15) REPEAT. Delete the file 15.3 located in C:\MILL\DAT.

Create Pop-up Boxes

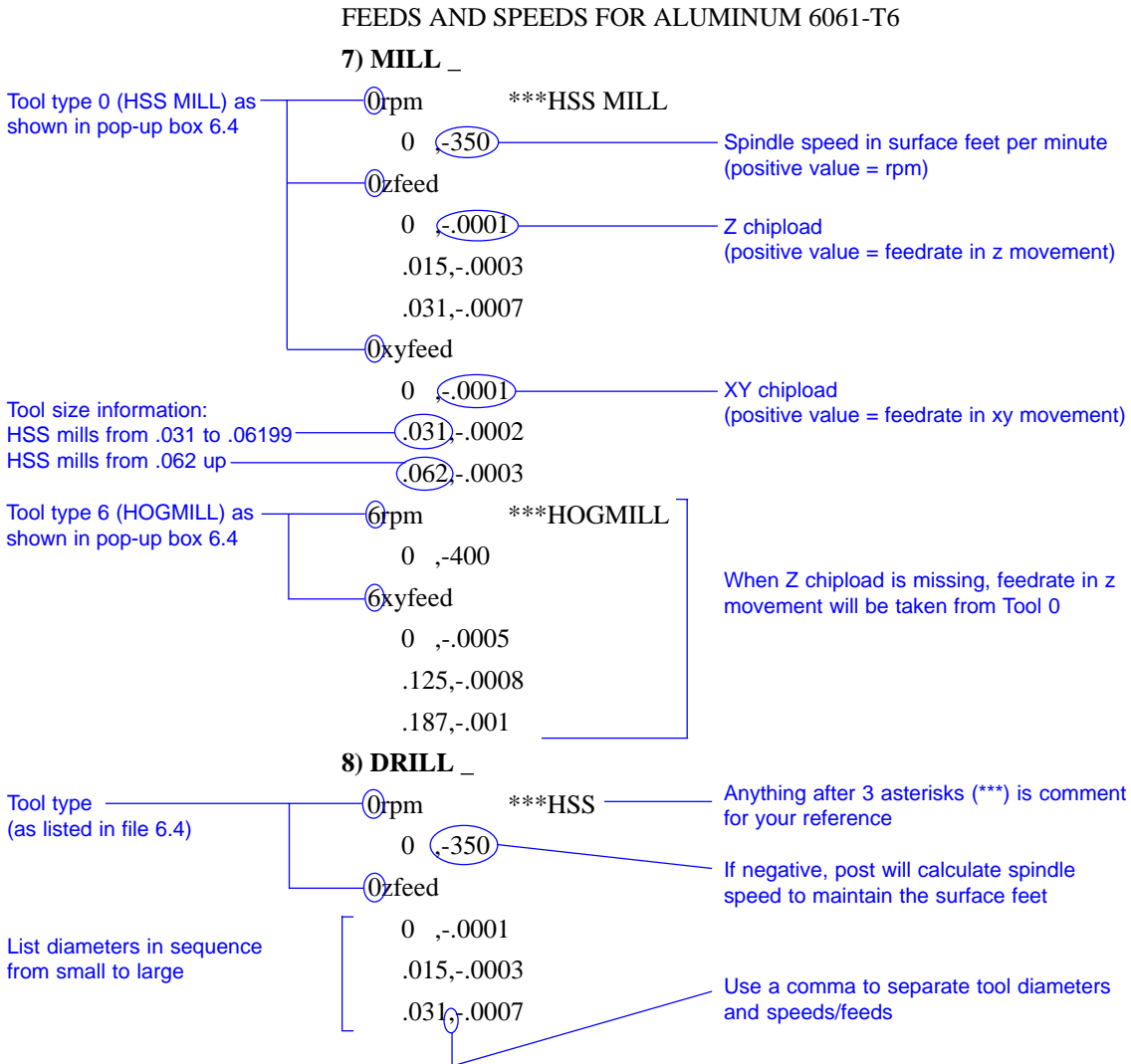
To create a new pop-up box, you first need to know the command number and order of question in which you want the pop-up box to appear. This file needs to be a text file located in the C:\MILL\DAT directory (assuming your Mill program is in your C: drive). The file name is formed by two numbers separated by a period. The first number corresponds to the command number, and the second number represents the sequence of the question.

