


G-ZERO Lathe

Reference - Basics



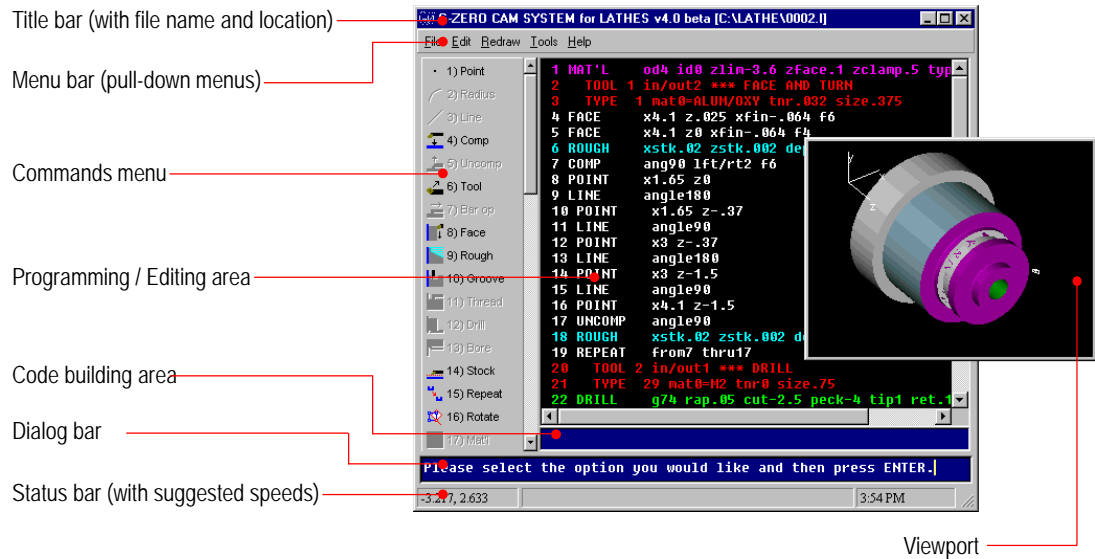
Contents

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	7
Dialog Boxes (from Tools Options)	8
Preferences	9
Command Menu	9
Color and Font	9
G-code Text Editor	9
Metric / Inch System	9
File Management	10
Open	10
Incremental search - WinLauncher	10
Save	10
Save As	10
Backup	10
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
Tool Library	16
FEEDS.EXE	16
Create a New .F File	16
Modify an Existing .F File	16
Tool Picture	17
TOOL GEOMETRY	18
REDUCE FEEDRATE (%)	18
FEEDRATE CONTROL (IPR)	18
Entering Feeds & Speeds	19
CLING-ON	20

Viewport	21
System	21
Clear Screen	22
Copy To ClipBoard	22
Print View	22
Zoom	22
Redraw	22
SnapShot (with WinLauncher)	23
Solid Verification 	24
Simulation	24
Tool Display	24
Part Display	24
Input Math	25
Operations and symbols	25
Hierarchy	26
Mainwin Calculator (Triangle)	26
Special Keys	27
Positioning	27
Programming	27
Tools	27
Viewport	27
Special Words	28
COLOR	28
% Commands	28
General	28
Tool change	28
Feedrate	28
Spindle speed	29
Coolant	29
Post	29
Blockskip	29

Screen Layout

This section of the manual is intended to overview the components of G-ZERO Conversational CAM System for Lathes 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 the 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.

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 application window and 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).

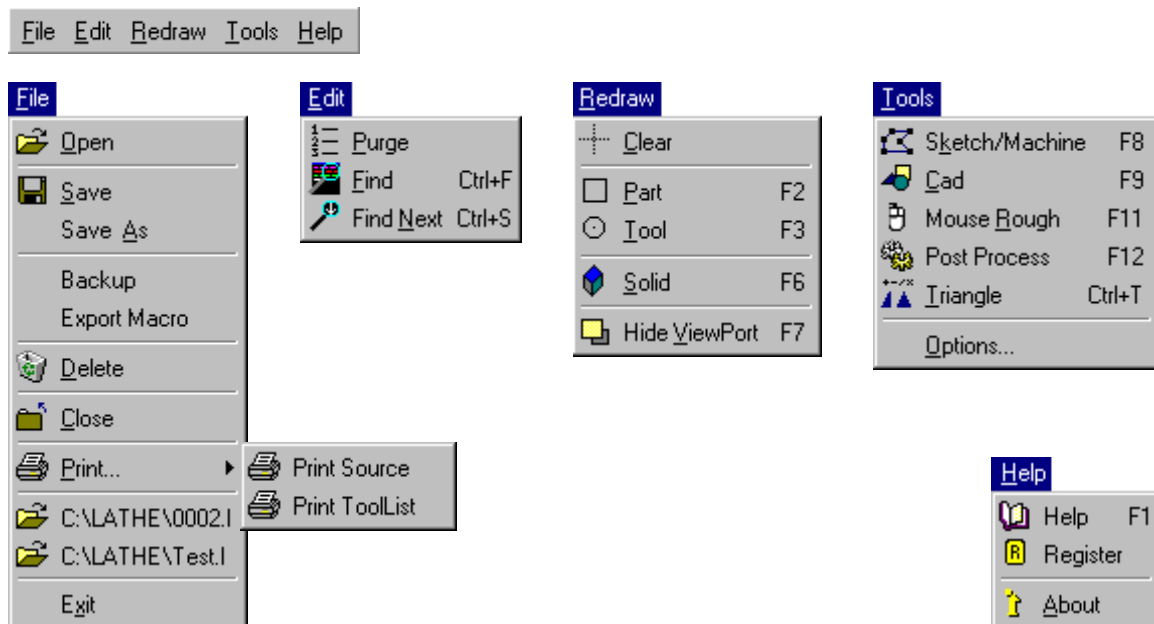
When the information on the programming area needs a larger display area, a vertical and/or horizontal scroll bar(s) will appear at the side and/or bottom of this section. Use your mouse to click the scroll arrows (at the end of the scroll bars) or click and drag the scroll box to move around the programming 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 Lathe currently has 39 programming commands (#18 and #38 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.

