

G-ZERO Lathe

Interactive Tutorial



Before You Begin

This tutorial was designed using G-ZERO Lathe v4 installed with the default settings. Newer versions of G-ZERO will, of course, have a look that is slightly different, especially on some settings pages.

Note 1: If you do not yet own our full working version, the G-ZERO Student version is recommended for users who intend to self-train using this Lathe Tutorial Manual. You may download the Student version from our website by typing the following (exactly) into your web browser:

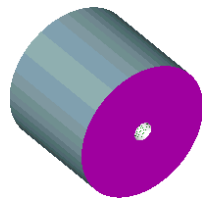
<http://www.g-zero.com/DownL/InstallLatheStudent.exe>

Note 2: All files used in this tutorial are on your C:\LATHE\TUTORIAL directory after installing the Student version.

In this manual, each project starts with a list of the main topics covered in that assignment along with a blueprint of the part to be programmed. Usually, there are many ways to program a part; here, we describe just one solution.

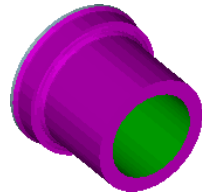
Projects in this tutorial follow a logical progression, so each project assumes you master the concepts presented in previous projects.

Contents



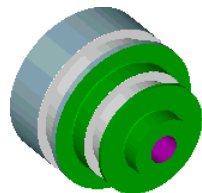
1. Initial Tour 5

Open G-ZERO Lathe. Start a new source file. Use basic commands. Use Help. Simple editing. View part. Save a source file. Convert a source file into G-code. Exit G-ZERO Lathe.



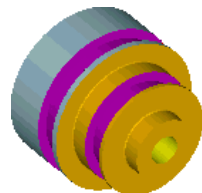
2. Bushing 13

Start a new source file. Define the material. Use the FACE and DRILL commands.



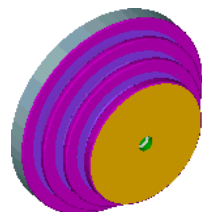
3. Hub 17

Rough and outside (OD) shape. Use COMP and UNCOMP to compensate offsets for cutter radius. Repeat source program lines with REPEAT command. Use of GROOVE and THREAD.



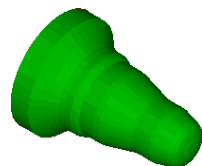
4. Hub (revisions) 23

Duplicate a source file using Save As. Print source file. Change value using ALTER command. Change value using mouse and keypad. Insert a new command. Move source lines. Delete source lines.



5. Pulley 27

Use RADIUS to define roughing contour. More on COMP and UNCOMP. Use of BORE command to enlarge previously drilled hole. ID Groove. Use of PECK before an OD Groove.



6. Punch 35

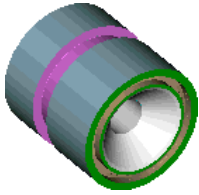
Program RADIUS after a COMP command. Use of various types of RADIUS (center, corner, unknown) to define roughing contour.



7. Housing 39

SIDE A: How to enter fractional dimensions. Use POINT to define sharp corner between 2 lines. Use DRILL to countersink a drilled hole. Use DEADTAP to tap a hole.

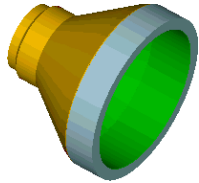
SIDE B: Angled face and finish OD. Use Face with ID tool.



8. Seal Collar

49

Convert radial degree to decimal degree. Use PIPE and THREAD commands for tapered threading. Use GROOVE command for OD grooving. Use FGROOVE for face grooving



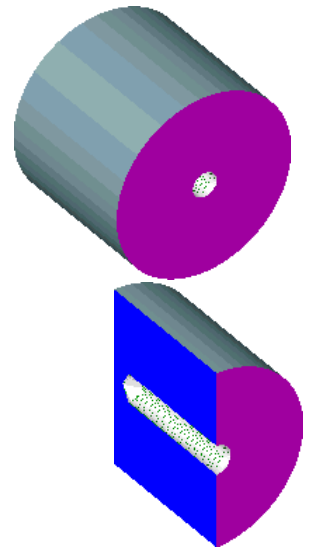
9. Funnel

55

Program unknown Line. Flip a part: multiple Material command, negative Tool number. Use of %OFF (disactivate G-code) and %ON (activate G-code). Use Rotate command. Use Line-Point-Line technique.

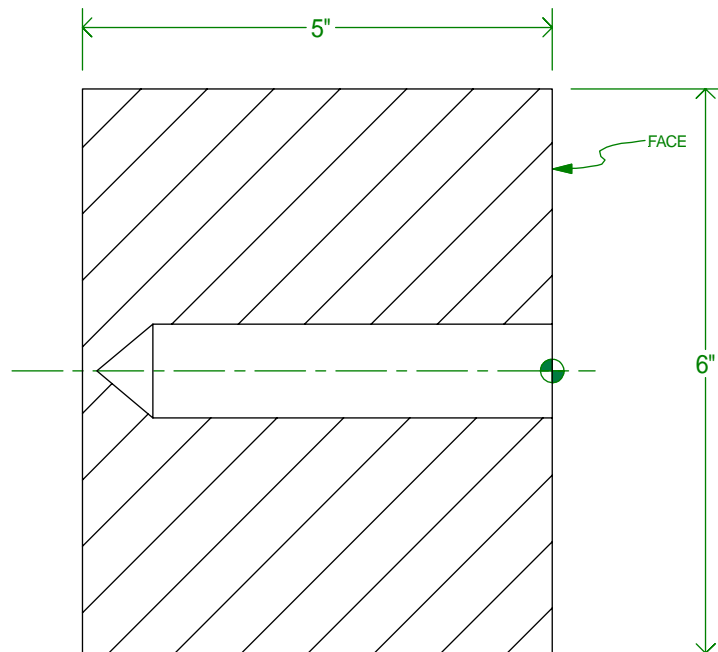
Project I


Initial Tour



What you will learn:

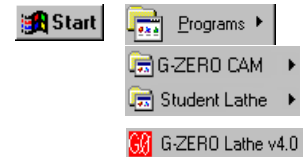
- ◆ Open G-ZERO Lathe
- ◆ Start a new source file
- ◆ Use basic commands
- ◆ Use Help
- ◆ Simple editing
- ◆ View part
- ◆ Save a source file
- ◆ Convert a source file into G-code
- ◆ Exit G-ZERO Lathe



 RAPID OUTPUT Co. 888-656-1945	TITLE: Initial Tour	NOTES: 1. All measurements in inches (not to scale) 2. Material amount in chuck = .5"	RECOMMENDED OP & TOOL LIST: 1. Face & Turn OD (.032" trr x .5" ic) (CNMG 432 - 80° diamond insert) 2. Drill (3/8" dia.)
	MATERIAL: Aluminum		
	DRAWING: L0000		

1 Start G-ZERO Lathe

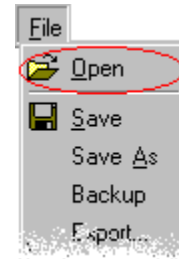
- ♦ From the **Start** menu, choose **(All) Programs**.
- ♦ From the **G-ZERO CAM \ Lathe** folder, select **G-ZERO Lathe v4**.



2 Start a new source file

- ♦ From the File pull-down menu, select Open to open the “Source File to Open” window.
- ♦ Type **Initial Tour** in the “File Name” box, and click the **Open** button. (Note that G-ZERO Lathe accepts long file names)

Because the file does not exist, G-ZERO asks you if you want to create it. Click the **Yes** button.





3 Use basic commands

Start your source code with a Material command that describes the dimensions and type of material of the part you are going to work on.

Then, you need to describe the Tool you are going to use, and follow it by a cutting operation (e.g.: face, drill, or thread)

You can invoke a command in two different ways:

- Use your mouse to select the command from the left graphic (or full) menu. Example: 
- Use the numeric key pad to key in the number that corresponds to the specific command you want to enter, followed by the **Enter** key. Example: 

To answer the questions that correspond to each command, key in the value and/or comment requested followed by the **Enter** key.

*Note 1: Any error you type can be immediately undone by pressing the Backspace key (Esc key will undo the last value entered)
After a command has been completely programmed, press the Esc key to undo the last source line.*

*Note 2: G-ZERO checks every value you key in to verify it is in the normal range of values. If the value is out of this range, it will beep and will not accept it. **However, if you are sure that the value you entered is correct, overwrite G-ZERO's validation to enforce your input by pressing the letter O key before the Enter key.***

Note 3: Each section below starts with source codes followed by their corresponding descriptions or comments. For a detailed explanation of each command, see the G-ZERO Lathe Reference Manual.

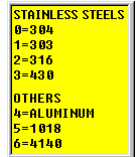
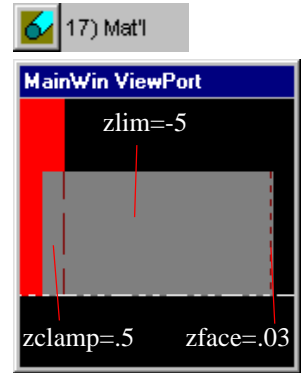
1 MAT'L od6 id0 zlim-5 zface.03 zclamp.5 type4=ALUMINUM

Select the command **17) Mat'l** and complete this command by answering questions about the material.

WHAT IS THE MAXIMUM DIAMETER OF THIS PART? **6**
WHAT IS THE MINIMUM DIAMETER OF THIS PART? (often 0) **0**
WHAT IS THE FURTHEST Z OF THIS PART? (usually minus) **-5**
HOW MUCH EXTRA MATERIAL WILL YOUR PART FACE FIRST HAVE? **.03**
HOW MUCH EXTRA MATERIAL WILL BE CLAMPED IN THE CHUCK? **.5**
WHAT IS THE PART MADE OF? **4**

G-ZERO helps you answer some questions (such as in the case of “type of material”) by popping up a yellow box with options that can be selected with your mouse.

After entering the answer for the last question, G-ZERO adds the command line to your source program (in this case, the first line of your source program) and displays a graphical representation of the part material in the ViewPort.



2 TOOL 1 in/out2 *** FACE TOOL

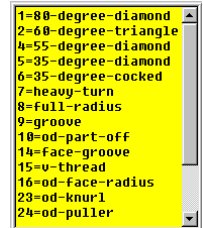
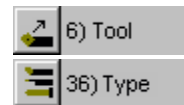
3 TYPE 1 mat1=TIN trn.032 size.375

Select **6) Tool** to program the first tool and answer its questions with these values:

WHICH TOOL STATION WOULD YOU LIKE? **1**
IS THIS TOOL SET FOR: 1) ID or 2) OD CUTTING? **2**
Describe what you will be doing with this tool (comment): **FACE TOOL**

Command **36) Type** is automatically invoked by G-ZERO to continue describing the tool: (*again, you can choose responses from the yellow pop-up boxes*)

WHAT TYPE OF TOOL IS THIS? **1**
WHAT IS THE TOOL (insert) MADE OF? **1** (*for TIN*)
WHAT IS THE TOOL'S NOSE RADIUS: (0 for drills and mills)? **.032**
HOW BIG IS THIS TOOL: (ic, dia, width, tpi)? **.375**

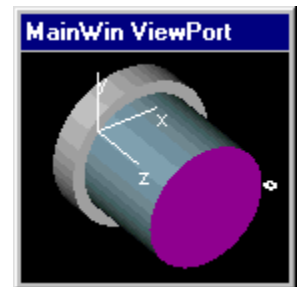
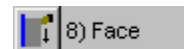


4 FACE x6.1 z.1 xcut-.04 zcut0 depth0 f.007

Program the **8) Face** command.

RAPID TO WHAT SAFE X DIMENSION? **6.1** (*press "O" to override if it beeps*)
START FACING AT WHAT Z DIMENSION? **.1**
FEED TO WHAT X DIMENSION? (usually minus to go past cl) **-.04**
CUT TO WHAT FINAL Z? (usually 0) **0**
DEPTH OF CUT PER PASS? (0 ofr single pass) **0**
WHAT FEEDRATE (ipr) WOULD YOU LIKE? **.007**

Note that after answering all the questions to this command, G-ZERO displays the actual cut in the viewport. Press *F6* to view the cut again in solid view.



5 TOOL 2 in/out1 *** DRILL

6 TYPE 29 mat0=M2 trn0 size1

7 DRILL g74 rap.1 cut-4.25 peck-4 tip1 ret.1 f.0105

Define the tool for this second operation (drilling) programming the **6) Tool** and **36) Type** commands.

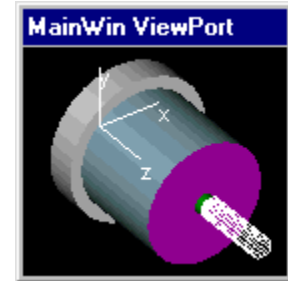
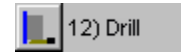
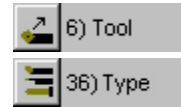


WHICH TOOL STATION WOULD YOU LIKE? **2**
 IS THIS TOOL SET FOR: 1) ID or 2) OD CUTTING? **1**
 Describe what you will be doing with this tool (comment): **DRILL**
 Command **36) Type** is automatically invoked by G-ZERO to continue describing the tool:

WHAT TYPE OF TOOL IS THIS? **29**
 WHAT IS THE TOOL (insert) MADE OF? **0** (for M2)
 WHAT IS THE TOOL'S NOSE RADIUS: (0 for drills and mills)? **0**
 HOW BIG IS THIS TOOL: (ic, dia, width, tpi)? **1**

Program the 12) Drill command.

WHAT G-CODE WOULD YOU LIKE TO DRILL WITH? **74**
 START FEEDING FROM (rapid to) WHAT DIMENSION? **.1**
 DRILL TO WHAT FINAL DIMENSION? **-4.25**
 PECK HOW MANY TIMES? (- for full retract) **-4**
 WOULD YOU LIKE THE DRILL TIP ADDED? (1=yes, 0=no) **1**
 RETRACT TO WHAT DIMENSION? (usually = rap) **.1**
 WHAT FEEDRATE (ipr) WOULD YOU LIKE? **.0105**



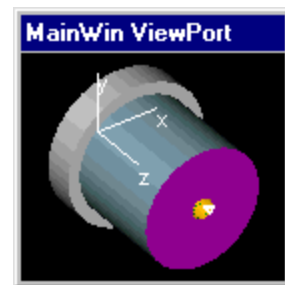
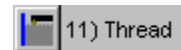
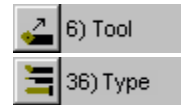
```
8 TOOL 3 in/out1 *** THREAD
9 TYPE 15 mat0=TIN tnr0 size8
10 THREAD maj1.125 tpi8 min.971637 S1528 z.25 zfin-2 num10 a60
```

Program the **6) Tool** and **36) Type** commands to define the threading operation.

WHICH TOOL STATION WOULD YOU LIKE? **3**
 IS THIS TOOL SET FOR: 1) ID or 2) OD CUTTING? **1**
 Describe what you will be doing with this tool (comment): **THREAD**
 WHAT TYPE OF TOOL IS THIS? **15**
 WHAT IS THE TOOL (insert) MADE OF? **0** (for TIN)
 WHAT IS THE TOOL'S NOSE RADIUS: (0 for drills and mills)? **0**
 HOW BIG IS THIS TOOL: (ic, dia, width, tpi)? **8**

With the **11) Thread** command, G-ZERO rapids to the startpoint of the thread then feeds the specific number of times to the finish point (at a feedrate appropriate for the TPI).

WHAT IS THE MAJOR DIAMETER OF THE THREAD? **1.125**
 HOW MANY THREADS PER INCH? **8**
 WHAT IS THE MINOR DIAMETER OF THE THREAD? **.9716375**
 WHAT RPM WOULD YOU LIKE? **1528**
 WHERE, IN Z WOULD YOU LIKE TO START? **.25**
 WHERE, IN Z DOES THE THREAD END? **-2**
 HOW MANY PASSES WOULD YOU LIKE TO MAKE? **10**
 WHAT ANGLE WOULD YOU LIKE FOR THE INFEED? (usually 60) **60**




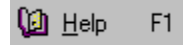
By now, your source code should look like this:

```
1 MAT'L od6 id0 zlim-5 zface.03 zclamp.5 type4=ALUMINUM
2 TOOL 1 in/out2 *** FACE TOOL
3 TYPE 1 mat1=TIN tnr.032 size.375
4 FACE x6.1 z.1 xcut-.04 zcut0 depth0 f.007
5 TOOL 2 in/out1 *** DRILL
6 TYPE 29 mat0=M2 tnr0 size1
7 DRILL g74 rap.1 cut-4.25 peck-4 tip1 ret.1 f.0105
8 TOOL 3 in/out1 *** THREAD
9 TYPE 15 mat0=M2 tnr0 size8
10 THREAD maj1.125tpi8min.971637 S1528z.25zfin-2num10a60
```


4 Use Help

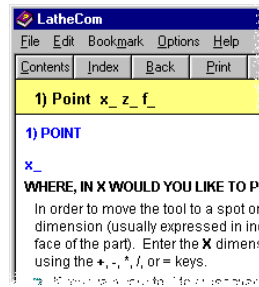
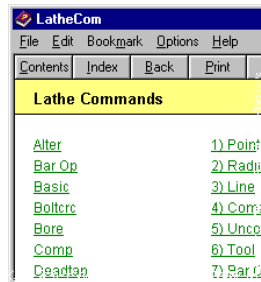
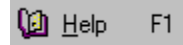
G-ZERO Lathe comes with a built-in Help system with descriptions of the commands along with some programming tips. Let's try it.

- Press the `'` function key to open the Lathe Commands Help.
(Click the green underlined commands to view their respective descriptions)
- Click the [close icon](#)  on the upper right corner of the screen to exit the Help window.