Example: You want to create a "reminder" to pop-up when you are answering the first question of command 15) REPEAT. Using *Windows Notepad*:

1. Start Notepad: Start | Programs | Accessories | Notepad
2. Type your message within Notepad. Example:
To reverse the direction your cutter takes along a surface, use repeat backwards.
Example: FROM 14 THRU 10
3. Save your file using the appropriate name and location:
File | Save As
C:\MILL\DAT\15.1

Feeds and Speeds

Feeds and speeds defaults are setup in separate files according to the type of part material. The file name should be the same as the text used in the pop-up box 17.6. G-ZERO Mill system comes with the speed files: ALUMALOY.S, 303.S, 304.S, 1018.S, and 4140.S.



The **7)MILL** and **8)DRILL** sections of this file contain sets of rpm, zfeed and xyfeed for the different **types of tools** starting from 0. 8)DRILL section has no xyfeed info, of course.

- ◆ Each tool type section contains the rpm, zfeed, and xyfeed for the different tool sizes. Whenever a zfeed or xyfeed section is missing, G-ZERO will take the values of tool type 0. This saves you time in having to duplicate all the information for similar types of tools.
- ◆ Be sure not to have any blank lines or missing values in the file.
- ◆ The number in front of rpm, zfeed, and xyfeed refers to the number of the type of tool (as listed in pop-up menu 6.4)
- ◆ The minus means that this value is to be calculated.
- ◆ Enter the tool size (diameter) in sequence from small to large.
- ◆ Separate the diameters and the speed/feed with a comma.
- ◆ Anything on the right side of the 3 asterisks (***) is simply a comment for your reference.

Example 1: 7) MILL_
 Orpm ***HSS
 0 ,6000
 .125 ,5000
 .25 ,4800

In this example, we are telling the post that all tools of type 0 (HSS) from a diameter of zero up to a diameter of .125 should run at 6000 rpm when you are milling. Tools with diameter between .125 and .25 should run at 5000 rpm, and tools with diameter larger than .25 should run at 4800 rpm.

Example 2: 7) MILL_
 1zfeed
 0 ,1
 .125 ,3.5
 .5 ,7

Here, we are telling the post that when we are milling with tool type 1 (CARBIDE MILL) we want to plunge in Z at 1 ipm (for tools smaller than .125), 3.5 ipm (for tools between .125 and .5) and 7ipm (for tools larger than .5).

Example 3: 8) DRILL_
 Orpm
 0 ,-350

These lines tell the post that it should calculate the spindle speed to maintain 350 surface feet for any sized tool.

Example 4: 8) DRILL_
 0xfeed
 0 ,-.0005
 .125 ,-.001

Here, we tell the post to allow .0005 chipload with very small drills (up to .125), and to use .001 chipload with .125 drills and bigger.

Edit feeds and speeds

All feeds and speeds files (files with extension .S) are text files, so you can use any text editor such as *Windows Notepad* to edit them. Feeds and speeds files are located in the MILL directory.

Note: Make sure not to leave blank lines or empty values in the feeds and speeds files.

New material - Create new feeds and speeds file

Whenever you introduce a new material (e.g.: Steel 5150), you need to:

1. **Add new material to file 17.6** to update the pop-up box for 17) MAT'L command. File 17.6 is located in the MILL\DAT directory.
 Example: if we want to call the new material "5150" and assign it the number 5, then just add the line **5=5150** to file 17.6.
2. **Create speed and feed file with file name as shown in file 17.6 and extension "S"** (e.g.: 5150.S).
 New feeds and speeds file needs to contain the 7)MILL and 8)DRILL sections to include the new feeds and speeds for all milling and drilling tools.

Note: To simplify your work, use an existing file as a template:

- Chose an existing material similar to the new one you are adding (e.g.: Steel 4140).
- Duplicate the existing material file and rename it with the new material (e.g.: use *Notepad* to open file 4140.S and use *File / Save As* to rename it to **5150.S**).
- Edit the tools, speeds and feeds values to suit your new material.


Viewport

The viewport is a window independent from the G-ZERO application window. Its size, location and display can be adjusted through a **right-mouse-button list box** and a **toolbar**. (To display the right-mouse-button list box, place your cursor in the Viewport and click the right mouse button.)

Note: Press the F7 function key to hide or display the viewport from the G-ZERO application window.

System



Move - (Undocked) Your mouse cursor will change to a four-arrow icon  (cursor needs to be on viewport), click and hold the left button of your mouse and drag the entire viewport to the location you want. This same action of dragging the viewport to a new location can be done by clicking the *Move* icon from the toolbar.

Full Screen - (Undocked) Viewport will take up the entire screen of your monitor. To go back to the previous screen size, click the right button of your mouse, select System, and the option *sizeable screen*.