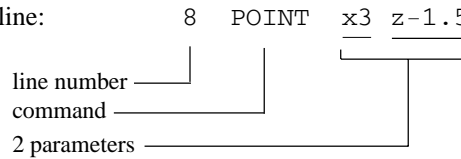
G-ZERO Lathe normally displays the full command menu (icons and name); however, you have the option of displaying just the icons or no commands at all. The advantage of reducing the visibility of the commands menu is that it frees monitor space to increase the programming area. To adjust the size of 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? 3
```

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, 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 Z? -1.5
```

Again, the default value is displayed and G-ZERO waits for your reply.

After you press the Enter key to the the last answer, 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. This is the area where you can find:

4.781, -2.041 . mouse cursor coordinates (only when mouse cursor is within the viewport display)

1:52PM current time (taken from computer clock)

Note for Lathe v3.x users: Suggested SFM and RPM are now displayed on pop-up boxes.

Viewport

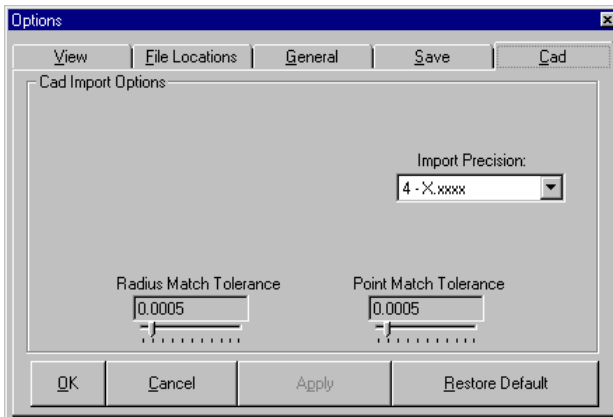
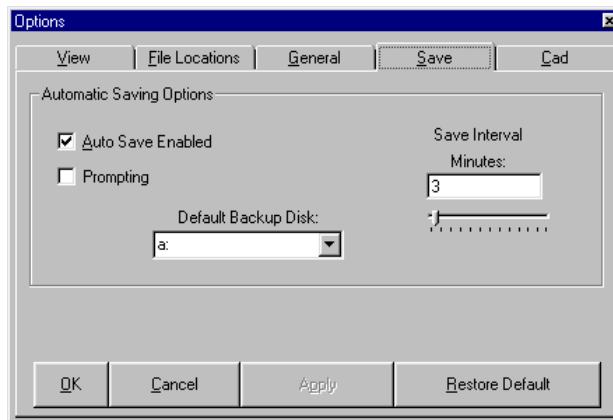
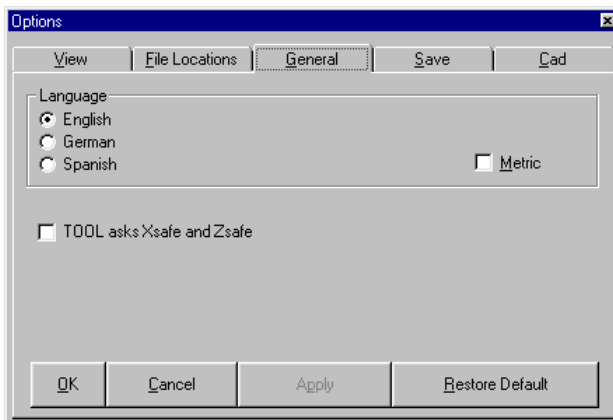
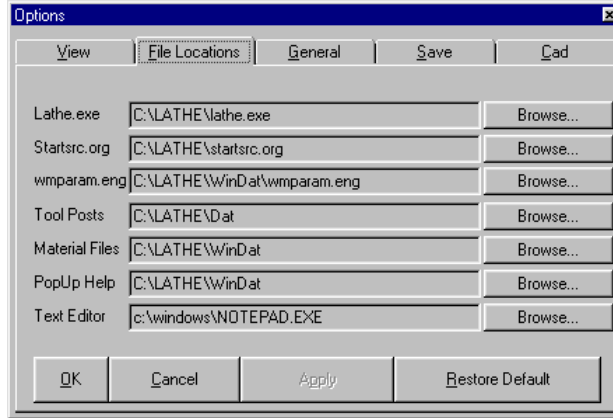
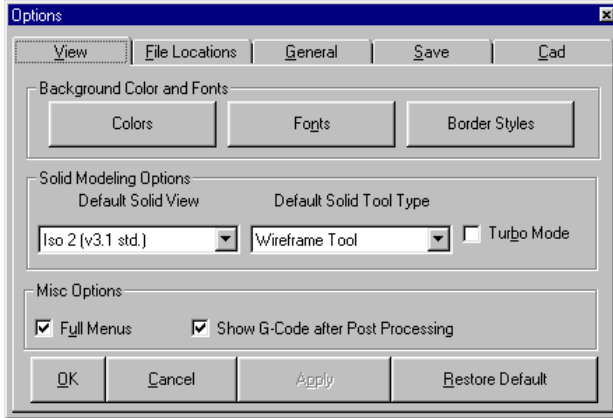
G-ZERO uses the viewport to display the graphical representation of the current source file in various formats (e.g.: solid view, wireframe, wireframe with tool path, etc.)