If you are in the middle of creating a command line, the `'` key will display the description of that specific command. To test it, let's create a new line with the command POINT.

- Select the [POINT](#) command.
- Now, that we are in the middle of this command, press the [F1](#) function key. Notice that it displays the descriptions of the POINT command.
- Click the [close icon](#)  on the upper right corner of the screen to exit the Help window.
- Press the [Esc](#) key to undo the unfinished POINT command.



5 Simple editing

Change a value: Let's say we want to change the tool's nose radius (tnr) value of the tool in line 6 from 0 to .015.

- Use your mouse to [click the value you want to change](#).
Notice that the value you selected is highlighted and the working screen background turned to blue. This indicates that you are in the "editing" mode.
- Type in `.015` to replace the highlighted value.
- Press the [Enter](#) key to accept the new value and exit the "editing" mode.

```
1 MAT'L   od6 id0 zlim-5 zface.03 zc
2 TOOL 1  in/out2 *** FACE TOOL
3 TYPE   1 mat1=TIN tnr.032 size.375
4 FACE   x6.1 z.1 xcut-.04 zcut0 dep
5 TOOL 2  in/out1 *** DRILL
6 TYPE   29 mat0=M2 tnr0 size1
7 DRILL  g74 rap.1 cut-4.25 peck-4
8 TOOL 3  in/out1 *** THREAD
9 TYPE   15 mat0=TIN tnr0 size8
10 THREAD maj1.125tpi8min.971637 S1
```

Replace a value with another value on the screen: Now, we are going to change the tool's nose radius (tnr) value of the tool in line 6 back to 0 by replacing it with the tnr value of the tool in line 9.




- Use your mouse to [click the value you want to replace](#) (tnr value in line 6). The screen turns into "editing" mode.
- Use your mouse to [click the tnr value \(0\) of the tool in line 9](#). Notice that the tnr value of line 6 has been replaced.
- Press the [Enter](#) key to accept new value and exit the "editing" mode.

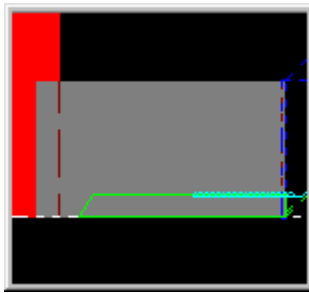
Delete last line: To delete the last line of your source code, just press the [Esc](#) key.

6 View / redraw the part

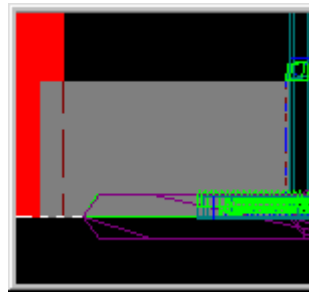
While you are programming, you may want to look at a graphical representation of your source code. Let's try it now.

- ◆ Press **F2** to redraw the entire source program.
- ◆ Press **F3** to redraw the entire source program with the tool path.
- ◆ Press **F6** to display a solid view of the part.

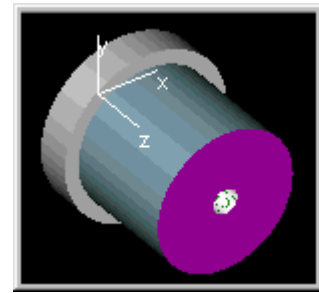
	Part	F2
	Tool	F3
	Solid	F6



F2 - Part



F3 - Tool



F6 - Solid

Partial redraw: G-ZERO gives you the option of collapsing a complete tool section. Let's try redrawing just Tool 1.

- ◆ Click the space on the left side of the number "2" of Line 2 to collapse all tools. [Fig. 1] (**Ctrl** + **←**) also collapses the source program)
- ◆ Click the **>>** symbol left to the number "2" of Line 2 to show details of Tool 1. [Fig. 2, 3]
- ◆ Now press all the function keys previously introduced (F2 - F6) and see that only Tool 1 is shown.
- ◆ To see whole program again, press **Ctrl** + **→** keys.

```
1 MAT'L    od6 id
2 TOOL 1 in/out
3 TYPE 1 mat1
4 FACE    x6.1 z
5 TOOL 2 in/out
6 TYPE 29 mat6
7 DRILL   g74 r
8 TOOL 3 in/out
9 TYPE 15 mat6
10 THREAD  maj1
```

Fig. 1

```
>>1 MAT'L    od6 id
>>2 TOOL 1 in/out
>>3 TYPE 1 mat1
>>8 TOOL 3 in/out
```

Fig. 2

```
>>1 MAT'L    od6 id
-2 TOOL 1 in/out
-3 TYPE 1 mat1
-4 FACE    x6.1 z
>>5 TOOL 2 in/out
>>8 TOOL 3 in/out
```

Fig. 3

Zoom: Zoom a section of your graphic to magnify its details. Follow the example below:

- ◆ Press **F2** to redraw the entire source program. [Fig. 4]
- ◆ **Place your mouse cursor** on the upper left corner of the area to be zoomed.
- ◆ **Hold the left button** of your mouse and **drag it** to frame the area to be zoomed. [Fig. 5]
- ◆ **Release mouse** and see how the framed area is zoomed in. [Fig. 6]

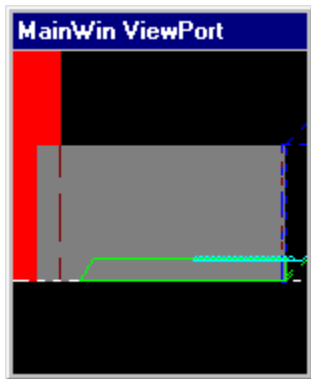


Fig. 4

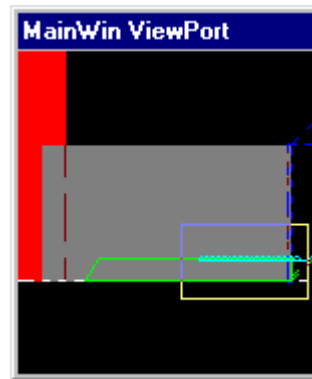


Fig. 5

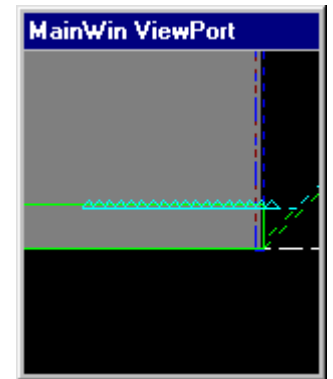
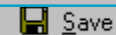


Fig. 6

7 Save a source file

It is a good idea to frequently save your program to avoid losing information. Let's do that now:



- ◆ From the **File** pull-down menu, select **Save**.

By default, G-ZERO Lathe automatically saves your file every 3 minutes.

Note: To save your file using another name, use Save As.

Note: Student version only saves the first 60 lines of your program.

WARNING: Never load a source program from your full working version into the Student version. Data will almost always be lost.

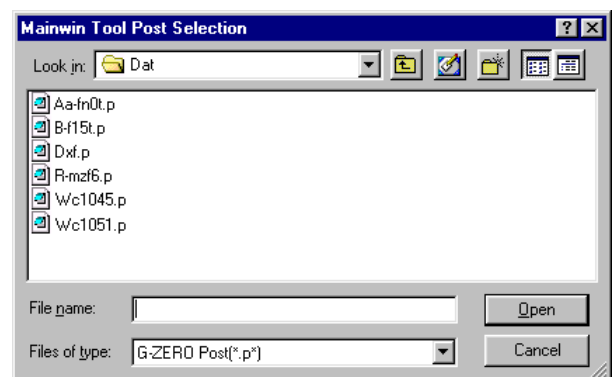
8 Convert a source file into G-code

Note: G-ZERO Student version generates Sample G-code.

F12: Press F12 to open the Tool Post Selection window. (Note that post processors are customized and sold separately)

Post: Select a tool post and watch how G-ZERO creates the appropriate G-code for the machine selected. G-ZERO also saves the G-code file (extension .t) and displays it on your screen using Notepad.

To exit Notepad, click the close icon on the upper right corner of its window.



[TOOL 1 in/out2 *** FACE TOOL	Z-1.47
[TYPE 1 mat1=TIN tnr.032 size.375	G1 Z-2.586
axes0=XZ	G0 Z.1
[.62:37s	Z-2.576
[G1 Z-3.586
[TOOL 2 in/out1 *** DRILL	G0 Z.1
[TYPE 29 mat0=ALUM/OXY tnr0 size1	Z-3.576
axes0=XZ	G1 Z-4.58
[1.72:103s	G0 Z.1
[G0 X10. Z9.75 M9
[TOOL 3 in/out1 *** THREAD	T0200
[TYPE 15 mat0=ALUM/OXY tnr0 size8	M1
axes0=XZ	
[.26:15s	(THREAD)
[M8
[2.6	N3 G50
[C:\LATHE\DAT\B-F15T.P	G54
%	G0 T0303
O0001	G97 S1500 M3
(FACE TOOL)	X.9316 Z.25 M8
M8	G92 X.9716 Z-2. F.125
N1 G50 S1500	X1.0036
G54	X1.0316
G0 T0101	X1.0516
G97 S387 M3	X1.0676
X6.1 Z.1 M8	X1.0796
G96 S619	X1.0916
G75 X-.04 Z0 I3.07 K.1 D100 F.007	X1.0996
G0 X10. Z9.75 M9	X1.1076
T0100	X1.1156
M1	X1.1206
	X1.123
(DRILL)	X1.124
M8	X1.125
N2 G50	G0 X.9316 Z.25
G54	G0 X10. Z9.75 M9
G0 T0202	T0300
G97 S275 M3	M1
X0 Z.1 M8	M30
G1 Z-1.48 F.01	%
G0 Z.1	

Initial-Tour.t

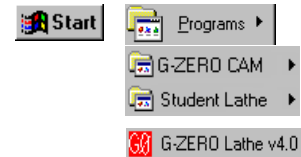
Note: Lines before the first % symbol are for your eyes only and are not sent to the CNC.

Exit G-ZERO Lathe

To exit G-ZERO Lathe, click the [close icon](#)  on the upper right corner of its window. ■

1 Start a new source file

- ◆ From the **Start** menu, choose **(All) Programs**.
- ◆ From the **G-ZERO CAM \ Lathe** folder, select **G-ZERO Lathe v4**.
- ◆ From the **File** pull-down menu, select **Open** to open the “Source File to Open” window.
- ◆ Type **Bushing** in the “File Name” box, and click the **Open** button.
- ◆ Because the file doesn’t exist, G-ZERO asks if you want to create it. Click the **Yes** button.



2 Define the material

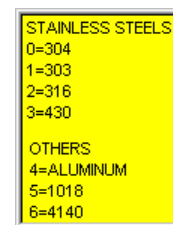
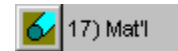
1 **MAT'L** od.625 id0 zlim-.52 zface.02 zclamp.1 type1=303

MAT'L tells G-ZERO important information about the size and proportions of the stock.

Check dimensions of stock from blueprint on previous page.

- od** the diameter of the bar stock or a little larger (enter all x dimensions as positive diameters)
- id** usually zero unless pipe or a predrilled material
- zlim** length sticking out of chuck plus a safe amount so that the tool doesn’t crash into the chuck. Negative because the front of the part (face) is z0
- zface** amount of material to remove from end of stock (face)
- zclamp** amount of material to be clamped in the chuck
- type** enter the number that correspond to the material type shown on blueprint (303 stainless)

Note: If you want to program a Material type that is not on the current pop-up list, refer to the Reference Manual to learn how to customize Feed Files.



3 Face and Turn OD

2 **TOOL** 1 in/out2 *** FACE AND TURN OD

3 **TYPE** 1 mat1 tnr.032 size.5

Define the tool for the first two operations (facing and turning the part).

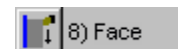
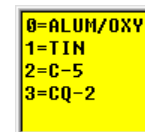
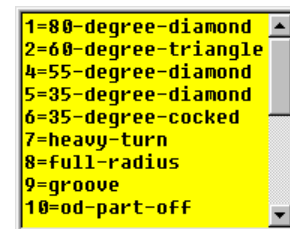
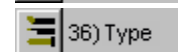
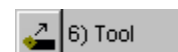
36) Type automatically follows the *6) Tool* command to continue describing the new tool (you do NOT have to select *36) Type*) Select an insert and holder from the pop-up tool library. Then describe the insert material, tnr and size that will face and turn the OD.

Note: To program an insert that is not shown on pop-up boxes, refer to the Reference Manual to learn how to customize Feed Files.

4 **FACE** x.65 z.1 xcut-.064 zcut0 depth0 f7

G-ZERO rapids to the start point, Faces the part, rapids away from the face and returns to the start point. This FACE command takes the material face to finished size (z0).

- x** safe x-position to rapid the tool and clear the part
- z** safe z-position to rapid the tool to and clear the part. Usually just in front of the part
- xcut** the final point in X. Usually 2x the tnr past centerline (minus)
- zcut** the finished Z dimension of the face
- depth** depth of cut per pass. Enter 0 to take it all in one pass. (For more than one pass to full depth, enter a depth of cut amount)



f Using the Feeds file, G-ZERO calculates and displays:
 $v = 600$ SFM (surface feet/minute)
 $s = 8526$ RPM (based on SFM and diameter of current op)
 Instead of a feedrate (in IPR), you can enter a surface finish value: from 3 (smoothest) to 9 (roughest). Each surface finish corresponds with a recommended feedrate from the Feeds file. Using a surface finish value instead of IPR has a distinct advantage; if you later change size/type of tool insert or material, G-ZERO automatically recalculates the IPR.
 In this case, enter 7 for an AA value of 63 as shown on blueprint.

v = 600 SFM s = 3526 RPM				
FINISH CHART				
F	I	AA	Ra	N
.001		4	0.1	N3
.002		8	0.2	N4
.003		16	0.4	N5
.004		32	0.8	N6
.005		63	1.6	N7
.007		125	3.2	N8
.010		250	6.4	N9

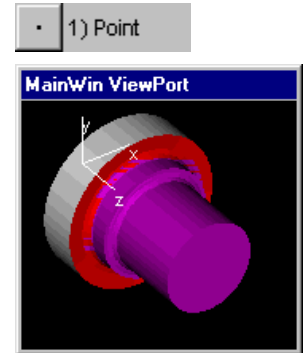
AA arithmetic average
 Aa roughness average
 N surface finish value

- 5 POINT x.5 z.02 f0
- 6 POINT x.5 z-.4 f.005
- 7 POINT x.6 z-.4 f.005
- 8 POINT x.6 z-.51 f.005

Since there is very little material to be removed, program this contour Point-to-Point.

Line 5. Enter the first OD point (x.5, z.02) with the z value a little in front of the face so you don't start the tool right on the material. A feedrate of 0 forces the tool to rapid to this Point.

Program the rest of the points according to the blueprint from the first page of this Project.



4 Drill

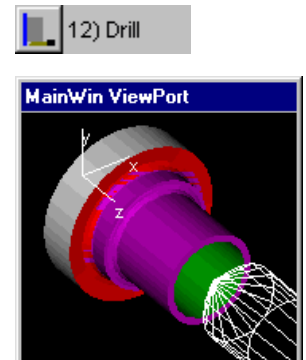
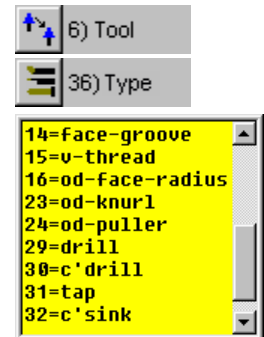
- 9 TOOL 2 in/out1 *** DRILL
- 10 TYPE 29 mat0=M2 tnr0 size.375

Program the 6) Tool and 36) Type for a drilling tool. Select a drill (29) from the tool library and describe the drill material, tnr (0 for a drill) and size.

- 11 DRILL g74 rap.05 cut-.52 peck2 tip1 ret.05 f.0047

Program a 12) Drill command and define the parameters for drilling the center hole. To have G-ZERO calculate and add the length of the drill tip to the overall depth of cut, enter a 1 for yes at the *add drill tip* prompt.

- g** you will normally drill with te 74 cycle
- rap** rapid to this Z location. Should be at a safe distance, usually in front of the part
- cut** drill to this final Z location
- peck** enter 2 as the total number of pecks the tool will make as it reaches the final depth
- tip** enter 1 to add the drill tip (G-ZERO adds 1/3 of the current tool diameter to the total cut distance)
- ret** safe z-position to bring the tool out of the hole and in front of the part for a safe tool change. Usually positive
- f** G-ZERO calculates a feedrate (IPR) based on the modifiable Feed files and displays the value on the yellow pop-up box. If you disagree with the SFM and RPM, customize the Feed files insert a % comment in front of the Drill command to change the SFM (%s) or RPM (%S). (See Reference Manual)



5 Save program and exit G-ZERO Lathe

By default, G-ZERO Lathe v4.0 saves your work every three minutes. However, it is always a good practice to save your work whenever you think you have spent a great deal of time on your program.

To save your work, click the File pull-down menu and select [Save](#).

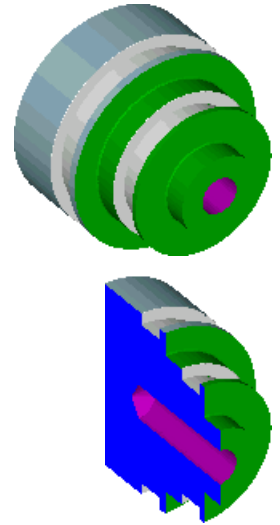
Note: G-ZERO Student version only saves 60 lines of program.