Show Editor - Hides the viewport and returns to the source code programming screen. The same action can be performed by pressing the F7 function key.

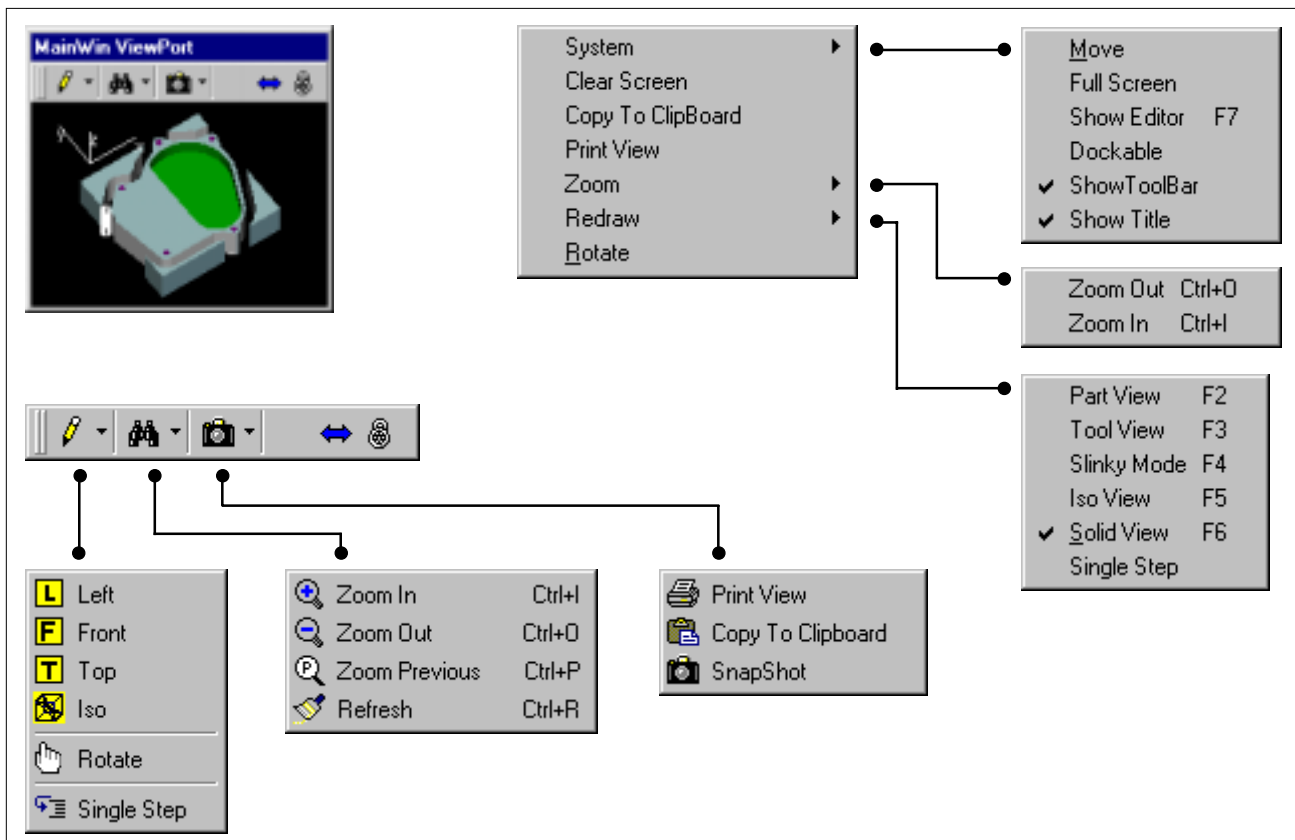


Dockable - If this option is selected, the viewport will be docked to the upper right corner of the source code programming screen. To undock the viewport, unselect the *Dockable* option from the *System* list. Docking or undocking the viewport can also be done by clicking the *Dockable* icon from the toolbar.




Show Toolbar - Select this option to display toolbar at the top of the viewport. To hide the toolbar, unselect this option.

The toolbar can be detached from the viewport. This is done by clicking on the gray area of the toolbar and dragging it to another location. To dock the toolbar to the viewport, drag the toolbar back to the top section of the viewport.

Show Title - (Undocked) Select this option to display a title bar at the top of the viewport. To hide the title bar, unselect this option. The title bar is also used to display the current Tool number during a simulation.