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 Locations* 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).

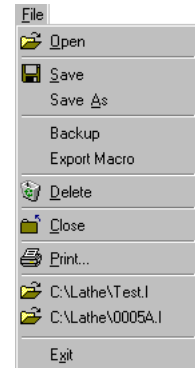
File Management

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 Lathe source files (extensions .l)

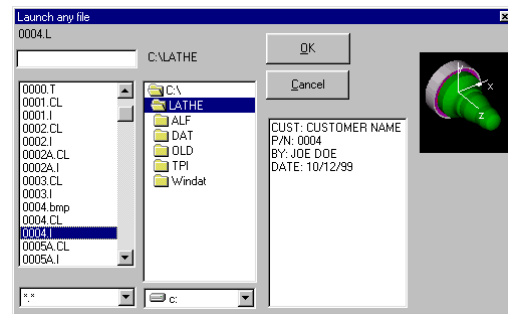
Open an existing G-code file by selecting it from the *Source File to Open* window (File | Open). Make sure that the lower section of this window (file type) is set to display G-code files (extensions .t). G-code files are opened with the default text editor. To change the default text editor, go to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

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.

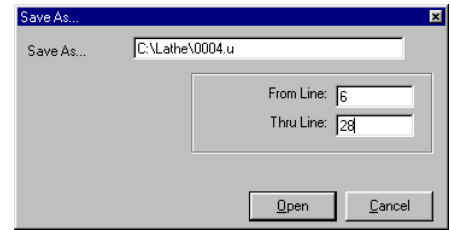
To change the default backup disk drive (A:), make the appropriate selection in the *Save* screen from Tools | Options.

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 (**F**ile | **E**xport 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 (**F**ile | **D**elete).

Close

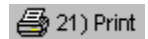
Close the current source file by selecting the *Close* option from the *File* pull-down menu (**F**ile | **C**lose). You can also click on the X button in the upper right corner of the application window (Title bar).

Print

Print entire source file

File Menu. Send the current file directly to the default printer by selecting the *Print Source* option from the *File / Print* pull-down menu (**F**ile | **P**rint | **P**rint Source).

21) Print. Use the 21) Print command and print from line 1 thru last line of source program.



Print selected source lines

21) Print. 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 toolsheet

File Menu. Print the tool list (all source lines with the TOOL command) by selecting the *Print Tool List* option from the *File / Print* pull-down menu (**F**ile | **P**rint | **P**rint ToolList).

21) Print. Call the 21) Print command and print from line **-1** to line **0**.

NOTE: 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.

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 to indicate tool number and whether it's going to be an ID or OD cutting. After answering these questions, G-ZERO automatically calls the Type command to get detail information about the tool you are using such as tool type, material, tool nose radius, and size.
- Third program a cutting operation such as facing (Face, FGroove), grooving (Groove), threading (Thread), turning (Point, Radius, Line), or milling (Mill). Once again, just answer the questions G-ZERO has for you about your cutting operation.

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, F3, or 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 MAT'L   od8 id0 zlin-3.9 z
>>2 TOOL 1 in/out2 *** FACE 8
>>6 MAT'L   od8 id0 zlin-3.9 z
>>8 TOOL 1 in/out2 *** ROUGH
>>37 TOOL 2 in/out1 *** DRILL
>>40 TOOL 3 in/out1 *** ROUGH
```

```
>>1 MAT'L   od8 id0 zlin-3.9 z
>>2 TOOL 1 in/out2 *** FACE 8
>>3 TYPE 1 mat1-TIN tr-.032
>>4 FACE   x8.05 z-.02 xfin-.0
>>5 BORE   x8 z-.1 zdepth-1 f5
>>6 MAT'L   od8 id0 zlin-3.9 z
>>8 TOOL 1 in/out2 *** ROUGH
>>37 TOOL 2 in/out1 *** DRILL
>>40 TOOL 3 in/out1 *** ROUGH
```

```
1 MAT'L   od8 id0 zlin-3.9 z
2 TOOL 1 in/out2 *** FACE 8
3 TYPE 1 mat1-TIN tr-.032
4 FACE   x8.05 z-.02 xfin-.0
5 BORE   x8 z-.1 zdepth-1 f5
6 MAT'L   od8 id0 zlin-3.9 z
7 FLIP PART TO MACHIN
8 TOOL 1 in/out2 *** ROUGH
9 TYPE 1 mat1-TIN tr-.032
10 ROUGH  xstk.01 zstk.005
11 COMP  ang90 lft/rt2 f5
12 POINT  x4.9 z0
13 LINE  angle90
```

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.