WARNING:

Now, that you have finished and saved your project, you can exit the program by selecting [Exit](#) from the File pull-down menu. If you want to continue with the next project, you can just simply close the file (File|Close). ■

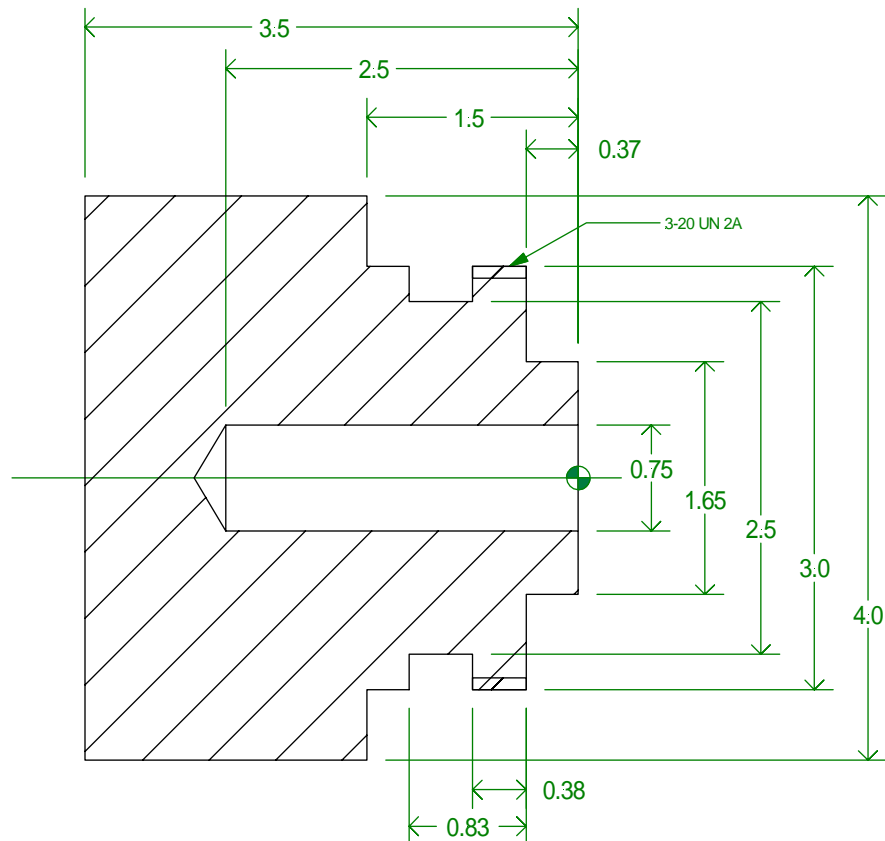
Project 3


Hub



What you will learn:

- ◆ Rough an outside (OD) shape
- ◆ Use 4) COMP and 5) UNCOMP to compensate offsets for cutter radius
- ◆ Repeat source program lines with 15) REPEAT
- ◆ Grooving - 10) GROOVE
- ◆ Threading - 11) THREAD



 CAM Software RAPID OUTPUT Co. 888-656-1945	TITLE: Hub		NOTES: 1. All measurements in inches (not to scale) 2. Deburr all sharp corners 3. Material amount in chuck = 0.5"	RECOMMENDED OP & TOOL LIST: 1. Face , Rough and Finish OD (0.032 tnr x 0.375 ic) (CNMG 332 - 80° diamond) 2. Drill (0.75" dia.) 3. OD Groove (0.125" wide insert) 4. Thread (1/4-20 tpi - 60° insert)
	MATERIAL: 316 Stainless Steel			
	DRAWING: L0002	INITIALS: MC		

1 Define the material

1 MAT'L od4 id0 zlim-3.6 zface.07 zclamp.5 type2=316

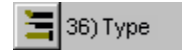
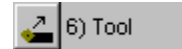


Begin your source program with the *17) Mat'l* command. Use information from blueprint shown on previous page to answer question for the Material command. Remember to enter all X dimensions as positive diameters.

2 Face and turn OD

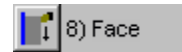
2 TOOL 1 in/out2 *** FACE AND TURN OD

3 TYPE 1 mat0=ALUM/OXY tnr.032 size.375



Define the tool for the first two operations (facing the part and roughing the OD) using information from the blueprint on the previous page. Check the recommended tool list to select the appropriate insert and holder from the pop-up library.

4 FACE x4.1 z.1 xcut-.064 zcut0 depth.05 f7



With the *8) Face* command, G-ZERO rapids to the start point, faces the part, rapids away from the face and returns to the start point.

depth limiting the depth of cut to .05, G-ZERO calculates the 2 passes down the face of the part to get to z0.

f enter a surface finish of 7 (N value on pop-up box) instead of a feedrate (ipr). Each surface finish corresponds with a recommended feedrate from the Feeds file.

overwrite if G-ZERO beeps

Follow these simple programming steps to Rough an outside (OD) shape:

1. 9) Rough command

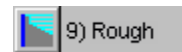
safang: For any angle other than 270°, G-ZERO adjusts all steep sections of the contour so that the tool is only plunged into reachable areas. Use another tool to reach the skipped material!

2. Define the contour using **Comp, Point, Radius, Line**, and **Uncomp**.

xstk / zstk: The amount of stock left for a finish pass. For Backturning / Backfacing operations (in/out 3/4), the zstk should be negative.

3. 9) Rough command again with the same parameters. Actual roughing does not occur until this second Rough is programmed.

5 ROUGH xstk.01 zstk.002 depth.1 x4.02 z.05 safang270 f2



Defines the *9) Rough* command.

xstk and **zstk** Leave some stock in x and z to be finished later

depth Depth determines the amount of material to be removed during each stroke of the tool

x and **z** These values define the point where roughing starts

safang Safang is the steepest safe angle when plunging

f Instead of a feedrate (ipr), enter an aggressiveness level ranging from 1 (light load) to 4 (heavy load). Using an aggressiveness level, if you later change size or type of tool, G-ZERO automatically recalculates the ipr. Enter 2 for this project.

FEED COMPARISON CHART					
Depth	SFH	F1	F2	F3	F4
.030	600	.001	.002	.003	.005
.075	500	.001	.003	.005	.008
.100	450	.002	.004	.006	.010 *
.200	400	.002	.004	.006	.010
.100	500	.002	.005	.009	.015

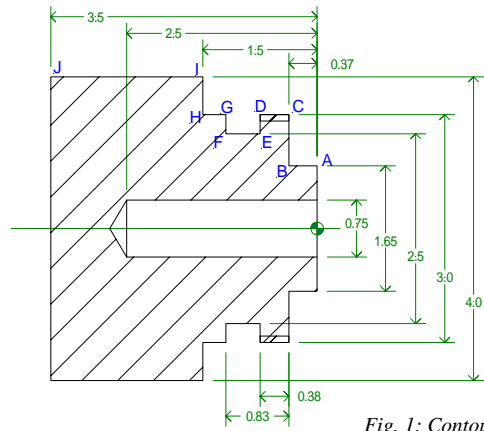


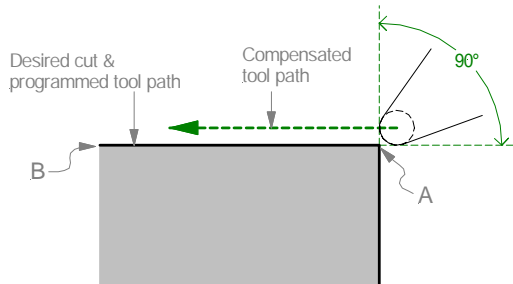
Fig. 1: Contour

To define the contour to be roughed, program:

- 4) *COMP* command to compensate offsets for cutter radius
- 1) *POINT* and 3) *LINE* commands to define contour
- 5) *UNCOMP* command to stop compensating for cutter radius

6 COMP ang90 lft/rt2 f5

To start the tool tangent to both the face (90°) and the first cut (line 180° - from point A to point B), program the COMP angle of 90°. Answer the lft/rt question with a 2 since the tool is on the right.



When roughing, the COMP feedrate is for the finish cut only. Using the Feeds file, G-ZERO calculates and displays SFM (surfacer feet/minute) and RPM (based on SFM and diameter of current op). Instead of a feedrate (in IPR), you can enter a surface finish from the pop-up chart. G-ZERO surface finishes range from 3 (smoothest) to 8 (roughest). Each surface finish corresponds with a recommended feedrate from the Feeds file. In this case, enter 5.

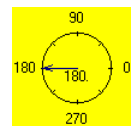
To define the contour, we are going to program Points and Lines starting from Point A, all the way to Point J. (See Fig. 1.)

7 POINT x1.65 z0

The first **POINT** brings the tool onto the contour. G-ZERO makes all cutter compensation calculations automatically once Comp is programmed. Define Point A with x=1.65 and z=0.

8 LINE angle180

The second element is a horizontal **LINE** going toward the 180° direction (from Point A to Point B); therefore, program the Line command with angle=180.

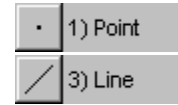


9 POINT x1.65 z-.37

Program a **POINT** command to define Point B with x=1.65 and z=-.37.

10 LINE angle90

Program a **LINE** for the next element: vertical line going toward the 90° direction (from Point B to Point C); therefore, line angle=90.



11 POINT x3 z-.37

Program POINT C with x=3 and z=-.37.

12 LINE angle180

Program a **LINE** command for the horizontal line going toward the 180° direction (from Point C to Point H). Note: groove (shown in Fig.1 with points D-E-F-G) will be cut later.

13 POINT x3 z-1.5

Program POINT H with x=3 and z=-1.5.

14 LINE angle90

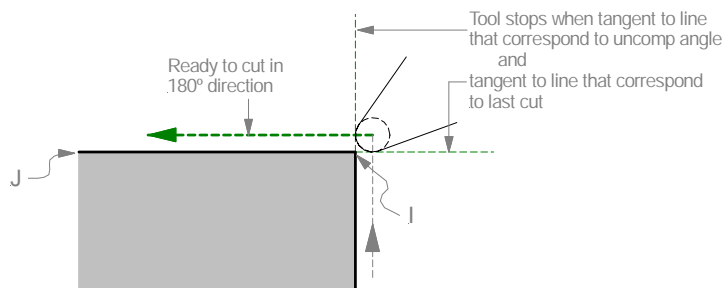
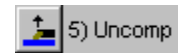
Program a **LINE** command for the vertical line going toward the 90° direction (from Point H to Point I).

15 POINT x4 z-1.5

Program the last corner Point I on the OD with x=4 and z=-1.5.

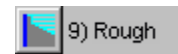
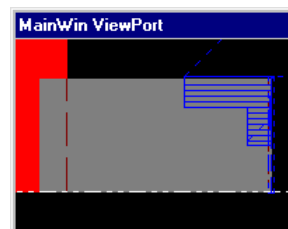
16 UNCOMP angle180

UNCOMP command tells G-ZERO to stop compensating (calculating) for cutter radius. To make sure that the tool cuts a little beyond the last point, program an Uncomp angle of 180°. This means that the tool stops in a position ready to cut in a 180° direction.



17 unROUGH

Turn the Roughing definition off by programming another 9) Rough command (will show as *unROUGH* in your source code). Now, G-ZERO roughs the profile given between the two Rough commands (lines 6 to 16) leaving the stock allowance defined in the Rough command.



18 REPEAT from6 thru16 ***

To remove remaining stock in a finish pass, Repeat the cutting path by reusing source lines. Program the 15) Repeat starting at Comp (line 6) and ending at Uncomp (line 16). Always repeat source program lines instead of retyping.



3 Drill

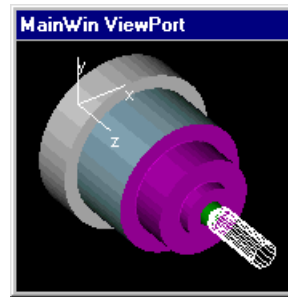
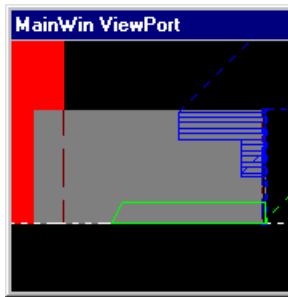
```

19 TOOL 2 in/out1 *** DRILL
20 TYPE 29 mat0=M2 tnr0 size.75
21 DRILL g74 rap.05 cut-2.5 peck-4 tip1 ret.1 f.0085

```

Change tools for the next operation (Drill). Check blueprint for recommended tool. Select a drill from the pop-up tool library and describe the drill material, tnr (0 for a drill), and size.

Use the **12) Drill** command to define the parameters for drilling the center hole. A minus (-) # of pecks triggers a full retract between pecks. To have G-ZERO calculate and add the length of the drill tip to the overall depth of cut, enter a 1 for “yes” at the “add drill tip” prompt. G-ZERO calculates a feedrate (ipr) based on the modifiable Feed Files and displays it on a pop-up box. For this project, use the suggested feedrate.



4 OD Groove

```

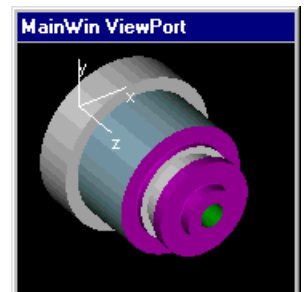
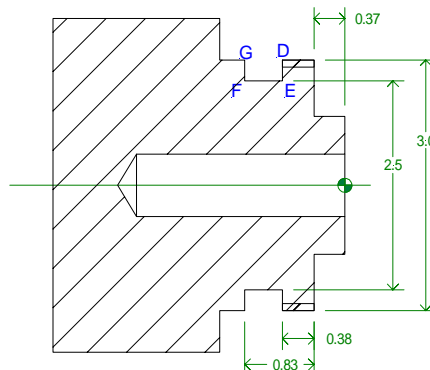
22 TOOL 3 in/out2 *** OD GROOVE
23 TYPE 9 mat1=C-5 tnr0 size.125
24 GROOVE x3 z-.75 dp.25 w.45 skip0 r.005 ir0 a0 f4

```

Now, we are going to change tools and define the new parameters for the OD groove. Check blueprint for suggested tool.

Program the **10) Groove** command with these values:

x = 3	groove starts on x=3
z = -.75	$0.37+0.38 = .75$ (negative side of coordinates)
dp = .25	depth of groove is $(3.0-2.5)/2 = .25$
w = .45	width of groove is $.83-.38 = .45$
skip = 0	groove rough & finish, and no shoulder (<i>see Ref. Manual</i>)
r = .005	need to “deburr all sharp corners”
ir = 0	there is no radius in bottom of groove
a = 0	groove walls are straight (no angle)
f = 2	enter aggressiveness level 4 instead of feedrate (ipr)

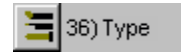


5 Thread OD

25 TOOL 4 in/out2 *** THREAD OD

26 TYPE 15 mat2=CQ-2 tnr0 size20

Program the 6) Tool and 36) Type commands to describe the tool to cut threads.



27 THREAD maj3 tpi20 min2.9371 S318 z-.25 zfin-.8 num8 a60

G-ZERO uses your choice of machine threading cycles.

maj = 3 major diameter as shown on blueprint

tpi = 20 tpi as shown on blueprint

min = 2.938655 .. minor diameter is calculated and displayed as default by G-ZERO based on a standard 60° thread.

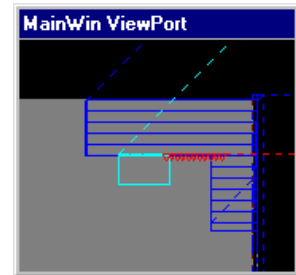
S = 255 G-ZERO recommends a maximum RPM (shown on pop-up box with an asterisk) based on the selected tool and material.

z = -.25 G-ZERO suggests to start threading just in front of your part. However, since the typical value would be twice the thread pitch in front of the thread, enter -.25.

zfin = -.8 enter the z location where you want the thread to end

num = 8 number of passes to cut thread

a = 60 to keep a uniform load on the threading tool, you may want to start each threading pass over a little in Z as well as in X so the tool is doing most of the cutting with the front of the tool. Enter the angle of the thread here.