Resize Viewport

Just like any Window, you can adjust the size of the viewport using your mouse. Place your mouse cursor on any edge or corner of the viewport to change the white arrow cursor to a double black arrow pointed up and down , or pointed left and right , or pointed diagonally . At this point, click and hold the left button of your mouse and drag the edge (or edges if it's a corner) of the window to increase or reduce the size of the viewport.

Clear Screen

Select this option to clear any graphic currently displayed in the viewport.

Copy To ClipBoard



Once you click on this option, G-ZERO will automatically send the graphic currently displayed in the viewport (without border and title bar) to the Windows Clipboard. You can use another program (such as Paint) to paste and print the image just captured. *(Note: For best results, set your computer color palette greater than 256 colors)*

Print View



Click on this option to send the graphic currently displayed in the viewport to your default printer. Note that printing with this option will include the black background.

Zoom



Zoom Out - Zooms out the current display; in other words, more (and smaller) graphic is displayed in the same size viewport. The keyboard shortcut is Ctrl+O.




Zoom In - Zooms in the current display; in other words, less (and larger) graphic is displayed in the same size viewport. The keyboard shortcut is Ctrl+I.


Zoom options NOT available through the right-mouse-button list box:

Zoom Window - Click (left button) and drag from one corner of the area you want to enlarge, to the opposite corner.



Zoom Previous - To zoom to the previous zoom window, select  from the Zoom options pull-down menu on the toolbar. The keyboard shortcut is Ctrl+P.



Refresh - To restart simulation at the current zoom window, select  from the Zoom options pull-down menu on the toolbar. The keyboard shortcut is Ctrl+R.

Redraw



Part View F2 - Redraws the entire source program without tool path.



Tool View F3 - Redraws the entire source program and also shows the tool path.



Slinky Mode F4 - Redraws the entire source program and also shows the tool path in slinky mode.



Iso View F5 - Redraws the entire source program in isometric wireframe view.



Solid View F6 - Starts the solid view simulation of the entire source program in the default solid view. (See Solid Modeling / Simulator section). Note: Press the A key to stop simulation.



Single Step - Redraws your entire source program one step at a time. The source code programming window highlights the line code it is simulating in the viewport. Each time you click the left button of your mouse (or press the space bar from your keyboard), the simulation advances one step (one line in the source program). Note: Press the S key to exit the Single Step mode and finish the redraw in normal simulation speed. Similarly, press the S key while redrawing (wireframe or solid) to continue the simulation in single step mode.

Redraw options available ONLY through the Toolbar:



Left View - Starts simulation of the entire source program from its Left View. Redraw will be done in wireframe or solid mode depending on the last redraw command used.




Front View - Starts simulation of the entire source program from its Front View. Redraw will be done in wireframe or solid mode depending on the last redraw command used.








Top View - Starts simulation of the entire source program from its Top View. Redraw will be done in wireframe or solid mode depending on the last redraw command set.

Rotate



Click on this option to set a new view to display your graphic. Your cursor will immediately change to a “white hand”  icon (cursor needs to be inside viewport). At this point, click and hold the left button of your mouse and drag it to different locations to rotate your graphical representation to the view you want. Release mouse button to start simulation.



The simulation (or redraw) will be done in wireframe mode if your last view was a part view , tool view , slinky mode , or isometric view . The simulation will be done in solid mode if the last view was a solid view .

Snapshot (with WinLauncher)



With a Snapshot (bitmap file), G-ZERO WinLauncher displays a graphic of the source file that is selected from the Launch File dialog box. In this way, you can see a graphical image of the source file selected before you decide whether you really want to open it or not.

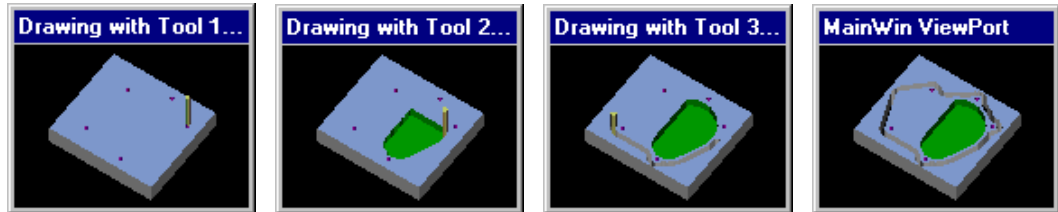
Note that your WinLauncher needs to be setup to use the “incremental search” method rather than the traditional Windows 95, 98 search. To do this, select the “Setup” hypertext from the WinLauncher main menu, click on the “Options” tab, and select the “Incremental Search” method in the “File Dialog” section.