```
25 LINE      angle180
26 POINT    x7.8 z-3
27 LINE      angle (161.5651)
28 POINT    x8 z-3.3
29 UNCOMP   angle180
30 ROUGH    xstk.01 zstk.005
31          NOW TAKE A PARALLE
32 ROUGH    xstk.005 zstk.002
33 COMP     ang0 lft/rt1 f4
34 REPEAT   from28 thru12
35 UNCOMP   angle270
36 ROUGH    xstk.005 zstk.002
37 TOOL 2  in/out1 *** DRILL
```

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 → 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 → 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 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)
- ◆ **Select Post.** Select a tool post (for example: MAZAK) and watch how G-ZERO creates the right G-code for the machine selected. Posts (.p files) are usually saved in the Lathe\Dat directory. However, if you have them stored in a different directory, used the *Look in:* section of the *Tool Post Selection* window to locate your posts.

G-ZERO saves the G-code file (extension .t) just generated 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:

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 LATHE\WINDAT directory. Some default pop-up boxes are:

File name	Command	Question #	Contents
17.6	17) MAT'L	6	Type of materials (e.g.: 4=ALUMINUM)
36.1	36) TYPE	1	Type of tools (e.g.: 1=80-degree-diamond)
36.2	36) TYPE	2	Tool material (e.g.: 1=TIN)
36.4	36) TYPE	4	Tool size (e.g.: 0.375)
8.6	8) FACE	6	Finish chart
9.7	9) ROUGH	7	Feed comparison chart

Edit Pop-up Boxes

If the standard pop-up boxes do not contain exactly what you need, the information can be changed. 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 36) TYPE. Use Windows Notepad to open the file C:\LATHE\WINDAT\36.4 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 5th question of command 11) THREAD. Delete the file 11.5 located in C:\LATHE\WINDAT.

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:\LATHE\WINDAT directory (assuming your Lathe 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:\LATHE\WINDAT\15.1

Tool Library

The Tool Library is a set of files located in the LATHE\WINDAT subdirectory. Their file names always start with either IDTOOL or ODTOOL followed by a number (Tool Type #), and the file extension is always .F. Example: IDTOOL2.F

Tool library is maintained by the FEEDS.EXE file. Each .F has five sections to store the details of one tool:

1. Macro to draw tool holder picture
2. Macro to draw tool picture
3. Tool geometry
4. Reduce feed rate (%)
5. Feed rate control (IPR)

FEEDS.EXE

Create a New .F File

1. At the DOS prompt, in the LATHE directory (Example: C:\LATHE>), type FEEDS to start the Feeds program.
2. The Feeds program will ask “What speed file would you like?” Type the name of the NEW file you want to create (Example: IDTOOL22.F)
3. The Feeds program will ask “What working .F file most closely resembles this new one?”
4. Use the arrow keys or the mouse to pick a file from the list shown.
5. The Feeds program will copy the information of the existing file into your new file. Now, you are ready to modify the new file to accommodate the information for your new tool. (See steps 4-5 below)

Modify an Existing .F File

1. At the DOS prompt (Example: C:\LATHE>), type FEEDS to start the Feeds program.
2. The Feeds program will ask “What speed file would you like?” Use the arrow keys or the mouse to pick the file you want to modify.
3. Use the arrow keys or the mouse to pick the file you want to modify.
4. Your cursor is the little green box. (When the cursor is at the end of a line, it contains a left arrow) When your cursor is inside a chart or box, anything typed will typeover the current character.
5. When done, press the **Esc** key. If you made any changes, the program will ask if you want to Save the changes. Type Y for Yes and N for No.

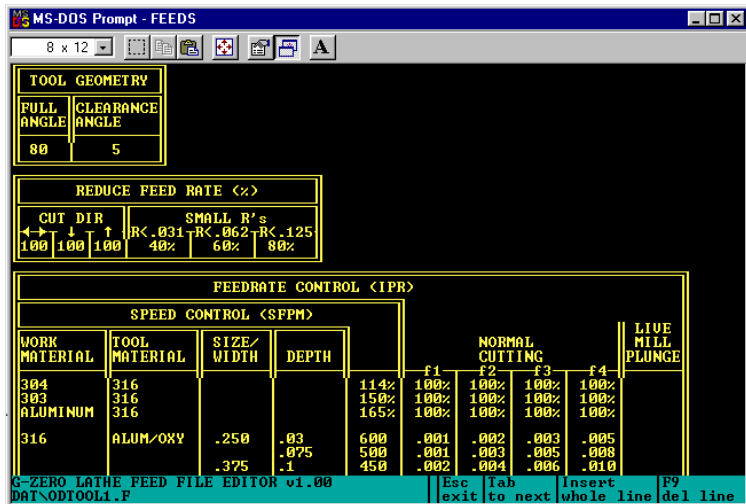
<Enter> Moves cursor to beginning of next line.

<Insert> Inserts a new line at cursor. Use this to add a new line to a Macro, or to the FEED & SPEED chart.


<F9> Deletes line of characters at the cursor. (Do not delete the chart or lines separating columns in the charts).

<Tab> When filling in or modifying the 3 bottom charts, <Tab> will move your cursor to the next value automatically.

<Backspace> Deletes a character to the left of the cursor.



Tool Picture

Every .F file starts with two macros that have the details needed to display the shape of the tool when the TOOL command is programmed and when the  is pressed (Redraw with tool).

Note: For details on Macros, see the Macro section of this manual.

Macro to Draw Tool Holder Picture

Here is an example:

```

37,0
0,COLOR 3
0,DEFAULT TNR=.08 SIZE=1
16,0,-#1-#1,#1
1,0,0
1,-.0875*#2*1.2*2,#2*1.2
1,-.0875*#2*1.2*2,#2*8
1,-#2*1.2*2,#2*8
1,-#2*1.2*2,.0875*#2*1.2
1,0,0
37,0
0,COLOR 9
1,-.0875*#2*1.2*2,#2*1.2
1,-#2*1.2*2,#2*1.2
16,0,0,0
    
```

Program a 37) Mill every time you want to stop drawing a line and begin a new line in your picture of a TOOL/ INSERT or HOLDER.

Specify the color of the HOLDER or INSERT using the macro comment COLOR.

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

Macro to Draw Tool Insert Picture

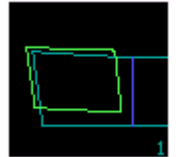
Here is an example:

```

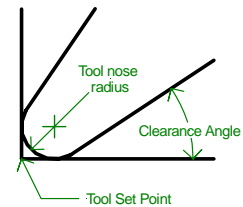
37,0
0,COLOR 10
0,DEFAULT TNR=.031 SIZE=1
16,-85,-#1-#1,#1
4,135,1
2,#1,1,0,0
2,#1,1,-#1+#2+#2,.14*#2
2,#1,1,-#1+#2+#2,1.07*#2
2,#1,1,0,.93*#2
2,#1,1,0,0
5,135
16,0,0,0
    