6 Save and exit

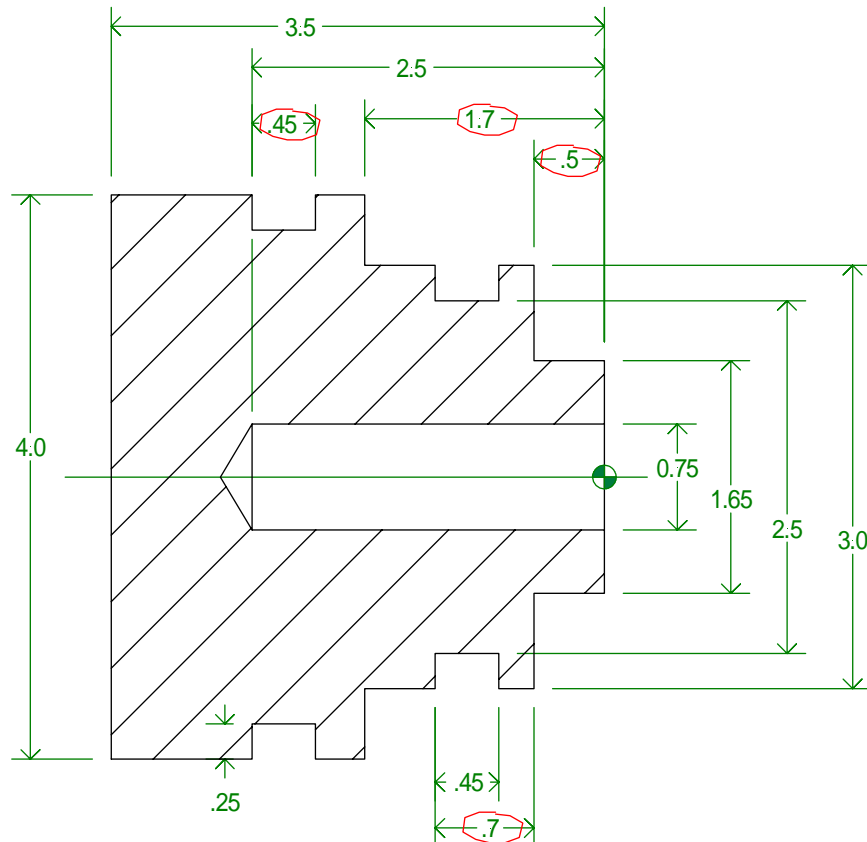
Now that you are done with this project, save your file and exit the program. If you want to continue with the next project, simply save and close the file. ■


Project 4

Hub (revisions)

What you will learn:

- ◆ Duplicate source file using Save As
- ◆ Print source file
- ◆ Change value using ALTER command
- ◆ Change value using mouse and keypad
- ◆ Insert a new command
- ◆ Move source lines
- ◆ Delete source lines



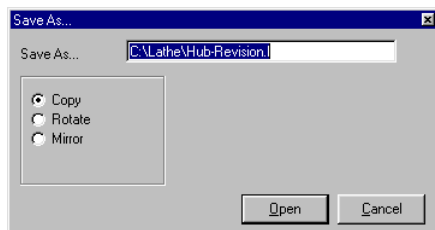
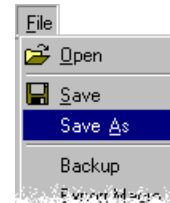
 CAM Software RAPID OUTPUT Co. 888-656-1945	TITLE:	Hub		NOTES: 1. All measurements in inches (not to scale) 2. Deburr all sharp corners 3. Material amount in chuck = 0.5"	RECOMMENDED OP & TOOL LIST: 1. Drill (0.75" dia.) 2. Face, Rough and Finish OD (0.032 trn x 0.375 ic) (CNMG 332 - 80° diamond) 3. Rough/Finish OD (.032trn x .375 ic) 4. 2 OD Grooves (0.125" wide insert) 5. Thread (1/4-20 tpi - 60° insert)
	MATERIAL:	316 Stainless Steel			
	DRAWING:	L0002 Rev A	INITIALS:		

In this project, we are going to practice editing techniques to use in any Source program. If you have a new part very similar to another one you created before, you may save time by copying an old source program and editing some figures.

1 Duplicate a source file using Save As

Project 3 needs some revisions but we want to keep a copy of the file the way it is. To do so, we are going to open the Hub file saved in Project 3, and save it again using a different name.

1. **Open** your source program **Hub.l** (or other file name you may have used in Project 3). If you are starting G-ZERO Lathe, select the file Hub.l in the “Source File to Open” window.
2. Once Hub.l is loaded, select **Save As** from the File pull-down menu.
3. Type the new file name **Hub-Revision** (use other name if you wish) in the File name section and click the Save button.
The Save As function allows you to create an identical copy (Copy), a copy of the same part but rotated at a certain degree (Rotate), or the same part but flipped (Mirror). In our case, we want an identical copy, so make sure the **Copy** option is selected before you click the **Open** button.



Your Hub.l file is still intact in your computer. A new file named Hub-Revision.l was created and saved in the same directory as Hub.l. The new file Hub-Revision is now the active program; any change you make is going to affect the new file.

2 Print source file

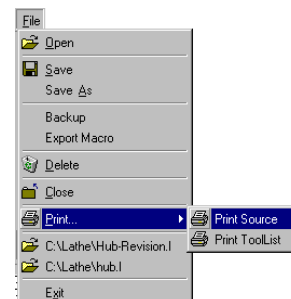
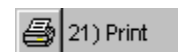
You may want to print the source program so you can see the lines that need changes.

Use your mouse to select command **21) Print** or you can just key in the corresponding command number (21).

G-ZERO Lathe v4.0 will ask you a couple of questions to determine the range of source codes you want to print. In this case, we want the whole source program, **from line 1 to line 27**. Your current source program will be printed on the default printer set up on your computer.

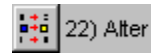
Note that this command line (PRINT from 1 thru27) is NOT added into your source program.

(Another way to print the entire source program is by selecting the Print option from the File pull-down menu.)

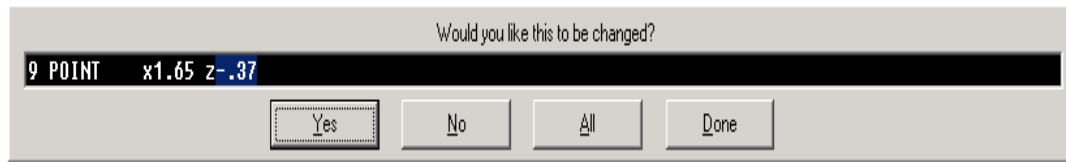


3 Change value using the ALTER command

The z values of points B and C in Fig. 1 of Project 3 need to be changed from -.37 to -.5. These points are in lines 9 and 11.



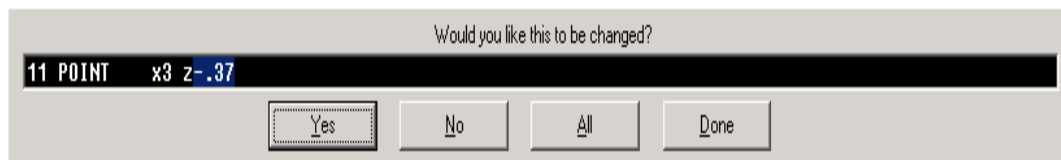
27 ALTER line 9 from-.37 to-.5



- Yes: G-ZERO will change the incorrect value and look down the Source program for another occurrence of the same incorrect value.
- No: G-ZERO will not change the incorrect value but will look down the Source program for another occurrence of the same incorrect value.
- Done: G-ZERO will not change the incorrect value and will not look for more occurrences.
- All: G-ZERO will change the incorrect value and EVERY number in the source that alsomatches the incorrect value -- without double-checking. WARNING: Using "All" can be very dangerous.

Click the [Yes](#) button to confirm this change.

G-ZERO displays a second screen to confirm another value -.37 it found on line 11.



Click the [Yes](#) button to confirm this change.

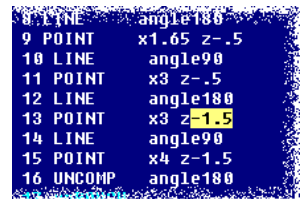
Note that this ALTER command line is NOT added to your source program.

4 Change value using mouse and keypad

Another way to change a value is by using your mouse and keypad.

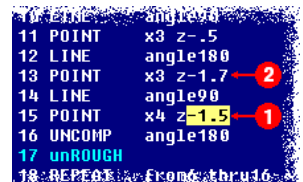
Change the z value on line 13 from -1.5 to -1.7 (key in new value):

- 1) Locate the value you need to change and select it with your mouse (value -1.5 on line 13). Now, the value you selected is highlighted and the working screen background turned to blue (you are in the "editing" mode).
- 2) Key in the new value -1.7 and press the Enter key.



Replace the z value on line 15 (-1.5) with the z value on line 13 (-1.7):

- 1) Locate the value you need to change and select it with your mouse (value -1.5 on line 15). Now, the value you selected is highlighted and the working screen background turned to blue.
- 2) At this time, use your mouse to select the value you want to change to (-1.7 on line 13). Notice that the z value on line 15 is replaced with the new value. Press the Enter key to exit the "editing" mode.



5 Insert a new command

Now, we are going to insert a new groove with an “approach” point after the Groove in line 24.

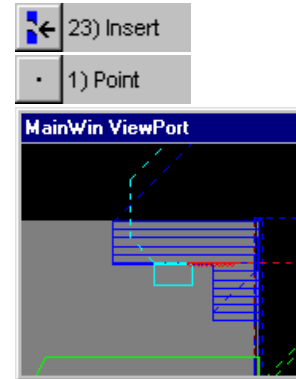
Insert a POINT after the Groove command:

27 INSERT after24

Program the 23) *Insert* command, when it asks WHAT LINE NUMBER WOULD YOU LIKE TO INSERT AFTER? answer 24. This will insert a new line after line 24 (resequences all line numbers after line 24 to accommodate new line) and switches to editing mode while waiting for you to program the new command.

25 POINT x5 z-1.5 f0

Program the approach point with x=5 (press to overwrite if necessary) and z=-1.5, and f=0 for rapid. If this Point is not programmed, the tool will move from the first groove, through the part to the second groove.



Insert a new GROOVE after the approach point:

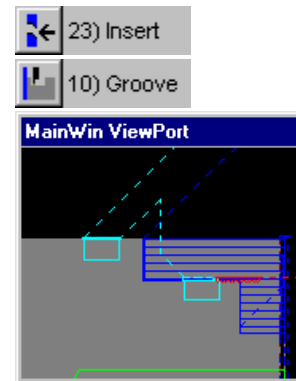
28 INSERT after25

Program the 23) *Insert* command to insert a new line after the approach point on line 25. Notice that the *Insert* command is an editing command; therefore, it is not included in your source program. G-ZERO now switches to the editing mode and waits for you to program the new command.

26 GROOVE x4 z-2 dp.25 w.45 skip0 r.005 ir0 a0 f2

This groove starts at x=4 and z=-2, the rest of the information is the same as the first groove.

After you finish programming the Groove command, G-ZERO exits the editing mode.



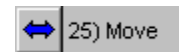
6 Move source lines

If you would rather Drill before Facing, move the drilling operation (Tool 2, Type, Drill) to the beginning of the program, right after the Material command. In this way, drilling becomes the first cutting operation of the program.

29 MOVE from19 thru21 after1

Program the *Move* command to move lines 19 through 21 after line 1.

The 25) *Move* command automatically renumbers all lines, including the references within the *Repeat* command.



7 Delete source lines

Finally, we need to delete the threading operation from line 27 to line 29.

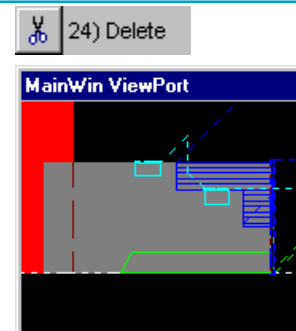
Since they are the last three lines in your program, you could use the *Esc* key to delete them one at a time.

However, a faster way is to use the 24) *Delete* command.

29 DELETE from27 thru29

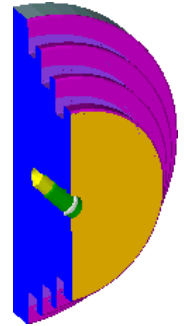
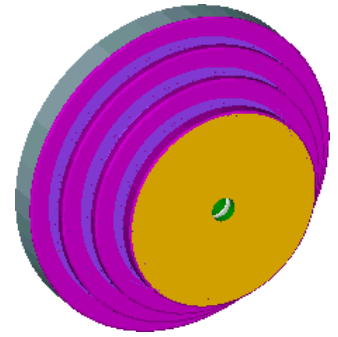
When you use the *Delete* command, G-ZERO pops a questions to make sure those are the lines you really want to delete.

Now, your program should have only 26 lines of codes with 3 tools operations. Save your file and exit the program. ■



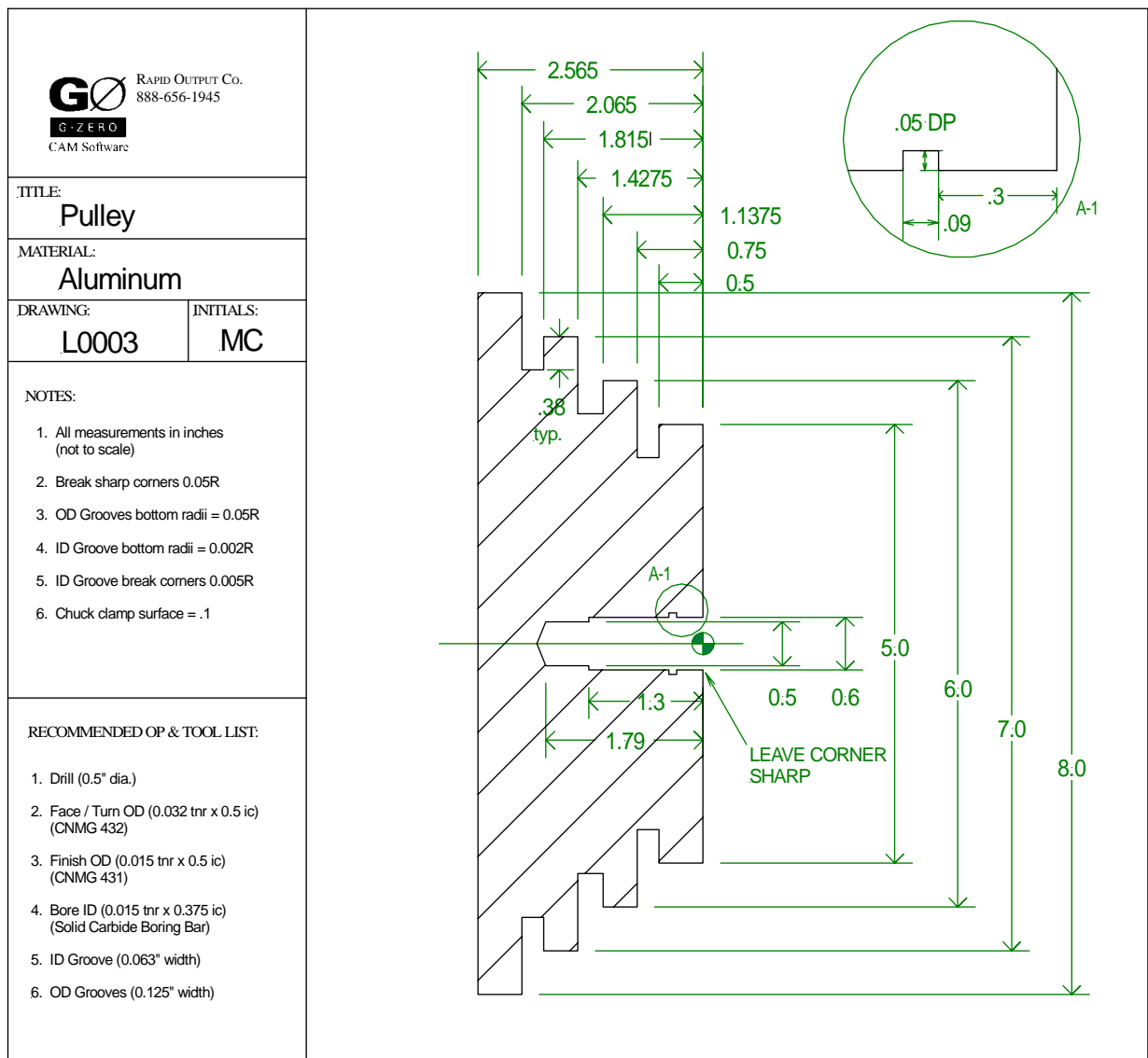
Project 5

Pulley



What you will learn:

- ◆ Use Radius to define roughing contour
- ◆ More on Comp and Uncomp
- ◆ Use of Bore command to enlarge previously drilled hole
- ◆ ID Groove
- ◆ Use of Peck command before OD Groove



1 Define material and drill center hole

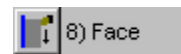
```
1 MAT'L od8 id0 zlim-2.8 zface.03 zclamp.1 type4=ALUMINUM
2 TOOL 1 in/out1 *** DRILL
3 TYPE 29 mat0=M2 tnr0 size.5
4 DRILL g74 rap.05 cut-1.79 peck-4 tip1 ret.05 f.006
```



Use the blueprint to get information for the *Mat'l* and *Tool* commands.
Program the *12) Drill* command to define parameters for drilling the center hole. Note that a minus (-) # of pecks triggers a full retract between pecks.

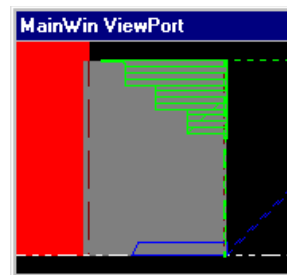
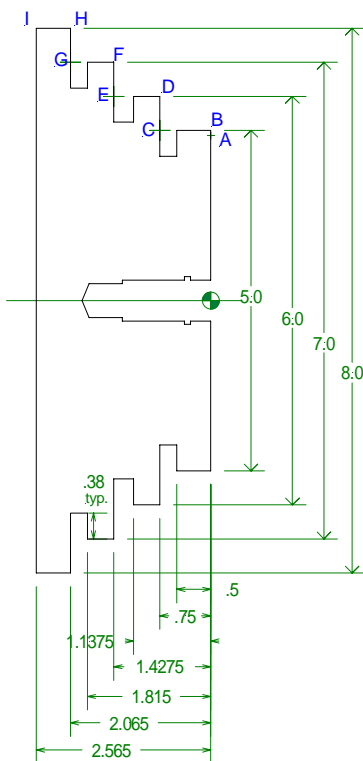
2 Face and turn OD - Use Radius to define roughing contour

```
5 TOOL 2 in/out2 *** FACE AND TURN OD
6 TYPE 1 mat1=TIN tnr.032 size.5
7 FACE x8.1 z.04 xcut-.064 zcut0 depth0 f5
```



Define the tool for the next operation: facing roughing the OD.

To do the facing operation, G-ZERO rapids to the start point, faces the part, rapids away from the face and returns to the start point. Note that the xcut is usually 2x the tnr pat centerline (minus). Instead of feedrate, enter s surface finish value of 5 (if you later change size or type of tool insert, G-ZERO automatically recalculates the ipr).