To take a Snapshot (Viewport of G-ZERO Mill), select the Snapshot  option from the  pull-down menu in the viewport Toolbar. Once you click on this option, G-ZERO redraws your image in order to take a good Snapshot and it creates a Bitmap file using the same source name and extension BMP. This bitmap file is placed in the same directory as your source file. (*Note: For best results, set your computer color palette greater than 256 colors*)

Solid Verification F6

Simulation

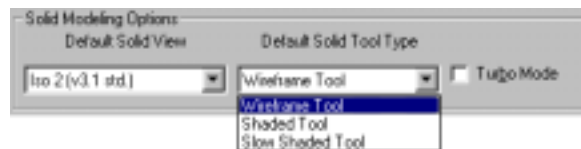
With solid modeling displayed on your viewport, you can watch a simulation of your machining while you are programming. Any time you press F6, G-ZERO Mill v4.0 will instruct the simulation program to start an animated representation of your entire source code.



Tool display

The tool can be shown in **wireframe**, **solid shaded**, and **slow solid shaded** modes.

To make any of these changes, select the appropriate option in the *Default Solid Tool Type* section of *Tools / Options / View* screen.



Tool type

If you are using special tools and you want to show accurate solid modeling of these tools, you can include the special commands `##TL`, `##TT`, and `##TA` to control the length, taper, and angle of the tool shown.

Example 1:

TOOL 7 dia.25 flutes1 type30=C'SINK rad0 *** Deburr pocket with 1/4" csink ##TA=45

Example 2:

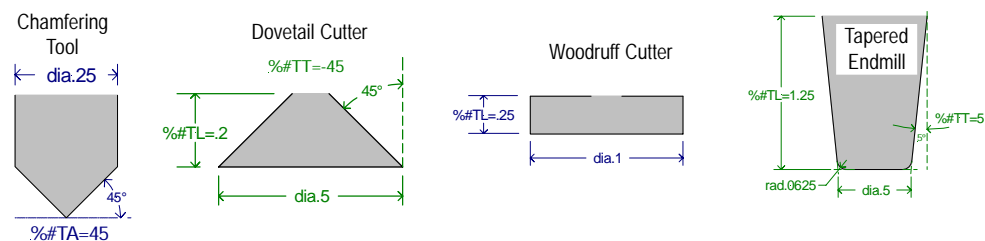
TOOL 1 dia.5 flutes4 type1=CARBIDE MILL rad0 *** 45 DEG DOVETAIL ##TL=.2 ##TT=-45

Example 3:

TOOL 2 dia1 flutes12 type1=CARBIDE MILL rad0 *** 3/16 THK WOODRUFF CUTTER ##TL=.1875

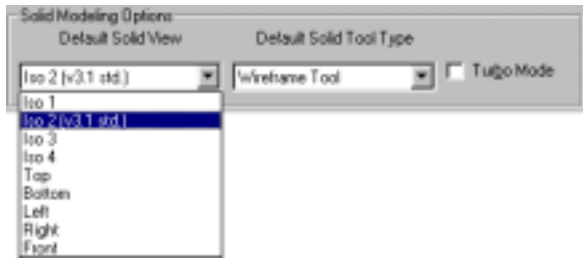
Example 4:

TOOL 6 dia.5 flutes4 type1=CARBIDE MILL rad.0625 *** ##TL=1.25 ##TT=5



Solid View

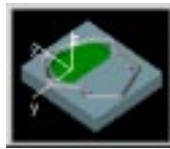
There are nine preset Solid Views that can be set in the *Default Solid View* section of *Tools / Options / View* screen.



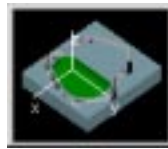
Iso 1



Iso 2 (v3.1 std.)



Iso 3



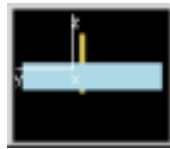
Iso 4



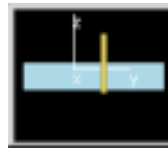
Top



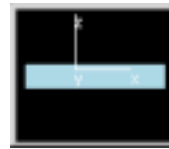
Bottom




Left

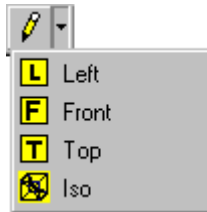


Right



Front

A quick way to display four of these views is done by selecting them from the Redraw option  in the viewport's toolbar: Left, Front, Top, Isometric. (Tool is displayed in wireframe or solid mode depending on what was last redrawn)



Turbo Mode

Place a check mark in the *Turbo Mode* option (*Solid Modeling Options* section of *Tools / Options / View*) to increase the speed of the creation of the solid representation. Turbo Mode works in the background to generate the solid view (without displaying cutting tool animation); so, you may notice a pause while G-ZERO is doing its calculations.