```

Variables used for correctly scale the size of the tool: **TNR, SIZE**.

Variables to shift (translate) the drawing of the tool to its correct position in the Tool Library display: **XOFFSET, ZOFFSET**.



16) ROTATE command adds a tool clearance angle.



Use 4)COMP and 5)UNCOMP to describe the geometry.

For INSERTS, be sure to unRotate with 16,0,0,0.

TOOL GEOMETRY

Fill in the full angle and the clearance angle of your insert.

These values should match the geometry programmed in the Tool Holder Picture Macro and the Tool Insert Picture Macro above this chart.

TOOL GEOMETRY	
FULL ANGLE	CLEARANCE ANGLE
80	5

REDUCE FEEDRATE (%)

CUT DIR

With some cutting tools, you may wish to slow down when cutting in certain directions (**CUT DIR**). Normally the tool will feed at 100%.

FEEDRATE OVERRIDE					
CUT DIR			SMALL R's		
←→	↓	↑	R<.031	R<.062	R<.125
100	100	100	40%	60%	80%

- If you would like the tool to slow down 25% (75% of normal) when cutting away from or towards the chuck, then enter **75%** in the box just below the left/right arrows.
- If you would like the tool to slow down 25% (75% of normal) when cutting upward, then enter **75%** in the box just below the up arrow.
- If you would like the tool to slow down 25% (75% of normal) when cutting downward, then enter **75%** in the box just below the down arrow.

SMALL RADS

With finish tools, you may wish to slow down cutting around small outside arcs.

If you make very small parts and you would like all arcs smaller than .003 to be cut at 30% of normal, then change the first **R<** to **R<.003**. Next, move down to the box just below **R<.003** and enter **30%**.

Remember that the small R's values must go from smaller (left) to bigger (right).

Example: **R<.003 R<.033 R<.016**
 Wrong! Small to big to smaller.
 R<.003 R<.016 R<.033
 OK - Small to big to bigger.

FEEDRATE CONTROL (IPR)

This chart controls all feeds and speeds for this cutter type.

PART MATERIAL (e.g.: 316)

Using 8 or less letters or numbers, enter the name or abbreviated name of a **PART MATERIAL** (or Work Material). It will exactly match one of the names in PART MATERIAL LIST (pop-up box 17.6)

TOOL MATERIAL (e.g.: ALUM/OXY)

Using 8 or less letters or numbers, enter the name or abbreviated name of a **TOOL MATERIAL**. If this tool is available in a range of materials, type just one **TOOL MATERIAL** here. (The next **TOOL MATERIAL** for this same **PART MATERIAL** goes on the next line down directly below this value.

STAINLESS STEELS
0=304
1=303
2=316
3=430
OTHERS
4=ALUMINUM
5=1018
6=4140

SIZE/WIDTH

Type in the size of this tool: insert: inscribed circle
 grooving tool: width
 drill or mill: diameter

If this tool is available in many sizes, enter the sizes in ascending order, one size per line. Type the next largest size directly below this answer.

FEEDRATE CONTROL <IPR>									
SPEED CONTROL <SFPM>					NORMAL CUTTING				LIVE MILL PLUNGE
WORK MATERIAL	TOOL MATERIAL	SIZE/WIDTH	DEPTH		f1	f2	f3	f4	
304	316			114%	100%	100%	100%	100%	
303	316			150%	100%	100%	100%	100%	
ALUMINUM	316			180%	100%	100%	100%	100%	
430	316			130%	100%	100%	100%	100%	
1018	316			160%	100%	100%	100%	100%	
4140	316			110%	100%	100%	100%	100%	
316	ALUM/OKV	.250	.03	600	.001	.002	.003	.005	
			.075	500	.001	.003	.005	.008	
		.375	.1	450	.002	.004	.006	.010	
			.2	400	.002	.004	.006	.010	
		.500	.1	500	.002	.005	.009	.015	
			.2	400	.003	.007	.012	.020	
	TIN	.250	.03	450	.001	.002	.003	.005	
			.075	350	.001	.003	.005	.007	
		.375	.1	375	.002	.004	.006	.008	
			.2	300	.002	.004	.006	.010	
		.500	.1	400	.002	.005	.008	.010	
			.2	350	.003	.007	.011	.018	
	C-5	.250	.05	250	.001	.002	.003	.005	
		.375	.1	250	.003	.005	.007	.010	
		.500	.125	250	.003	.005	.008	.010	
	CQ-2	.250	.05	250	.001	.002	.004	.006	
		.375	.1	250	.003	.005	.007	.010	
		.500	.125	250	.003	.005	.007	.010	

DOC/ROUGH (DEPTH)

If this tool is never used for ROUGHing, skip this (leave this column blank).

If this tool is used for ROUGHing, you may type in several depth of cut values (DOC) that correspond to different SFM (surface feet/minute) values.