8 ROUGH xstk.02 zstk.002 depth.002 x8.05 z.05 safang270 f3

Program the 9) *Rough* command to start roughing an outside (OD) shape. Note that we are skipping all three grooves to be cut later; therefore the rough safe angle can be 270°.

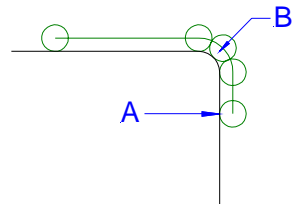
To define the outside contour, use the 1) *Point*, 2) *Radius* and 3) *Line* commands between the 4) *Comp* and 5) *Uncomp* commands to compensate for cutter radius.

The *Comp* command is always followed by a *Point* or *Radius type1* (defined location).



9 COMP ang90 lft/rt2 f6

COMP angle is 90 because the first cut (from Point A to Radius B) is going in the 90° direction. Remember to program print dimensions for the contour because G-ZERO makes all cutter compensation calculations automatically once Comp is programmed.

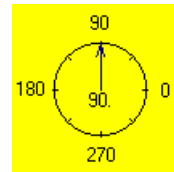


10 POINT x4.85 z0

POINT A (x=4.85; z=0) is the approach point, and it brings the tool onto the contour.

11 LINE angle90

When you call the LINE command, G-ZERO pops a compass for directional reference. Always imagine the cutter to be at the center of the compass. The direction of this first line angle is 90°. Note: visualize the Line as a “string” that connects Points and Radiuses to one another.



12 RADIUS .05 type2 x5 z0 (xc4.9 zc-.05)

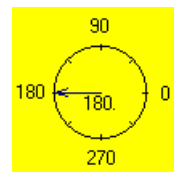
RADIUS B. To break corners, program a corner Radius instead of a Point.

radius: .05 it is positive because it is an outside Radius
type: 2 corner radius because x and z values belong to corner
x: 5 x value of corner
z: 0 z value of corner

G-ZERO will later calculate the exact x and z values of the center of the Radius and display them in parenthesis. See Reference Manual to find out more details on Radius.

13 LINE angle180

Program a **LINE** for the next element: cutter travels on a horizontal line going toward the 180° direction (from Radius B to Point C); therefore, line angle=180.



14 POINT x5 z-.75

Program a **POINT** to define the next element: Point C with x=5 and z=-.75.

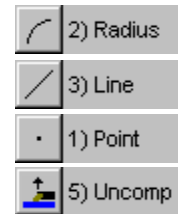
15 LINE angle90

Program a **LINE** for the next element: cutter travels on a vertical line going toward the 90° direction (from Point C to Radius D); therefore, line angle=90.

16 RADIUS .05 type2 x6 z-.75 (xc5.9 zc-.8)

The next element is a **RADIUS**; program it similarly to the previous Radius:

radius: .05 it is positive because it is an outside Radius
type: 2 corner radius because x and z values belong to corner
x: 6 x value of corner
z: -.75 z value of corner



17 LINE angle180

The next element is a horizontal line; cutter travels toward the 180° direction (from Radius D to point E); therefore, line angle=180.

18 POINT x6 z-1.4275

The next element is Point E with x=6 and z=-1.4275.

19 LINE angle90

Cutter travels from Point E to Radius F in a vertical line going toward the 90° direction; therefore, line angle=90.

20 RADIUS .05 type2 x7 z-1.4275 (xc6.9 zc-1.4775)

Radius F is also a corner radius (type 2) with x=7 and z=-1.4275

21 LINE angle180

Cutter travels from Radius F to Point G in a horizontal line going toward the 180° direction; therefore, line angle=180.

22 POINT z7 z-2.065

Point G is located at x=7 and z=-2.065.

23 LINE angle90

Cutter travels from Point G to Radius H in a vertical line going toward the 90° direction.

24 RADIUS .05 type2 x8 z-2.065 (xc7.9 zc-2.115)

Radius H is a type2 Radius (corner radius) with x=8 and z=-2.065.

25 LINE angle180

Cutter travels from Radius H to Point I in a horizontal line going toward the 180° direction.

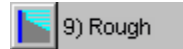
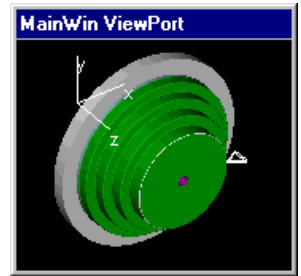
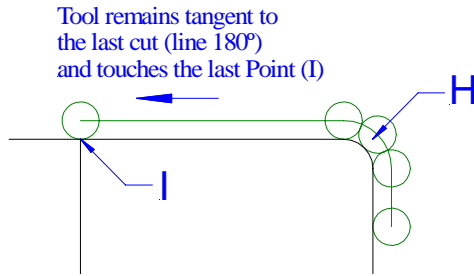
26 POINT x8 z-2.565

Point I is located at x=8 and z=-2.565.

27 UNCOMP angle180

UNCOMP tells G-ZERO to stop compensating for cutter radius. Describe the direction (angle) the tool will be moving at the end of the toolpath. This UnComp angle (180°) is the direction the tool is moving between the Radius (line 24) and the Point (line 26).

Note that G-ZERO knows not to gouge your part as it pulls off the part.



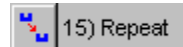
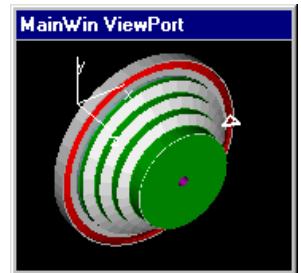
28 unROUGH

Turn the roughing definition off by programming another ROUGH command. After selecting Rough from the menu, it becomes UnRough in the Source. Now, G-ZERO roughs the profile given between the Rough (line 8) and UnRough (line 28) commands leaving the stock allowance defined in the Rough command.

```
29 TOOL 3 in/out2 *** FINISH OD
30 TYPE 2 mat1=TIN trn.015 size.5
31 REPEAT from9 thru27
```

Define the tool to finish the OD (final pass).

To remove remaining stock in a finish pass, REPEAT the cutting path; just re-use already written Source lines. Program the REPEAT command to repeat lines from Comp (line 9) to Uncomp (line 27).



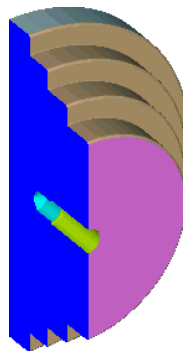
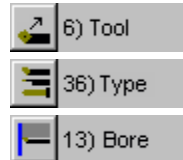
3 Boring

```
32 TOOL 4 in/out1 *** BORE ID
33 TYPE 1 mat2=C-5 trn.015 size.375
34 BORE x.49 z.05 xcut.6 zcut-1.3 depth.04 f6
```

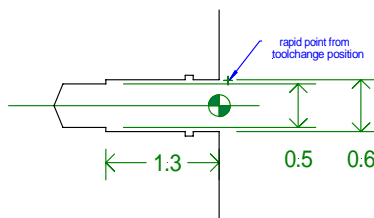
Change tools to bore the already drilled hole.

Program BORE to enlarge the inside of the drilled hole. This is the first of 2 Bore moves. This Bore takes the material to its finished, inside dimension.

x	.49	x location of rapid point from toolchange position
z	.05	z location of rapid point from toolchange position
xcut	.6	final x dimension regardless of number of passes
zcut	-1.3	final z dimension regardless of number of passes
depth	.04	radius amount (not diameter) to be removed. Example: to enlarge a .5" hole to .6" in 2 passes, depth of cut = .025.
f	6	instead of a feedrate (ipr), enter a surface finish from the pop-up chart. Each surface finish corresponds with a recommended feedrate from the Feeds file.



overwrite if G-ZERO beeps



4 ID Groove

35 TOOL 5 in/out1 *** ID GROOVE

36 TYPE 9 mat0=TIN tnr0 size.063

Change tools to cut the ID Groove. Note that *tnr* could range from 0 to half the width of the tool, and *size* is the width of the grooving tool.

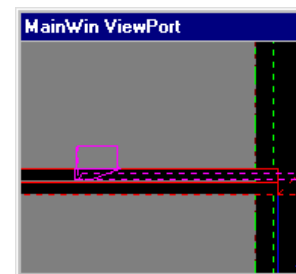
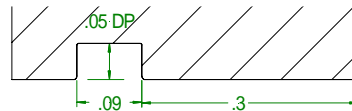
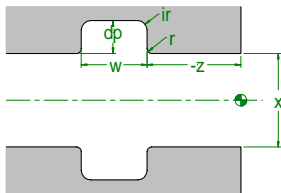
37 POINT x.58 z.1

This approach Point ensures that the tool will not hit the part on its way to the groove. A safe “approach” Point is almost always needed for ID grooves.

38 GROOVE x.6 z-.3 dp.05 w.09 skip0 r.005 ir.002 a0 f1

Program the Groove command to describe and cut the ID groove with these values:

x	.6	x dimension of the groove surface
z	-.3	z dimension of the right-most edge
dp	.05	depth of the groove
w	.09	width of the groove
skip	0	left edge of groove is NOT against a shoulder -- both walls need radii. (Answer 1 for radius on right edge only)
r	.005	breaks outside corners with .005 radius (a negative radius tells G-ZERO to chamfer instead of radiusing the edges)
ir	.002	radius in bottom of groove
a	0	angle of walls
f	1	instead of a feedrate (ipr), enter an aggressiveness level of 1 (light load); if you later change size or type of insert, G-ZERO automatically recalculates the ipr



39 POINT x.58 z.1 f0

This retract Point ensures that the tool will not hit the part on its way to the toolchange position. A safe “retract” Point is almost always needed for ID grooves. A feedrate of 0 (zero) moves the tool to this Point at rapid.

5 OD Grooves

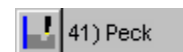
40 TOOL 6 in/out2 *** 3 OD GROOVES

41 TYPE 10 mat1=C-5 tnr0 size.125

Change tools to cut the 3 OD grooves.

42 PECK amt.375 liftup.005 groovestop0

Program the Peck command to break up long stringy chips and is programmed before a Groove, Rough, Bore or Partoff command.



When Peck is programmed with Groove and groovestep=0, G-ZERO uses its own calculated stepover amount. Any other groovestep value equals the actual stepover amount.

Peck is active until a new Tool is programmed or Peck is programmed again with 0 (zero) for all three parameters (amt, liftup, groovestep). This Peck command governs all of the next 3 OD grooves.

```
amt          .375  peck distance
liftup       .005  distance to lift up between pecks
groovestep   0     distance to stepover between each pass when
                   grooving
```

43 GROOVE x5 z-.5 dp.38 w.25 skip1 r.05 ir.05 a0 f2

44 POINT x6.2 z-.5 f0

Program a Groove command to cut the right-most groove. Skip 1 will radius the right groove edge only.

Program a retract Point to ensure that the tool will not hit the part between grooves. A feedrate of 0 moves the tool to this Point at rapid.



45 GROOVE x6 z-1.1375 dp.38 w.29 skip1 r.05 ir.05 a0 f2

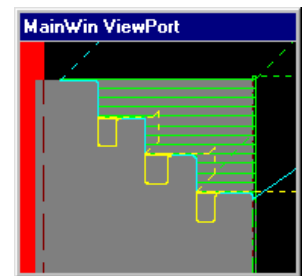
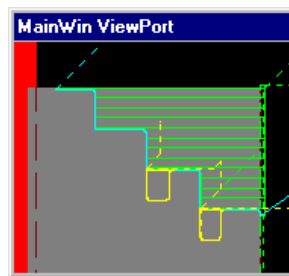
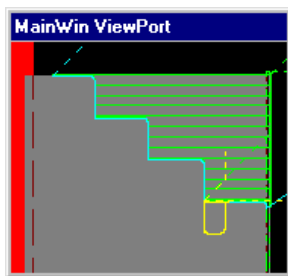
46 POINT x7.2 z-1.25 f0

Program a Groove command to cut the middle groove. Make sure to enter a skip value of 1 so that only the right groove edge will have a radius.

Again, a retract point ensures that the tool will not hit the part on its way to the third groove. A feedrate of 0 moves the tool to this Point at rapid.

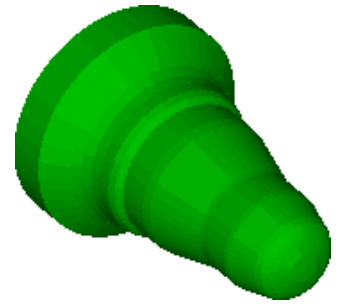
47 GROOVE x7 z-1.815 dp.38 w.25 skip1 r.05 ir.05 a0 f2

Program a Groove command to cut the last groove (left-most groove).



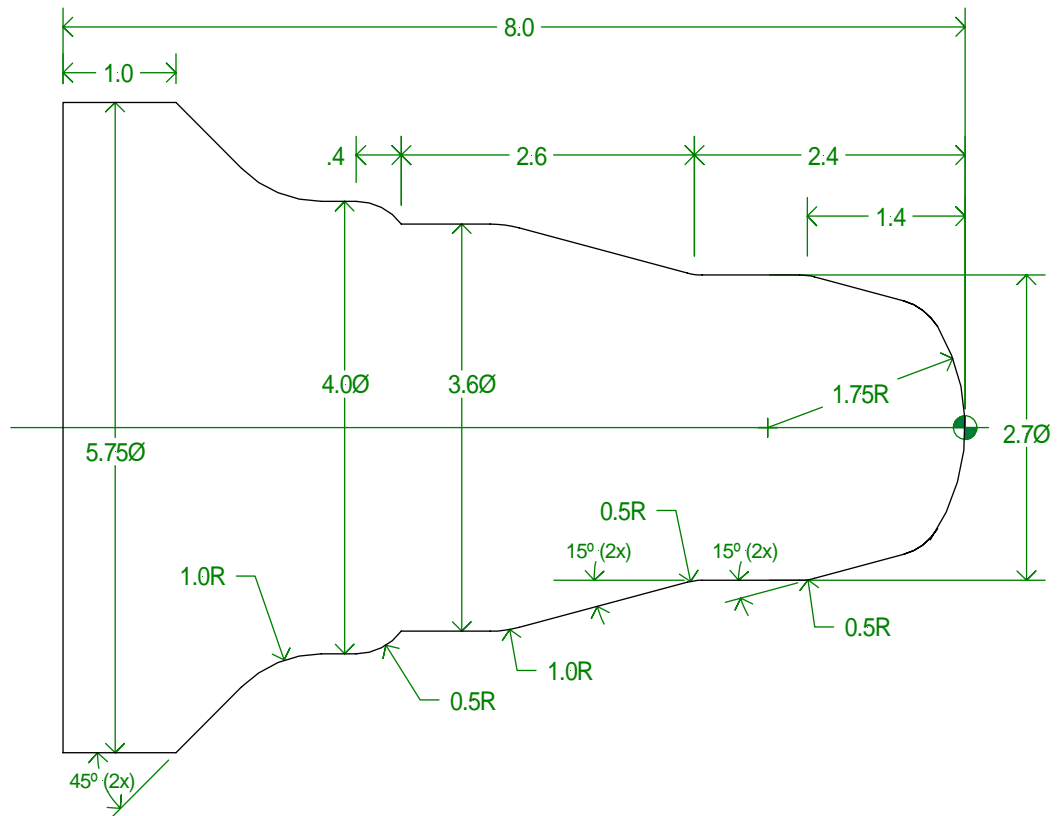
Project 6


Punch



What you will learn:

- ♦ Programming RADIUS after a COMP command
- ♦ Use of various types of RADIUS (center, corner, unknown) to define roughing contour



 CAM Software RAPID OUTPUT Co. 888-656-1945	TITLE:	Punch		NOTES: 1. All measurements in inches (not to scale) 2. Deburr all sharp corners 3. Material amount in chuck = 0.1"	RECOMMENDED OP & TOOL LIST: 1. Face and Rough OD (0.032" trr x 0.375" ic) (CNMG 332) 2. Finish OD (0.015" trr x 0.375" ic) (CNMG 331)
	MATERIAL:	Aluminum			
	DRAWING:	L0004	INITIALS:		

1 Define material and face the part

```

1  MAT'L      od5.8 id0 zlim-8 zface.02 zclamp.1 type4=ALUMINUM
2  TOOL 1    in/out2 *** FACE AND ROUGH OD
3  TYPE 1    mat1=TIN tnr.032 size.375
4  FACE      x5.9 z.03 xcut-.064 zcut0 depth0 f5

```

Use the blueprint to get information for the *Mat'l* and *Tool* commands.

Facing: G-ZERO rapids to the start point, faces the part, rapids away from the face and returns to the start point.



2 Use Radius (various types) to describe Rough contour

```

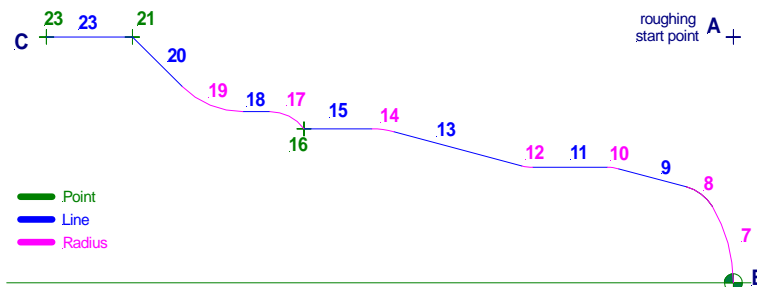
5  ROUGH     xstk.02 zstk.002 depth.2 x5.8 z.05 safang270 f3

```

Program the Rough command leaving some stock for a finish pass.

Define Point A (see graphic below) as the point where roughing starts.

The tool we are using can reach every section of the contour we are going to define; we don't need G-ZERO to adjust steep sections to skip unreachable areas. Therefore, program a safe angle of 270°.