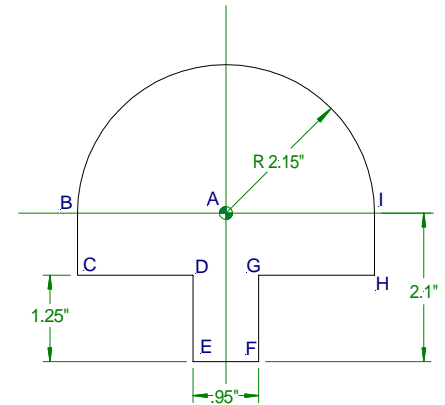
Input Math

We usually program our G-ZERO source codes entering values relative to the point of origin (0;0). On the other hand, most blueprints show dimensions in a different way, making it a little harder to get the values you need for G-ZERO directly from your blueprint. This is the reason why G-ZERO developers included this input method through Math Calculation.

For example, if we want to mill the profile of the symmetrical part shown below, we need to know the coordinates of 9 points (A through I). Using point A as our point of origin (where X=0 and Y=0), our source code will need the relative coordinates of the rest of the points (X and Y distance between point A and the rest of the points).

Using the dimensions given in the graphic, we can calculate the relative coordinates of points B through I as follows:

	X	Y
A	0	0
B	-2.15	0
C	-2.15	-2.1+1.25
D	-.95/2	-2.1+1.25
E	-.95/2	-2.1
F	.95/2	-2.1
G	.95/2	-2.1+1.25
H	+2.15	-2.1+1.25
I	+2.15	0



Operations and symbols

These are the symbols you can use when programming:

- + plus: addition operation
- minus: subtraction operation
- * asterisk: multiplication operation
- / forward slash: division operation
- \ back slash: integer division operation
- ^ caret: exponential (power) operation
- (open parenthesis: symbol used to indicate the start of a group
-) close parenthesis: symbol used to indicate the end of a group
- = equal: symbol used to display the result of the calculation before entering the value with the Enter key
- @ at: symbol used to make the default value available as the first number in the calculation. Example: if you just drilled a hole at x=1.23 and you now want to drill a hole that is over .5506 from the last hole, then enter @+.5506 (The key "@" saves you from typing "1.23" in the calculation)
- right arrow: 0°
- ↑ up arrow: 90°
- ← left arrow: 180°
- ↓ down arrow: 270°
- ' apostrophe: converts degrees in dd.mmss format to decimals. Example: if you need to enter 35°10'30" in decimal degrees, type 35.1030' and G-ZERO will make the conversion: 35.175
- “ quote: converts degrees in dd.mmss format to decimal degrees. (Same as apostrophe symbol)

Hierarchy

G-ZERO Mill v4.0 uses the same symbol hierarchy used in Algebra:

- 1st priority: () *G-ZERO allows unlimited number of nested parenthesis*
 2nd priority: * / \
 3rd priority: + -

Examples:

$$5.4 - 1.88 / 2 = 5.4 - 0.94 = 4.46$$


$$5.4 - (1.88 / 2) = 5.4 - 0.94 = 4.46$$

$$(5.4 - 1.88) / 2 = 3.52 / 2 = 1.76$$

$$2 + 3^2 - 5 / 2 = 2 + (3^2) - (5 / 2) = 2 + 9 - 2.5 = 8.5$$








$$9 \setminus 2 = \text{INT}(9 / 2) = 4$$

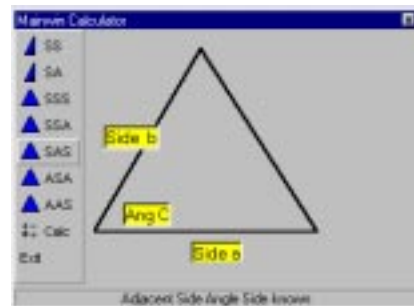
Mainwin Calculator (Triangle)

G-ZERO Mill v4.0 comes with a calculator to assist you calculate unknowns in triangles, as well as trigonometry and basic algebra. To open the Mainwin Calculator, select  Triangle from the Tools pull-down menu, or use the keyboard shortcut Ctrl+T.