Type in the smallest DOC value (for this insert) now. Type in the next DOC value directly below. The next DOC value goes below this value etc.

PEED CONTROL (SFM)

Type in the surface feet/minute for this tool keeping in mind: TOOL MATERIAL, PART MATERIAL and SIZE (and DOC if a roughing insert). If you don't know the SFM, ask your tooling salesman for a recommended value.

NORMAL CUTTING

If this tool is never used for ROUGHing, GROOVing or MILLing, skip this.

These four columns are feedrates for different load levels ranging from **f1** (lightest) to **f4** (heaviest). Type in the feedrate for each load. **HINT:** The lighter the load, the smaller (slower) the feedrate.

LIVE MILL PLUNGE

Use this when milling with Live Tool Option only. If you don't have live tooling, skip this.

Entering Feeds & Speeds

If this is a **ROUGHing** tool and you want to enter another DOC for this same tool, move your cursor to the next line down and fill in these columns with the correct values:

DOC/ROUGH
SPEED CONTROL
NORMAL CUTTING

If there is **another** (bigger) tool for this same PART MATERIAL, move your cursor to the next line down and fill in these columns with the correct values:

TOOL MATERIAL
SIZE/WIDTH
DOC/ROUGH
SPEED CONTROL
NORMAL CUTTING

To program a new PART MATERIAL, see *CLING-ON* or move your cursor to the next line down and fill in these columns with the correct values for the new PART MATERIAL:

- PART MATERIAL
- TOOL MATERIAL
- SIZE/WIDTH
- DOC/ROUGH
- SPEED CONTROL
- NORMAL CUTTING

CLING-ON

How to use the same TOOL data with a different PART MATERIAL

Once you have input all the numbers for 316 Stainless, you may want to use them again for 304 Stainless. Instead of typing all that information again for each insert material and size, you can just put one line at the top of the **FEED & SPEED Chart**. That line is called a **CLING-ON**.

You can cling-on to other materials that are programmed further down the chart. Instead of IPR or SFM values, you just put % values in the chart.

To duplicate (cling-on) exactly to another material, just put 100% in the SFM column and in the f1 through f4 columns.

PART MATERIAL

Type in the name or abbreviated name of the NEW material. This name must match exactly the PART MATERIAL LIST.

TOOL MATERIAL

Type in the **PART MATERIAL** to be referenced.

Ex: To have 304 use the values of 316, type **316** as the **TOOL MATERIAL**. G-ZERO will then use the modified values of 316 automatically.

SIZE/WIDTH

Skip this!

DOC/ROUGH

Skip this!

SPEED CONTROL (SFM)

Type in the percentage of the referenced material's surface feet/minute.

Ex: To keep the same spindle speed, put **100%** in the SFM column. To double the spindle speed, put **200%**.

NORMAL CUTTING

Type in the percentage of the referenced material's loads (f1, f2, f3, f4) for roughing tools.

Ex: To keep the same load ratio, put **100%** in each column. To double each load, put **200%** in each column.

PART	TOOL	SIZE	DOC	NORMAL CUTTING			
MAT'L	MAT'L	WIDTH	ROUGH	SFM	F1	F2	F3 F4
PMAT-1	TMAT-1	SIZE 1		X			
		SIZE 2		X+			
		SIZE 3		X++			
	TMAT-2	SIZE 1	.075	Y	SLOW-->---	---	FAST
			.095	Y+	SLOW-->---	---	FAST
			.115	Y++	SLOW-->---	---	FAST
		SIZE 2	.125	Z	SLOW-->---	---	FAST
			.135	Z+	SLOW-->---	---	FAST
			.150	Z++	SLOW-->---	---	FAST
		SIZE 3		B	SLOW-->---	---	FAST
	TMAT-3	SIZE 1		A			
	TMAT-4	SIZE 1		C	SLOW-->---	---	FAST
PMAT-2	TMAT-1	SIZE 1		D			
		SIZE 2		D+			
	TMAT-2	SIZE 1	.075	F	SLOW-->---	---	FAST
		SIZE 2	.125	G	SLOW-->---	---	FAST
			.135	G+	SLOW-->---	---	FAST
			.148	G++	SLOW-->---	---	FAST
		SIZE 3		J	SLOW-->---	---	FAST
	TMAT-3	SIZE 1		K	SLOW-->---	---	FAST

This is a generic idea of what a chart might look like.

LIVE MILL PLUNGE

Skip this!


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.

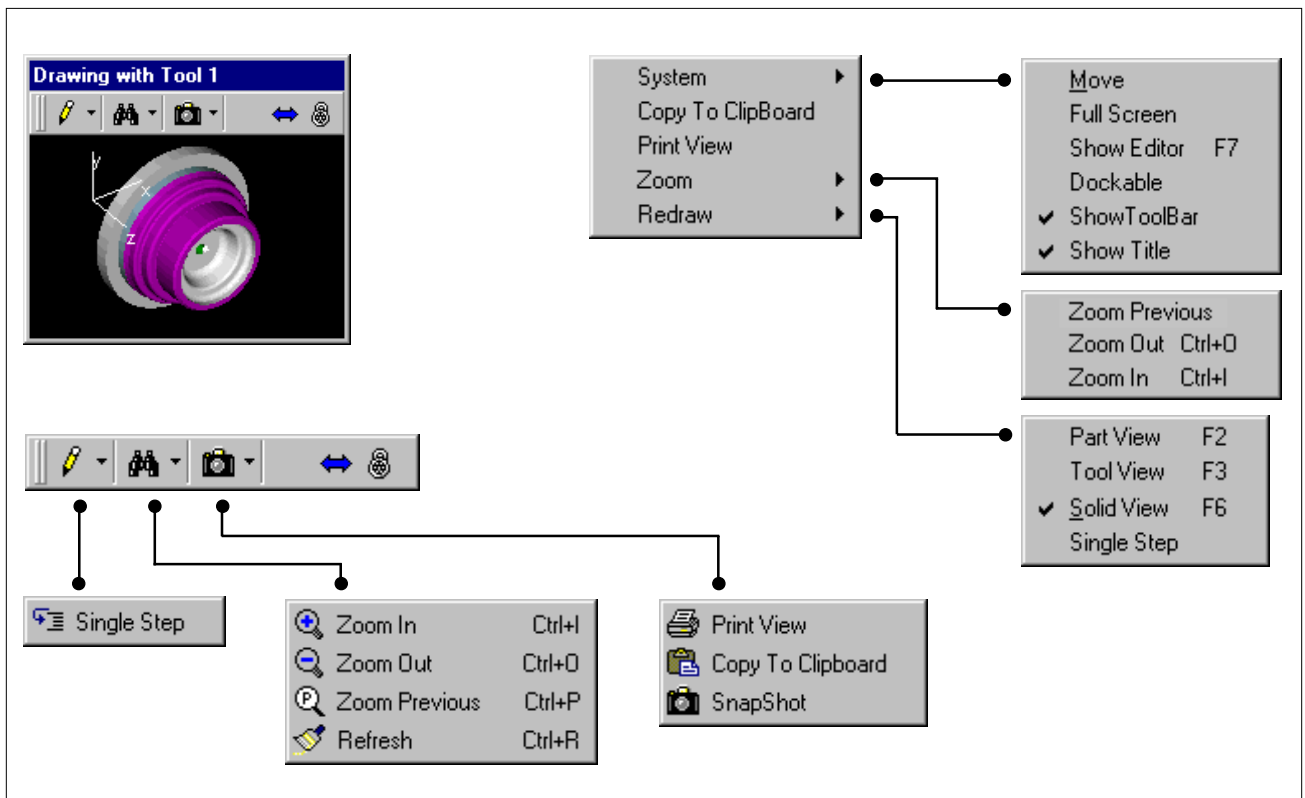


Dockable - If this option is selected, the viewport will be docked to the upper right corner of the source code programming screen. To undock the viewport, unselect the *Dockable* option from the *System* list. Docking or undocking the viewport can also be done by clicking the *Dockable* icon from the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