The numbers in this illustration correspond to the line numbers in the source code below.

```

6  COMP      ang90 lft/rt2 f5

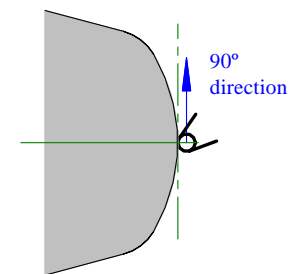
```

Program the Comp command to compensate instead of calculating offsets for cutter radius. We are going to define the contour starting from the origin, on Point B, and finishing on Point C.

```

ang    90      90° is the tangent angle to the Point where we want the
               tool to start cutting
lft/rt  2      tool is on the right side (when describing contour, tool
               is moving toward the chuck)
f       5      when roughing, the COMP feedrate is for the finish cut
               only. Instead of ipr, we can enter a surface finish from
               the on-screen chart.

```

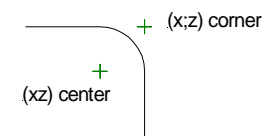
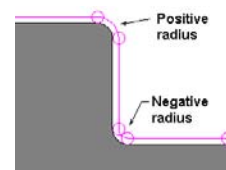


```

7  RADIUS    1.75 type1 x0 z-1.75

```

This outside (+) Radius is dimensioned as a center (type1). When programming a Radius after a Comp command, the Comp angle dictates exactly the direction the tool will be travelling as it begins to interpolate this Radius. A Radius following Comp should always be a type1 (center).

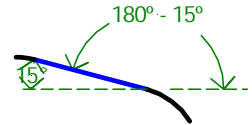


8 RADIUS .5 type0 (xc1.276286 zc-.6751633)

This outside Radius is unknown (type0) because we don't have the dimensions of its center nor corner. G-ZERO will calculate the center location of this blending Radius between the previous Radius (line 7) and the Line (line 9) and place it in parenthesis.

9 LINE angle165

Enter the direction the cutter will be travelling to the next location. The drawing doesn't directly show the angle we need; however, we can key in a math calculation (180-15) and G-ZERO will compute it for us (165).



10 RADIUS .5 type2 x2.7 z-1.4 (xc1.7 zc-1.465826)

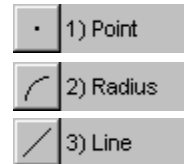
This outside (+) Radius is dimensioned as a corner (type2) between two lines. G-ZERO will later calculate the exact location of the center of this radius and place it in parenthesis)

11 LINE angle180

Cutter travels in a horizontal line toward the 180° direction.

12 RADIUS -.5 type2 x2.7 z-2.4 (xc3.7 zc-2.334174)

This inside (-) Radius is dimensioned as a corner (type2) between two lines.



13 LINE angle165

This line is parallel to the first one we defined; therefore, enter angle 165.

14 RADIUS 1 type0 (xc1.6 zc-4.211075)

This outside (+) Radius is unknown (type0) because we don't know the location of its center nor its corner (intersection of lines before and after this radius)

15 LINE angle180

This line is a horizontal line going toward the 180° direction.

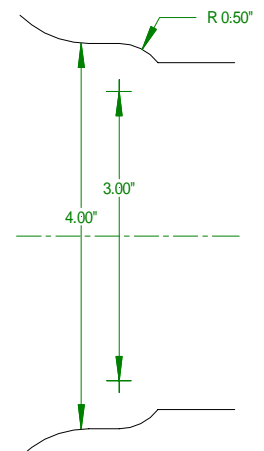
16 POINT x3.6 z-5

This Point defines the x and z position of a sharp corner. The z value of this point is shown on the blueprint as -2.4 - 2.6; just enter it like that (as a calculation) and let G-ZERO compute it for you.

Note that after you enter this Point, G-ZERO has enough information to calculate the center of the previous unknown Radius (line 14) and displays it in parentheses.

17 RADIUS .5 type1 x3 z-5.4

This outside (+) Radius is dimensioned as a center (type1). The x value of the center of the radius is 3 (see illustration on the right) and the z value can be entered as the calculation: -2.4-2.6-.4.

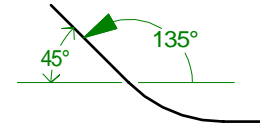


18 LINE angle180

Enter the angle the cutter will be travelling to the next location (180°).

19 RADIUS -1 type0 (xc6 zc-5.710786)
This is an inside (-) Radius and it's unknown (type0) because it's not dimensioned by neither center nor corner.

20 LINE angle135
Enter the angle the cutter will be travelling to the next location (135°). You can enter this value as a calculation (180-45), and let G-ZERO compute for you.



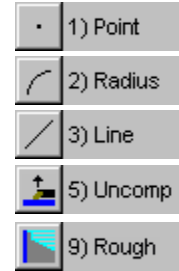
21 POINT x5.75 z-7
This Point defines the x and z position of a sharp corner.
Note that after you enter this Point, G-ZERO calculates the center of the previous unknown Radius (line 19) and displays it in parentheses.

22 LINE angle180
Enter the angle the cutter will be travelling to the next location (180°).

23 POINT x5.75 z-8
This Point moves the tool to its final position on the part before retracting.

24 UNCOMP angle180
Uncomp disengages tool nose compensation (like G40) because the profile is now completely defined. Uncomp angle 180° stops the cutter tangent to the last defined Point (line 23).

25 unROUGH
Turn the roughing definition off by programming another Rough command. After selecting Rough from the menu, it becomes unROUGH in the Source. Now, G-ZERO roughs the profile given between the Rough (line 5) and UnRough (line 25) commands leaving the stock allowance defined in the Rough command.



3 Finish OD

26 TOOL 2 in/out2 *** FINISH OD
27 TYPE 2 mat1=TIN tnr.015 size.375
28 REPEAT from6 thru24

Define the tool for the final operation: finishing the OD.
To remove remaining stock in a finish pass, use the Repeat command to reuse source lines for the cutting path. Start at Comp (line 6) and end at Uncomp (line 24). ■



Project 7

Housing



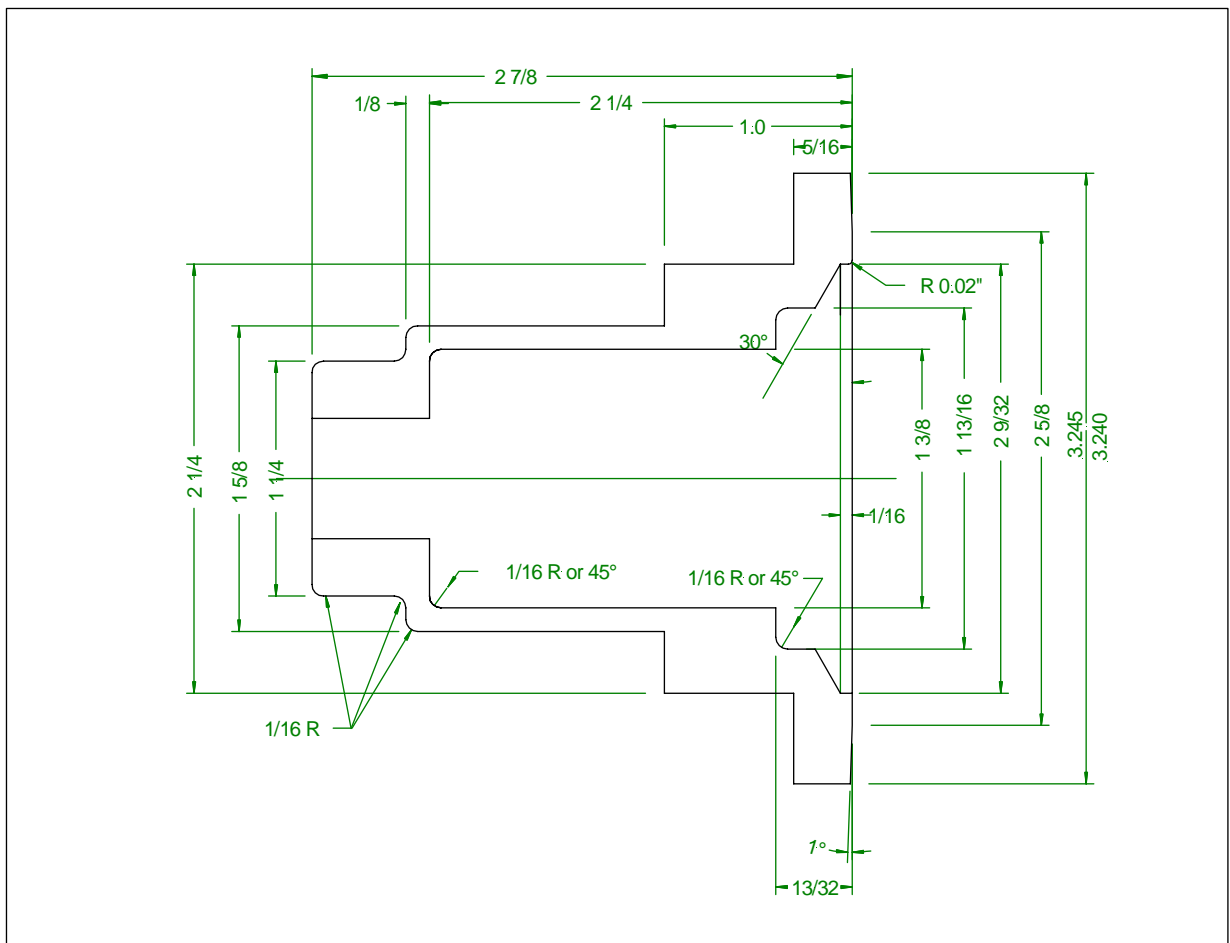
What you will learn:


Housing - Side A

- ◆ How to enter fractional dimensions
- ◆ Use Point to define sharp corner between 2 lines
- ◆ Use Drill to countersink a drilled hole
- ◆ Use DeadTap to tap a hole

Housing - Side B

- ◆ Angled face and finish OD
- ◆ Use Face with ID tool



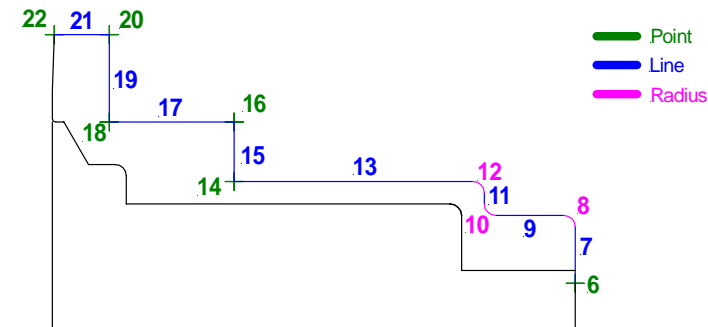
 RAPID OUTPUT CO. 888-656-1945	TITLE: Housing		NOTES: 1. All measurements in inches (not to scale) 2. Hand deburr all sharp corners 3. Dimensional tolerances: fractional = 1/64 ; angular = 1/2° 4. 3/4" dia. 90° countersink	RECOMMENDED OP & TOOL LIST:	
	MATERIAL: Aluminum			SIDE A 1. Rough OD 2. Face and Finish OD (angled surface) 3. Drill (9/16" dia.) 4. Countersink (3/4 dia. x 90°) 5. Tap (5/8-18)	SIDE B 1. Turn OD and Face (angled surface) 2. Drill (1.25" dia. indexable insert drill) 3. Rough and Finish ID
	DRAWING: L0005 A/B	INITIALS: MC			

HOUSING - SIDE A

1 Define material and rough OD

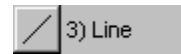
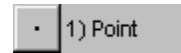
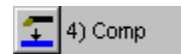
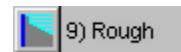
- 1 MAT'L od3.25 id0 zlim-3 zface.02 zclamp.4 type4=ALUMINUM
- 2 TOOL 1 in/out2 *** ROUGH OD
- 3 TYPE 1 mat1=TIN tnr.032 size.5

Use blueprint to get information for the Material and Tool commands.



The numbers in this illustration correspond to the line numbers in the source code below.

- 4 ROUGH xstk.02 zstk.003 depth.2 x3.3 z.05 safang270 f3
Program the Rough command leaving some stock for a finish pass.
- 5 COMP ang90 lft/rt2 f6
To define the contour to be roughed, program the Comp command to compensate instead of calculating offsets for cutter radius. To force the tool to travel UP, the Comp angle is 90° (tangent angle to the Point where you want the tool to start cutting)
- 6 POINT x.5 z0
This point brings the tool onto the contour. This is the first Point on the OD contour definition
- 7 LINE angle90
Enter the angle the cutter will be travelling to the next location (90°).
- 8 RADIUS .0625 type2 x1.25 z0 (xc1.125 zc-.0625)
This outside (+) Radius is dimensioned as a corner (type2) between two intersecting lines (line 7 and line 9). G-ZERO will later calculate the exact location of the center of this radius and display it in parenthesis.
Since the radius is dimensioned in fraction, you can enter it as the division 1/16 and let G-ZERO do the computation.
- 9 LINE angle180
Enter the angle the cutter will be travelling to the next location (180°).



10 RADIUS `-0.0625 type2 x1.25 z-.5 (xc1.375 zc-.4375)`

This inside (-) Radius is dimensioned as a corner (type2) between two intersecting lines. The x value of this corner is shown as 1 1/4, or 1.25. The z value is dimensioned as (negative) 2 7/8 - 2 1/4 - 1/8. This can be entered as -2-7/8+2+1/4+1/8 and let G-ZERO do the calculation.

$$-2\frac{7}{8} + 2\frac{1}{4} + \frac{1}{8} = \\ = -2-7/8 + 2+1/4 + 1/8$$

11 LINE `angle90`

Enter the angle the cutter will be travelling to the next location (90°).

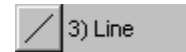
12 RADIUS `.0625 type2 x1.625 z-.5 (xc1.5 zc-.5625)`

This outside (+) Radius is dimensioned as a corner (type2). X value of this corner is 1 5/8, or 13/8. Z value of this corner is calculated as the previous Radius (line 10).



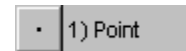
13 LINE `angle180`

Enter the angle the cutter will be travelling to the next location (180°).



14 POINT `x1.625 z-1.875`

Program a Point command to describe the intersection of two Lines (a sharp corner). Z value is (negative) 2 7/8 - 1, or -2-7/8+1.



15 LINE `angle90`

Enter the angle the cutter will be travelling to the next location (90°).

16 POINT `x2.25 z-1.875`

X value of this sharp corner is 2 1/4, or 2.25. Z value is -2-7/8+1.

17 LINE `angle180`

Enter the angle the cutter will be travelling to the next location (180°).

18 POINT `x2.25 z-2.5625`

X value of this sharp corner is 2 1/4, or 2.25. Z value is (negative) 2 7/8 - 5/16, or -2-7/8+5/16.

19 LINE `angle90`

Enter the angle the cutter will be travelling to the next location (90°).

20 POINT `x3.2425 z-2.5625`

X value of this corner is defined as a value between 3.240 and 3.245; so, enter the middle value 3.2425. Z value is (negative) 2 7/8 - 5/16 and it can be entered as -2-7/8+5/16.

21 LINE `angle180`

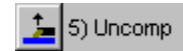
Enter the angle the cutter will be travelling to the next location (180°).

22 POINT `x3.2425 z-2.9`

Program this last point just past the finish Z location.

23 UNCOMP angle180

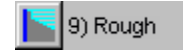
The Uncomp angle (180°) is the direction the tool was traveling at the final Point (Point from line 22). Note that the Uncomp angle is the same Line angle as line 21.



24 unROUGH

Turn the roughing definition off by programming another Rough command. After selecting the command 9) *Rough* (or 9 from the numeric keypad), it becomes *UnRough* in the Source.

Now, G-ZERO roughs the profile given between the ROUGH (line 4) and unROUGH (line 24) commands leaving the stock allowance defined in the Rough command.



2 Face and Finish OD

25 TOOL 2 in/out2 *** FACE AND FINISH OD

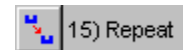
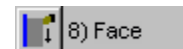
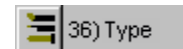
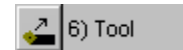
26 TYPE 1 mat1=TIN tnr.015 size.5

27 FACE x1.35 z.025 xcut-.03 zcut0 depth0 f6

28 REPEAT from5 thru23

With the FACE command, G-ZERO rapids to the start point, faces the part, rapids away from the face and returns to the start point.

To remove remaining stock in a finish pass, REPEAT the cutting path by reusing Source lines. Start the Repeat at Comp and end at UnComp.



3 Drill, countersink, tap

29 TOOL 3 in/out1 *** 37/64 DRILL FOR 5/8-18 TAP

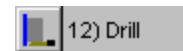
30 TYPE 29 mat0=M2 tnr0 size.578125

Define the tool for the next operation: drilling the center.

For the Type command, select a drill from the pop-up tool library. Then describe the drill material, tnr (0), and size (diameter). To enter a fractional drill size like $\frac{37}{64}$, type 37/64. The size chart displays both a size smaller (.562) and a size larger (.625) drill size. G-ZERO will select the feedrate of the size closest to $\frac{37}{64}$. If you will be using this drill size often, customize this Feed file.

31 DRILL g74 rap.1 cut-1.25 peck-2 tip1 ret.1 f.0066

A minus (negative) number of pecks triggers a full retract between pecks. To have G-ZERO automatically calculate and add the drill tip length to the cut depth, enter tip 1.



32 TOOL 4 in/out1 *** COUNTERSINK

33 TYPE 29 mat0=M2 tnr0 size.64

Define the tool for the next operation: countersinking the drill hole.

Select a countersink from the pop-up tool library. Then describe the countersink material, tnr (0 for a countersink) and size (diameter of the tool).