Triangle functions

Mainwin Calculator offers seven situations for you to calculate unknown values in triangles:

- | | |
|---|-----------------------------------|
|  | ◆ 2 sides known |
|  | ◆ 1 side, 1 angle known |
|  | ◆ 3 sides known |
|  | ◆ adjacent side-side-angle known |
|  | ◆ adjacent side-angle-side known |
|  | ◆ adjacent angle-side-angle known |
|  | ◆ adjacent angle-angle-side known |



Just enter 2 or 3 values of a triangle (sides or angles), and the calculator will give you the values for the rest of the sides and/or angles. You can use the drag-and-drop method to select any one of these results and drop it on the dialog bar of G-ZERO application window and use it as the answer to your programming question.

Math functions

Functions available are: ABS, SIN, COS, TAN, ATAN, LOG, SQRT, INT, FRAC, CIEL, FLOOR, LOG10, EXP.

Use the bottom Edit bar to type your Math function. After you press the Enter key, the result will appear in the same Edit bar. Use the drag-and-drop method to select the result and drop it on the dialog bar of G-ZERO application window to use it as the answer to your programming question.



Math functions need to be typed in upper case; they work in radians and the results are also given in radians (not decimal).

Use Math functions in Mainwin Calculator only; do not use them directly in the dialog bar of G-ZERO application window.

Math operators

- Operators available are:
- | | | | |
|---|----------------|-----|-----------|
| + | plus | > | greater |
| - | minus | < | smaller |
| * | multiply | <=> | not equal |
| / | divide | & | and |
| \ | integer divide | | or |

Results of all math calculations can be dragged and dropped into the dialog bar of G-ZERO application window to use it as the answer to your programming question.

Special Keys

Positioning

Page Up	Up one screen
Page Down	Down one screen
Up Arrow	Up one line
Down Arrow	Down one line
Home	To beginning of source program
End	To end of source program
Ctrl+L	Goto a specific line (key in line number)
Ctrl+F	Find
Ctrl+S	Find next

Programming

Esc	Undo last source line. Undo last value entered if programming a command
O	(Letter O). Override: forces G-ZERO to accept values out of normal range
Any Letter(s)	Comment
%	Machine readable comment or post command (key in ONLY machine readable comments after the % symbol to avoid an alarm on your machine). <i>See %Command section.</i>
#	Calls data or program from another file
F1 or ?	Help on command currently programming. Main Help menu if you are not in the middle of programming a command

Tools

Ctrl+T	Mainwin calculator (Triangle)
F1	Help
F8	Sketch/Machine (<i>Option</i>)
F9	CAD Reader
F11	Mouse Rough
F12	Post Process

Viewport

F2	Part view
F3	Tool view
F4	Slinky mode
F5	Iso view
F6	Solid view
F7	Show/Hide viewport

Special Words

Some words entered as Comments perform specific functions. These reserved words are:

COLOR

Type the word COLOR followed by a number to change the color of your graphics from the current line until the the next Tool or COLOR comment. This is quite helpful when a tool will be used for several different layers of your part.

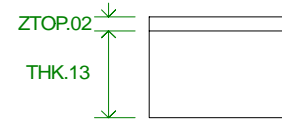
Example: COLOR 1 will draw in blue.

0 Black	4 Red	8 Gray	12 Light Red
1 Blue	5 Magenta	9 Lilght Blue	13 Light Magenta
2 Green	6 Brown	10 Light Green	14 Yellow
3 Cyan	7 White	11 Light Cyan	15 Hi-intensity White

ZTOP

Enter a ZTOP comment line right before the MAT'L command to show material above the *top of your part* (usually Z0).

Example: We are going to flycut (0.02") off the material. In order to show in the viewport 0.02" of material to be cut, we need to enter ZTOP.02 right before the MAT'L command.



```
1          ZTOP.02
2  MAT'L   xmin0 xmax4 ymin0 ymax3 thk.13 type0=ALUMALOY
```

Note: Only solid view (F6) shows ZTOP in the viewport.

% Commands

- %Machine readable comment or post command (key in ONLY machine readable comments after the % symbol to avoid an alarm on your machine).
- %+Continue comment from previous line, or add G-code to line above.

General

- %(*comment*)Send man-readable comment to machine. Example: %(CARBIDE ENDMILL)
- %OSet program number (place before the first Tool). Example: %O0234 sets program number to be “0234”
- %STARTSUBSet subprogram starting number - override location of SUB.
Example: %STARTSUB 3000 places SUB in the 3000 area of memory to avoid collisions (post processors for most Fanuc-/Yasnac-type controllers put SUBs in the 6000, 6500 and 7000 area)
- %N_ S_Output whole line without processing. Example: %N25 S20 (CNC sees this, G-ZERO does not)
- %OFFFollowing source lines are for profile definition only.
- %ONCancels %OFF.