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 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.



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.



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

F2

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

F3

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

F6

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.



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.

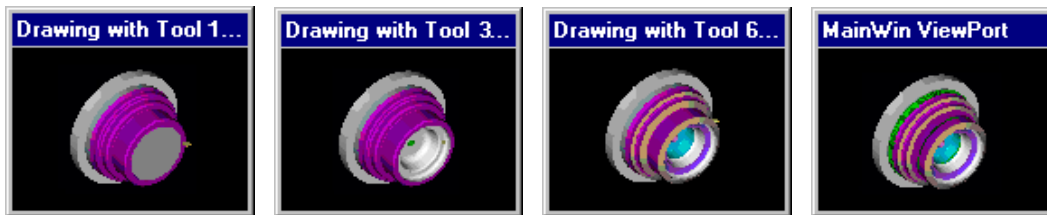
Note that your WinLauncher needs to be setup to use the “incremental search” method rather than the traditional Windows 95, 98 search. To do this, select the “Setup” hypertext from the WinLauncher main menu, click on the “Options” tab, and select the “Incremental Search” method in the “File Dialog” section.

To take a SnapShot (Viewport of G-ZERO Lathe), 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 will instruct the simulation program to start an animated representation of your entire source code.



Tool display

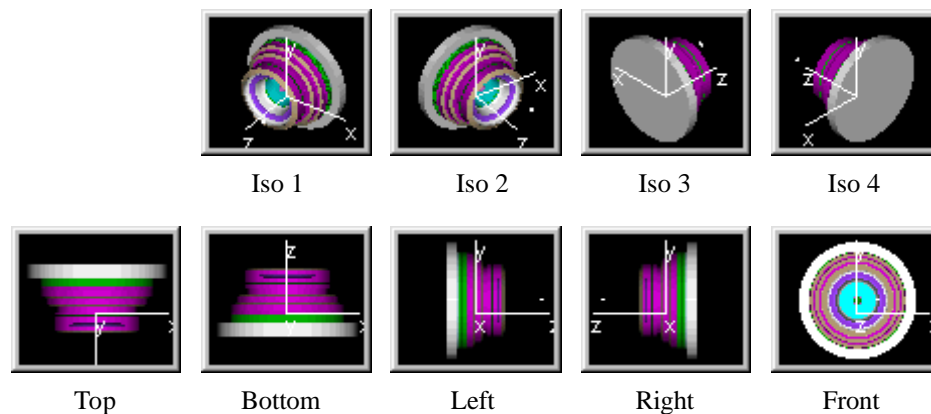
The tool can be shown in **wireframe** and **shaded** modes.

To set the mode, right-click on the viewport (picture area), and select Solid View Settings|Tool Display. Turbo mode may also be selected from this menu. This mode hides the tool so the finished part comes up in the shortest possible time.



Part Display

There are nine preset part orientations that can be set in the *Part Display* section of *Solid View Settings*. Right click on the viewport (picture area) to change this setting.



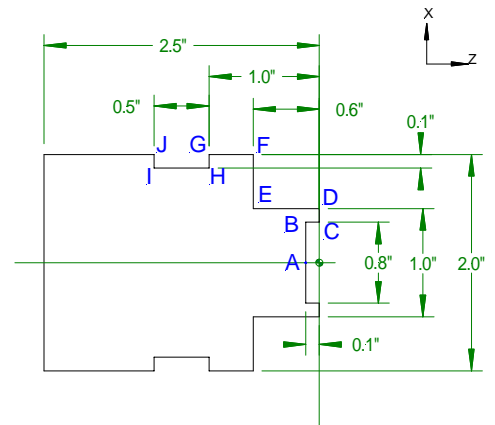
Input Math

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	Z
A	0	-.1
B	.8/2	-.1
C	.8/2	0
D	1/2	0
E	1/2	-.6
F	2/2	-.6
G	2/2	-1
H	(2/2) -.1	-1
I	(2/2) -.1	-1-.5
J	2/2	-1-.5



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 uses the same symbol hierarchy used in Algebra:

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

Examples:

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


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

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

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








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

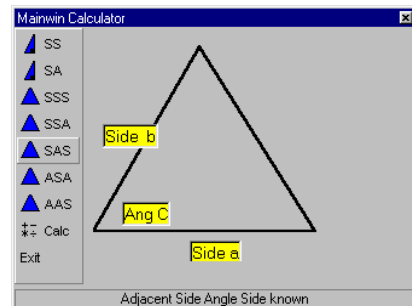
Mainwin Calculator (Triangle)

G-ZERO 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.

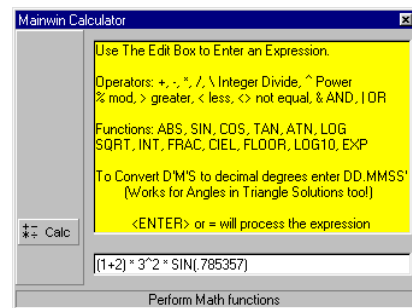
Math functions

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). *See %Command section.*
..... 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 (*Option*)
F9 CAD Reader
F11 Mouse Rough
F12 Post Process

Viewport

F2 Part view
F3 Tool 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

% 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"

%N_ S_Output whole line without processing. Example: %N25 S20 (CNC sees this, G-ZERO does not).

%OFFFollowing source lines are for definition only.

%ONCancels %OFF.

%CSets C-axis when Y-axis is active (livetool mode only)

Tool change

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

%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: %F.002 sets the feed at .002 inches per revolution.

%F99.99 unlocks a feed and lets G-ZERO start its automatic feedrate selection again.

Spindle speed

- `%S` Override G-ZERO automatic spindle speed selection. Example: `%S2500` sets the spindle at 2500 rounds per minute.
- `%SMAX` Set maximum allowable rpm for all tools. Program before first tool. Example: `%SMAX1200` limits spindle to 1200 rpm.
- `%s300` Example: `%s300` uses 300 surface feet per minute.

Coolant

- `%COOL OFF` Turn coolant off.
- `%COOL ON` Turn coolant back on again.
- `%COOLIN` Set coolin flag: var #26 in UNIL.

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.

Blockskip

- `%BS` Turns on blockskip “/” mode. (Also works in the *** comment section of the tool command)
- `%BS1` thru `%BS9` .. Generates “/1” thru “/9” for machines with multiple levels.
- `%BS OFF` Turns off blockskip mode.