34 DRILL g74 rap.05 cut0 peck1 tip90 ret.1 f.003

These Drill prompts have special “flags” to automatically calculate how far to push the tool to get the correct diameter chamfer:

G-code G74

Depth of cut The top surface of the countersink (usually 0)

of pecks The full countersink angle (usually 90°)

Drill tip The diameter of the chamfer to be left on the part.

35 TOOL 5 in/out1 *** 5/8-18 TAP

36 TYPE 31 mat0=M2 tnr0 size.625

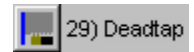
Define the tool for the next operation: tapping the center hole.

Select a tap from the pop-up tool library. Then describe the tap material, tnr (0 for a tap), and size (major diameter of the thread for a tap).

37 DEADTAP tpi18 zstart.5 zdepth-.75

With the DeadTap command, G-ZERO rapids to the specified startpoint and feeds to the depth at a feedrate appropriate for the specified TPI. It then reverses the spindle and feeds back to the startpoint.

Note: the zstartpoint should be a minimum of 2x the pitch of the thread.



HOUSING - SIDE B

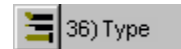
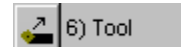
1 Define a material that was previously cut

- 1 MAT'L od3.25 id.578 zlim-1 zface.02 zclamp2.375 type4=ALUMINUM
- Typically, the front of the part is z0, so the back end dimensions will be negative (-) number. Because this part was drilled in *HOUSING -SIDE A*, use an id.578 to show where material was already removed. (See *drilling tool size in Housing - Side A.*)
- zlim-1** and **zclamp2.375** because the majority of the material in in the chuck. You will need to press the *O* key to override the zclamp amount because it is a very large number.

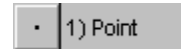


2 Angled face and finish OD

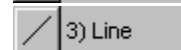
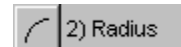
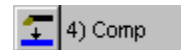
- 2 TOOL 1 in/out2 *** ANGLED FACE AND FINISH OD
- 3 TYPE 1 mat1=TIN tnr.015 size.5
- Define the tool for the first operation: angle the face and finish ID.



- 4 POINT x3.3 z-.32
- This “approach” or “safety” Point brings the tool just a little off the part before it begins cutting along the contour compensating for tool nose radius (tnr).



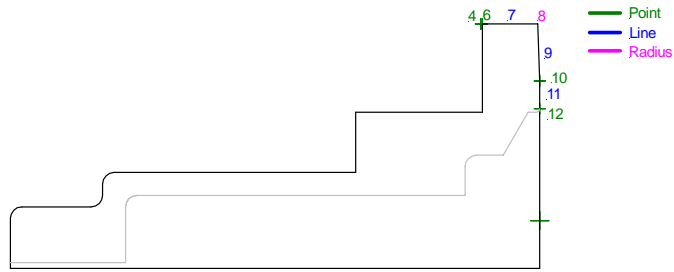
- 5 COMP ang0 lft/rt1 f6
- Describe the contour of the face. To cut the angled face, the tool must be the left (lft/rt1) of the part. The Comp angle of 0° is the angle between the Point (line 6) and Radius (line 8).



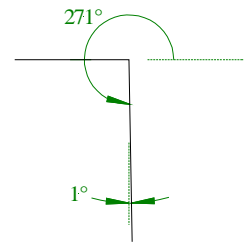
- 6 POINT x3.2425 z-.3125
- This approach point brings the tool onto the workpiece. Program print dimensions; do not add offsets for tool nose radius because G-ZERO makes all cutter compensation calculations automatically once Comp is programmed.
- X value of this point is the mid value of the maximum (3.245) and minimum (3.240) values shown on blueprint.
- Z value is entered as -5/16.

- 7 LINE angle0
- Enter the angle the cutter will be travelling to the next location (0°).

- 8 RADIUS .005 type0 (xc3.2325 zc-.01030271)
- Enter the first Radius of the OD. This *very small*, outside (+) Radius is unknown (type0). Program small Radii (type2) instead of Points to debur corners, saving a secondary operation.



The numbers in this illustration correspond to the line numbers in the source code below.



9 LINE angle271
Enter the angle the cutter will be travelling to the next location (271°).
See graphic on the right.

10 POINT x2.625 z0
X value of this point is $2 \frac{5}{8}$; enter it as 2+5/8 and let G-ZERO do the calculation. Z value is 0.



11 LINE angle270
Enter the angle the cutter will be travelling to the next location (270°).

12 POINT x1.1 z0
Program the X value of this point in 1.1. Z value is 0.

13 UNCOMP angle270
The Uncomp angle (270°) is the direction the tool was travelling between Point (line 10) and Point (line 12).

3 Carbide insert drill

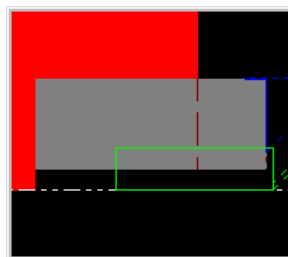
14 TOOL 2 in/out1 *** CARBIDE INSERT DRILL
15 TYPE 17 mat0=TIN trr0 size1.25
16 DRILL g74 rap.1 cut-2.2 peck3 tip180 ret.1 f.003

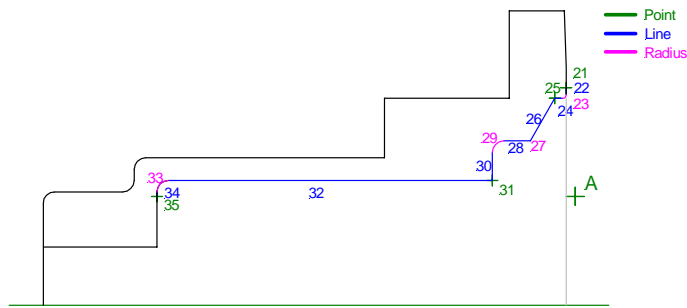


Define the tool for the next operation: drilling the center.

Because of the large ID radius at the bottom of the cavity, drill to only 2.2" depth.

At the *add tip?* prompt, enter the included angle of the indexable insert drill (180 for 180°). You will have to press "o" for override immediately after typing 180.





The numbers in this illustration correspond to the line numbers in the source code below.

4 Rough ID

17 TOOL 3 in/out1 *** ROUGH ID

18 TYPE 1 mat1=TIN tnr.032 size.375

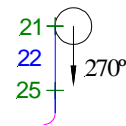
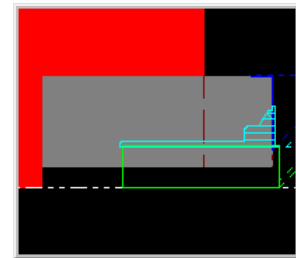
19 ROUGH xstk.01 zstk.003 depth.1 x1.2 z.05 safang90 f4

Define the tool for the next operation: roughing ID.

Start roughing from point A at x=1.2 and z=.05 (see illustration above).

20 COMP ang270 lft/rt1 f6

Define contour using Comp, Point, Radius, Line, and Uncomp. To force the tool to travel down, the Comp angle is 270° (tangent angle to the Point on the first contour).



21 POINT x2.4 z0

This Point brings the tool onto the contour. Program print dimensions; do not add offsets for tool nose radius because G-ZERO makes all cutter compensation calculations automatically once Comp is programmed.

22 LINE angle270

Enter the angle the cutter will be travelling (270°) between Point (line 21) and Radius (line 23).

23 RADIUS .0175 type2 x2.28125 z0 (xc2.31625 zc-.0175)

This radius is shown on the blueprint as .015-.020, so enter their midpoint .0175 as radius. This is an outside (+) radius and it's dimensioned as a corner (type2) between 2 intersecting Lines.

X value can be entered as 2+9/32. Z value is 0.

24 LINE angle180

Enter the angle the cutter will be cutting towards the next location (180°).

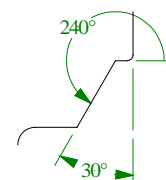
25 POINT x2.28125 z-.0625

Program a Point to describe the intersection of 2 Lines (sharp corner).

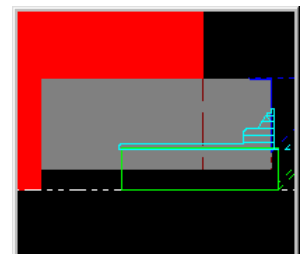
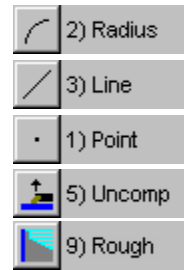
X value can be entered as 2+9/32. Z value is -1/16.

26 LINE angle240

Enter the angle the cutter will be cutting towards the next location (240°).



- 27 RADIUS .001 type0 (xc1.8145 zc-.1983938)
This *very small* outside (+) Radius is unknown (type0).
- 28 LINE angle180
Enter the angle the cutter will be cutting towards the next location (180°)
- 29 RADIUS -.0625 type2 x1.8125 z-.40625 (xc1.6875 zc-.34375)
This inside (-) Radius is dimensioned as a corner (type2). Its radius can be entered as -1/16.
X value can be entered as 1+13/16. Z value is -13/32.
- 30 LINE angle270
Enter the angle the cutter will be cutting towards the next location (270°)
- 31 POINT x1.375 z-.40625
Describe this Point as the intersection of two Lines (a sharp corner).
- 32 LINE angle180
Enter the angle the cutter will be cutting towards the next location (180°).
- 33 RADIUS -.0625 type2 x1.375 z-2.25 (xc1.25 zc-2.1875)
This inside (-) Radius is dimensioned as a corner (type2). Its radius can be entered as -1/16. X value of the center of the radius is 1+3/8; Z value is -2-1/4.
- 34 LINE angle270
Enter the angle the cutter will be cutting towards the next location (270°).
- 35 POINT x1.2 z-2.25
Describe this last Point on the ID.
- 36 UNCOMP angle270
The Uncomp angle (270°) is the direction the tool was travelling at the final Point (line 35).
- 37 unROUGH
Turn the roughing definition off by programming another Rough command. After selecting the 9) Rough command, it becomes UnROUGH in the Source. Now, G-ZERO roughs the profile given between the Rough (line 19) and UnRough (line 37) commands leaving the stock allowance defined in the Rough command.
- 38 POINT x1.2 z.2 f0
Program this “retract” point after the Rough command to ensure that the tool does not crash into the part on its way to the start of the finish pass. (A retract Point is often needed when doing ID work.) A feedrate of 0 (zero) moves the tool to this Point at rapid.



5 Finish ID - Facing with ID tool

39 TOOL 4 in/out1 *** FINISH ID

40 TYPE 4 mat1=TIN tnr.015 size.375

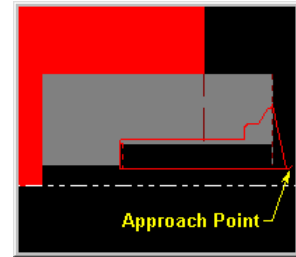
Define the tool to face and finish the ID.

41 POINT x.5 z.2

This “approach” Point ensures that the tool does not crash into the part before facing.

42 FACE x.5 z-2.2 xcut1.25 zcut-2.247 depth0 f5

Use Face to quickly finish the back wall of the cavity, eliminating 4 or 5 long roughing pass that would be cutting air. When using Face with an ID tool, you will have to press “o” for Override after typing a positive xcut value.



 Override

43 POINT x.5 z.2 f0

This “retract” Point ensures that the tool will not crash into the part before repositioning for the contour finish pass. A feedrate of 0 moves the tool to this Point at rapid.

44 REPEAT from20 thru34

To remove remaining stock for the Roughed and Faced ID material, REPEAT the cutting path by reusing source lines. Start the Repeat at Comp (line 20) and end at line 34 (2 lines before Uncomp).

45 POINT x.5 z-2.25

Describe the last location on the ID (part of the Face move).

46 UNCOMP angle270

The Uncomp angle (270°) is the direction the tool was travelling at the final Point (line 45).

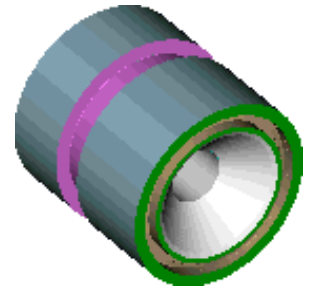
47 POINT x.5 z.2 f0

Program this “retract” Point after Uncomp to ensure that the tool will not crash into the part on its way to the tool change position. A retract Point is often needed when doing ID work. A retract Point is often needed when doing ID work. A feedrate of 0 moves the tool to this Point at rapid.



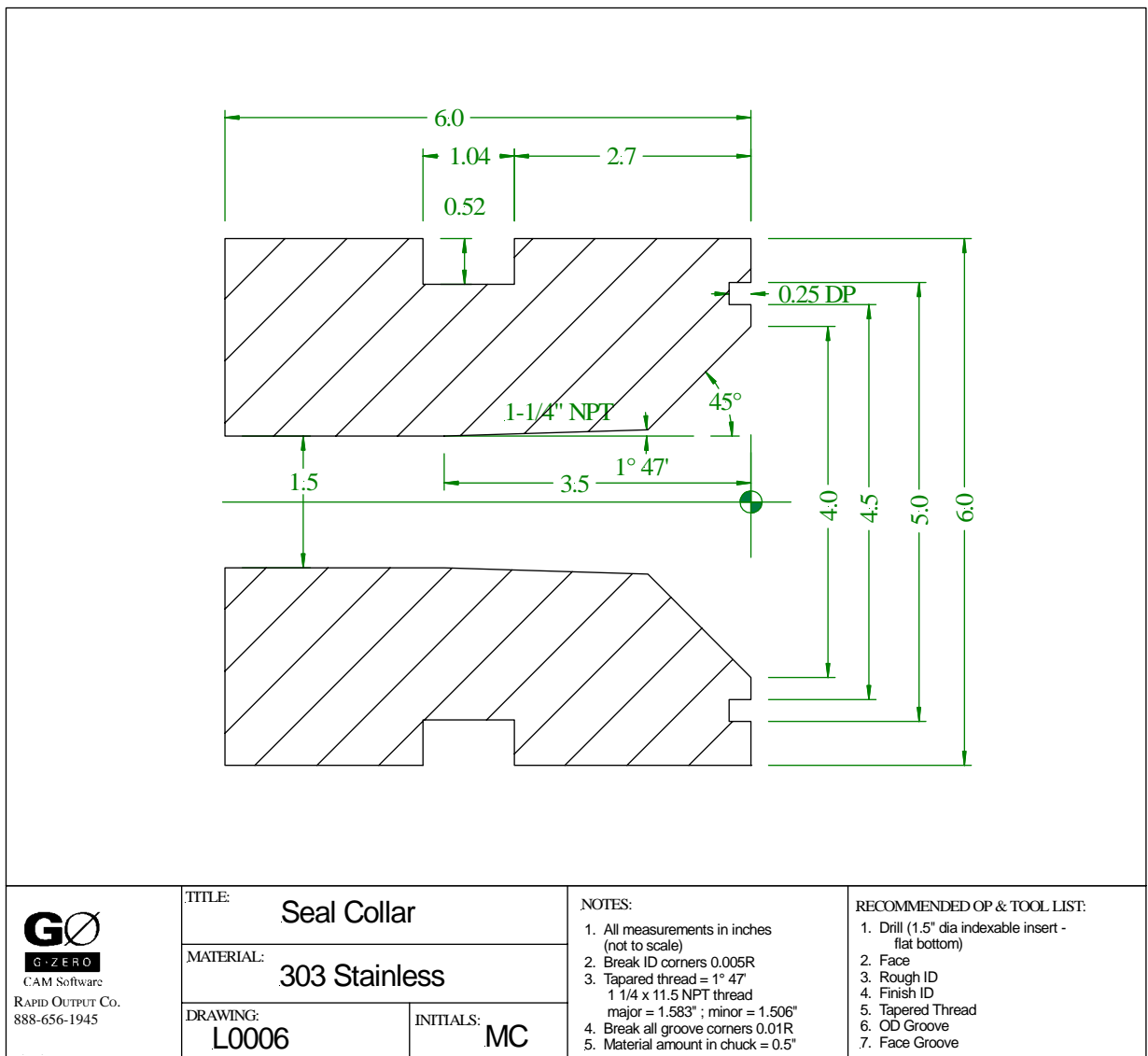
Project 8

Seal Collar



What you will learn:

- ◆ Convert radial degree to decimal degree
- ◆ Use Pipe and Thread commands for tapered threading
- ◆ Use Groove command for OD grooving
- ◆ Use FGroove command for face grooving



1 Define material - Drill central hole - Face part

1 MAT'L od6 id1 zlim-6.1 zface.03 zclamp.5 type1=303

Program the Material command to define the dimensions of the stock.
Pay attention on the following:

od 6 diameter of the bar stock (or a little larger)
id 1 predrilled material with id=1"
zlim -6.1 length sticking out of chuck (6") plus a safe amount so that the tooling doesn't crash into the chuck (0.1").
This value is negative (-) because the front of the part (face) is z0.



2 TOOL 1 in/out1 *** DRILL

3 TYPE 17 mat0=TIN tnr0 size1.5

Define the tool for the first operation: drilling.

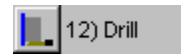
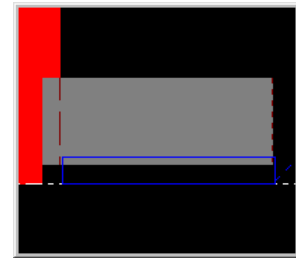
Tool's nose radius (tnr) is always 0 when drilling. If you don't have tool 17 (flat bottom drill) add it to your tool library. (*See Reference Manual*)

4 DRILL g74 rap.1 cut-6.02 peck-6 tip180 ret.1 f.0036

Enter a *cut* value of -6.02; and extra .02" to make sure it drills through the material.