Tool change

Note: %Anything can be programmed in the TOOL *** comment area.

- %TC X_ Y_Set tool change location for all tools. Example: %TC X12.1 Y6.6 forces tool change at location x=12.1 and y=6.6 (machines normally change tools wherever the last cut ended).
- %TCXSet var #69 in UNIM.
- %TCYSet var #70 in UNIM.
- %TCZSet var #71 in UNIM.
- %M0Set var #56 in UNIM.
- %S0Set var #55 in UNIM.

Feedrate

- %FOverride G-ZERO automatic feedrate selection from that point in the source program to the next tool change. Example: %F10 sets the feed at 10 inches per minute.
%F99.99 unlocks a feed and lets G-ZERO start its automatic feedrate selection again.
- %F-Force deceleration for radii (percentage of feedrate). Example: %F-25 slows down 25% around circles.
%F-0 shuts off radius deceleration.
- %FMAXSet maximum allowable feedrate. Example: %FMAX25 limits feedrate to 25 ipm.
- %CHIPOverride feedrate and give it in chipload. Example: %CHIP.001 calculates feedrate from .001 chipload.

Spindle speed

- %SOverride G-ZERO automatic spindle speed selection. Example: %S2500 sets the spindle at 2500 rounds per minute.
- %SMAXSet maximum allowable rpm for all tools. Program before first tool. Example: %SMAX1200 limits spindle to 1200 rpm.

Coolant

`%COOL OFF` Turn coolant off.
`%COOL ON` Turn coolant back on again.
`%COOLIN` Set coolin flag: var #30.
`%COOLHI` Set var #28.

Viewport display (F6 - Solid mode only)

In TOOL comment only (See Solid Verification section):

`##TL` Set tool length. Example: `##TL=1.5` sets tool length to 1.5 inches.
`##TT` Set tool taper. Example: `##TT=5` sets tool taper to 5°.
`##TA` Set tool angle. Example: `##TA=45` sets tool angle to 45°.

Post

`##POST` Resets .P file variables. Example 1: `##POST(21)=1` turns on metric.
Example 2: `##POST(52)=1` turns on tracing.
`##ARG` Reset post global variables. Example: `##ARG(101)=HI MOM` resets tool comment string.
`%GOALF` Tell FADAL post to skip canned lettering cycle.
`%JSHIFT` Set var #53. It is usually used in G10 areas of post. Used to base G54 offsets at 55, 56, etc.
`%PALLET` Set var #35.

Math

`%POLAR ON` Switch to polar coordinate system, where XY points become R (radius), THETA (degrees).
`%POLAR OFF` ... Turn off polar coordinate system.

Stepover

`%STEP` Set stepover amount or percentage for RECT and ROUND.
Example 1: `%STEP65` steps 65%
Example 2: `%STEP.002` steps 0.002 inches.

Rapid

`%ZBUFF` Set the distance between zrap and top of part for MILL passes calculation.
Example: `%ZBUFF.05` tells passes how close to rapid: 0.05".
`%WSAFE` Set var #60.
`%ZSAFE` Set var #43. it is used in B area for safe rapid up or down.

Station

Note: Station option is sold separately. See Station manual.

`%ORDER` Replace MULTIPLY usage with STATION to link stations.
Example: `%ORDER 1A,2B,3A,4B`
`#+ORDER` Continuation of %ORDER command just above it.
`%BCOPY` Tell station to copy on other side(s) of stone. Example: `%BCOPY2` to copy on 2 sides of stone.

Wrap

Note: Wrap option is sold separately. See Wrap manual.

- %WRAP Normal wrap. Example: %WRAP Y RAD1.3 STP.05 ZC0 wraps around the Y axis, being the cylinder radius 1.3” and its center at Z=0, and tool step=.05”.
- %WRAP CONTROL ... Wrap control: sets wrap control curve to define part wall for simultaneous XYZA motion. Example: %WRAP CONTROL STP.05 wraps around the X axis with a tool step over of 0.05”. Program right before control curve definition.
- %WRAP LOCKED Wrap locked: locks 4th axis at a certain degree to project onto cylinder without 4th axis motion. Example: %WRAP RAD1.2 STP.05 LOCK120 wraps around the X axis locking the 4th axis at 120°.
- %WRAP OFF Cancel Wrap settings.