A negative number of pecks triggers a full retract between pecks.

At the *add tip?* prompt, enter the included angle of the indexable insert drill (180 for 180° - flat bottom drill). Need to press "o" for override immediately after typing 180.



5 TOOL 2 in/out2 *** FACE

6 TYPE 1 mat1=TIN tnr.032 size.375

7 FACE x6.1 z.035 xcut1.4 zcut0 depth0 f5

Define the tool for the next operation: facing OD.

With the Face command, G-ZERO rapids to the start point (x=6.1, z=.035), faces the part, rapids away from the face and returns to the start point. Enter a positive xcut because the material was previously drilled. Note that xcut looks for a negative x value so you will have to press "o" for override.



2 Rough and finish ID -- Convert radial to decimal degrees

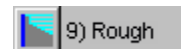
8 TOOL 3 in/out1 *** ROUGH ID

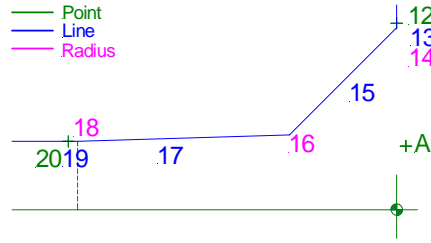
9 TYPE 2 mat1=TIN tnr.032 size .375

10 ROUGH xstk.01 zstk.005 depth.1 x1.4 z.1 safang90 f4

Define the tool for roughing the inside (ID) of the part.

Start roughing from point A at x=1.4 and z=.1 (*See next illustration*)





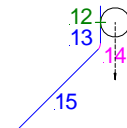
The numbers in this illustration correspond to the line numbers in the source code below.

11 COMP

`ang270 lft/rt1 f5`

Before defining the contour to be roughed, program the Comp command to tell G-ZERO to calculate offsets for cutter radius.

This Comp angle (270°) is the compass direction between the Point (line 12) and the Radius (line 14).



12 POINT

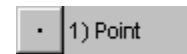
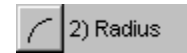
`x4.1 z0`

This Point brings the tool onto the contour.

13 LINE

`angle270`

The line angle 270° pushes the tool down the face. Notice that this Line angle is the same as the Comp angle.



14 RADIUS

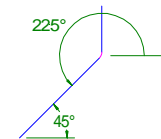
`.005 type2 x4 z0 (xc4.004142 zc-.005)`

This very small outside (+) Radius is dimensioned as a corner (type2) between two lines. This is to debur the corner.

15 LINE

`angle225`

Enter the angle the cutter will be travelling towards the next location (225°).



16 RADIUS

`.005 type0 (xc1.654479 zc-1.179832)`

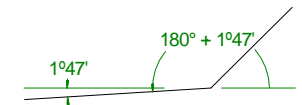
To calculate and break the edge of this undimensioned Point, define it as a very small, outside (+) Radius dimensioned as an unknown (type0).

17 LINE

`angle181.7833`

To program an angle dimensioned in degrees/minutes/seconds, type in the angle using the format **dd.mmss** followed by the quote key (") to convert to decimal.

In this case, to convert 1°47', type 1.4700" and G-ZERO will immediately convert it to the decimal value 1.783333. Now add 180°: 1.783333+180 and G-ZERO enters the value 181.7833.



18 RADIUS

`.005 type2 x1.5 z-3.5 (xc1.51 zc-3.500078)`

To calculate and break this Point, define it as a very small, outside (+) Radius dimensioned as a corner (type2) between two lines.

19 LINE

`angle180`

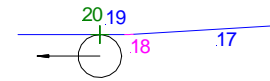
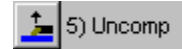
Enter the angle the cutter will be travelling towards the next location (180°).

20 POINT x1.5 z-3.6

Describe the last location on the contour.

21 UNCOMP angle180

The Uncomp angle (180°) is the direction the tool was travelling at the final Point (line 20).



22 unROUGH

Turn the roughing definition off by programming another Rough command. G-ZERO now roughs the profile given between the Rough (line 10) and UnRough (line 22) commands leaving the stock allowances defined in the Rough command.



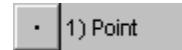
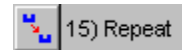
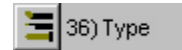
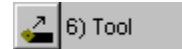
23 TOOL 4 in/out1 *** FINISH ID

24 TYPE 2 mat1=TIN trr.015 size.375

25 REPEAT from11 thru21

Define the tool for the next operation: finish the ID.

To remove remaining stock in a finish pass, Repeat the cutting path by reusing Source lines. Start the Repeat at Comp (line 11) and end at Uncomp (line 21).

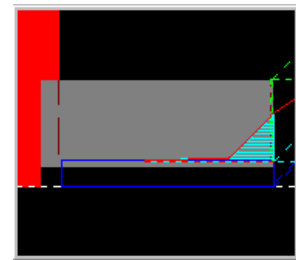


26 POINT x1.4 z-3.6 f.01

This Point moves the tool safely off the part before retracting out of the ID.

27 POINT x1.4 z.1 f0

This “retract” Point safely brings the tool out of the ID before returning to the tool change position. A retract point is almost always needed for ID work. A feedrate of 0 moves the tool to this Point at rapid.



3 ID pipe (tapered) threading

28 TOOL 5 in/out1 *** PIPE THREADING

29 TYPE 15 mat1=C-5 trr0 size11.5

Define the tool for the next operation: cutting a tapered thread.

30 POINT x1.4 z.2

This “approach” Point ensures that the tool will not hit the part. An approach point is often needed when doing ID work.

31 PIPE ret0 tpf.7472 x1.5 z-3.5

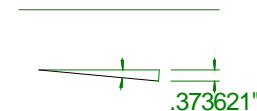
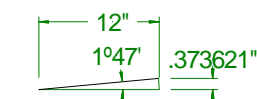
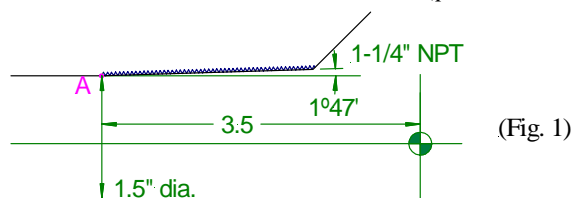
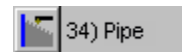
Pipe command is always programmed just before the Thread command.

ret 0 straight retraction

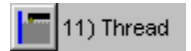
tpf .7472 number of inches per foot. This is the difference in diameter (in inches) for every foot of thread length: .3736+.3736

x 1.5 pitch diameter: major diameter (point A in Fig. 1)

z -3.5 Z finish dimension of the threads (point A in Fig. 1)



32 THREAD maj1.583 tpi11.5 min1.506 S905 z-1.17 zfin-3.5 num10 a60



G-ZERO rapids to the start point of the thread then feeds the specified number of times to the finish point (at a feedrate appropriate for the TPI).

maj 1.583 major diameter of the thread. See blueprint note
tpi 11.5 number of threads per inch. See blueprint note
min 1.506 minor diameter of the thread. See blueprint note
S 905 spindle speed (rpm)
z -1.17 z value of threading start point
zfin -3.5 z value of threadint end point
num 10 number of passes to cut the thread
a 60 angle of thread

33 POINT x1.4 z.2 f0

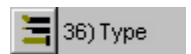
Program a “retract” Point to ensure that the tool will not hit the part.

4 OD groove

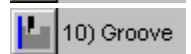
34 TOOL 6 in/out2 *** OD GROOVE



35 TYPE 9 mat0=TIN tnr0 size.25



Define the tool for the next operation: OD groove.



36 GROOVE x6 z-2.7 dp.52 w1.04 skip0 r.01 ir.01 a0 f2

x 6 x dimension of the groove surface
z -2.7 z dimension of the right-most edge
dp .52 depth of groove
w 1.04 width of groove
skip 0 to radius both top edges of the groove answer “0” here
r .01 outside corner of groove
ir .01 radius in bottom of groove
a 0 angle of walls
f 2 feedrate

5 Face groove

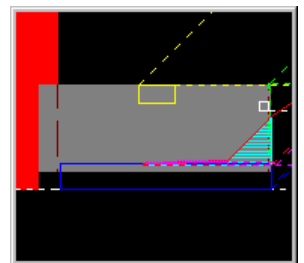
37 TOOL 7 in/out2 *** FACE GROOVE

38 TYPE 14 mat0=TIN tnr0 size.125

Define the tool for the next operation: face groove

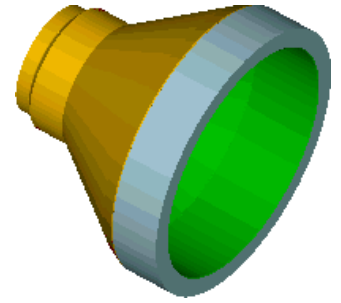
39 FGROOVE x4.5 z0 dp.25 w.25 skip0 r.01 ir.01 a0 f.007

x 4.5 x dimension of the groove ID
z 0 z dimension of the groove’s face (usually 0)
dp .25 depth of groove
w .25 width of groove
skip 0 to radius both top edges of the groove answer “0” here
r .01 outside corner of groove
ir .01 radius in bottom of groove
a 0 angle of walls
f .007 feedrate



Project 9

Funnel



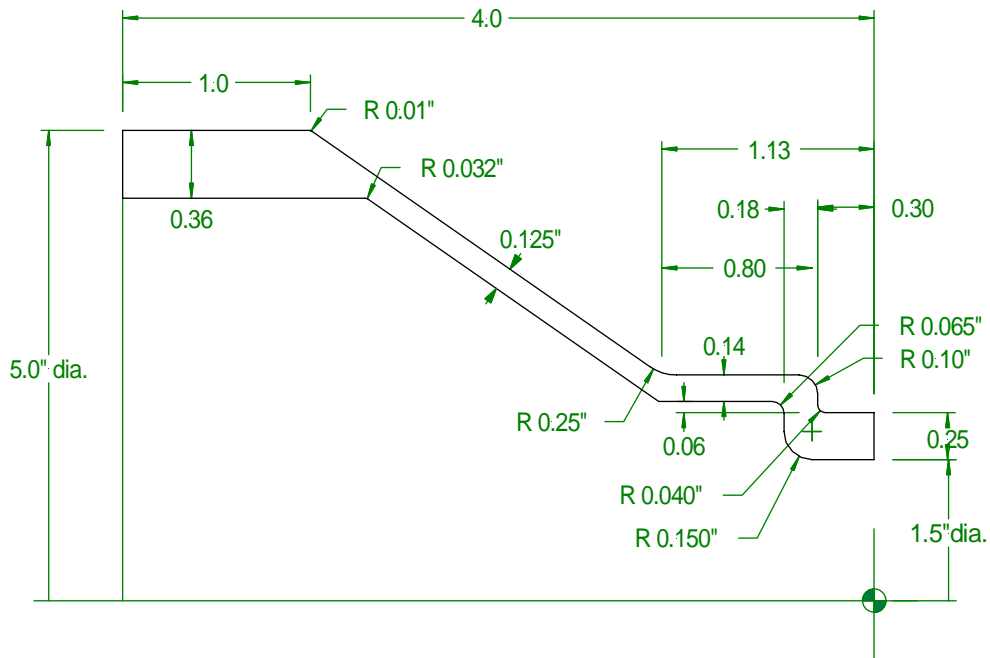
What you will learn:


Funnel - Side A: machine OD

- ◆ How to program an unknown Line

Funnel - Side B: machine ID

- ◆ How to flip a part
 - Multiple Mat'l command
 - Negative Tool number
- ◆ Use of %OFF (disactivate G-code) and %ON (activate G-code)
- ◆ Use Rotate command
- ◆ Use Line-Point-Line technique

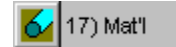


 RAPID OUTPUT Co. 888-656-1945	TITLE: Funnel	NOTES: 1. All measurements in inches (not to scale) 2. Break sharp corners 0.05R	RECOMMENDED OP & TOOL LIST: SIDE A 1. Drill (1.5" dia inexecutable insert drill) 2. Rough OD 3. Finish OD SIDE B 1. Rough ID 2. Finish ID
	MATERIAL: Aluminum		
	DRAWING: L0007 INITIALS: MC		

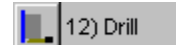
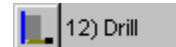
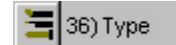
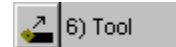
FUNNEL - SIDE A (OD)

Machine OD of part

1 MAT'L od5 id0 zlim-3.5 zface0 zclamp.5 type4=ALUMINUM
 Define the material using dimensions from the blueprint in the beginning of this project.



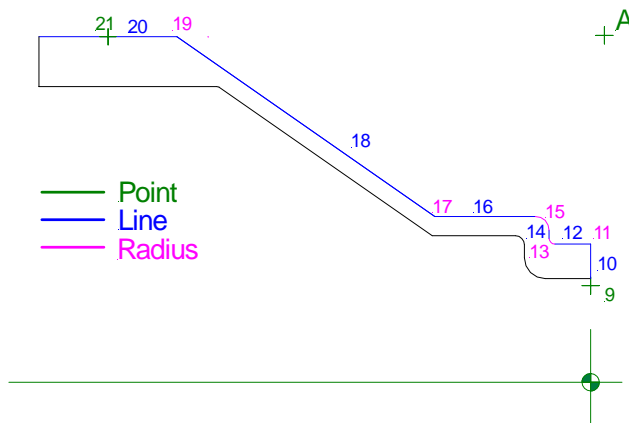
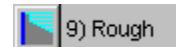
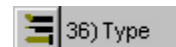
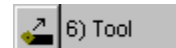
2 TOOL 1 in/out1 *** DRILL
 3 TYPE 17 mat0=TIN tnr0 size1.5
 Define the tool for the first operation: drilling.



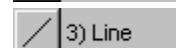
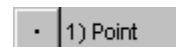
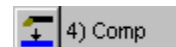
4 DRILL g74 rap.05 cut-4.05 peck-3 tip180 ret.1 f.004
 Enter the parameters necessary for drilling a hole. A minus (-) number of pecks triggers a full retract between pecks. At the *Would you like the drill tip added?* prompt, enter the included angle of the indexable insert drill (180 for 180° -- flat bottom).

Override if G-ZERO beeps

5 TOOL 2 in/out2 *** ROUGH OD
 6 TYPE 2 mat1=TIN tnr.032 size.375
 7 ROUGH xstk.01 zstk.005 depth.1 x5.02 z.1 safang270 f3
 Define the tool for the next operation: roughing the OD.
 Start roughing from point A at x=5.02 and z=.1 (*See illustration below*)



8 COMP ang90 lft/rt2 f6
 This Comp angle (90°) is the compass direction between the Point (line 9) and the Radius (line 11).
 When roughing, the Comp feedrate is for the finish cut only.



9 POINT x1.4 z0
 Program this first Point on the contour on x=1.4 and z=0.

10 LINE angle90
 Enter the angle the cutter will be travelling towards the next Radius (90°).

11 RADIUS .05 type2 x2 z0 (xc1.9 zc-.05)
Define this outside (+) Radius by its corner (type2). You can enter the corner's x value as the calculation $.25 * 2 + 1 . 5$.



12 LINE angle180
Enter the angle the cutter will be travelling towards the next location (180°)

13 RADIUS -.04 type2 x2 z-.3 (xc2.08 zc-.26)
Define this inside (-) Radius by its corner (type2).

14 LINE angle90
Enter the angle the cutter will be travelling towards the next location.

15 RADIUS .1 type2 x2.4 z-.3 (xc2.2 zc-.4)
Define this outside (+) Radius by its corner (type2). You can enter the corner's x value as the calculation $.14 * 2 + .06 * 2 + .25 * 2 + 1 . 5$.

16 LINE angle180
Enter the angle the cutter will be travelling towards the next location (180°)

17 RADIUS -.25 type2 x2.4 z-1.13 (xc2.9 zc-1.051639)
Define this inside (-) Radius by its corner (type2).

18 LINE angle (145.1935)
Unknown Line. The angle of this Line is unknown (not dimensioned). By typing only the open parenthesis `[]`, G-ZERO automatically calculates the unknown angle after the next known location is given (after you enter the values for the next Radius on line 19).

19 RADIUS .01 type2 x5 z-3 (xc4.98 zc-3.003134)
Define the outside (+) Radius by its corner (type2). You can enter the corner's z value as the calculation $-4 + 1$.

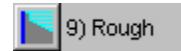
20 LINE angle180
Enter the angle the cutter will be travelling towards the next location (180°).

21 POINT x5 z-3.4
Program this Point to move the tool to the final Z dimension on the OD at $x=5$ and $z=-3.4$. (In this program, we assume that the original material was cut and finished to required size; therefore, we are not roughing/finishing the section that is clamped plus extra room to avoid tool crash.)

22 UNCOMP angle90
Stop compensating (like G40) for cutter radius.

23 unROUGH

Turn the roughing definition off by programming another Rough command. G-ZERO roughs the profile given between Rough (line 7) and UnRough (line 23) leaving the stock allowance defined in the Rough command.

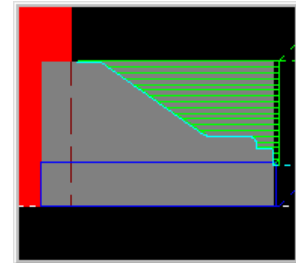


```
24 TOOL 3 in/out2 *** FINISH OD
25 TYPE 2 mat1=TIN trr.015 size.375
26 REPEAT from8 thru22
```

Change tools to finish the OD.

Finish the OD contour by repeating the cutter path defined by the Comp and Uncomp between the Rough commands.

%M0 (You can program this comment line here to stop the spindle, OR program a negative Tool as shown in line 40)

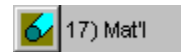


2 Flip part, machine ID

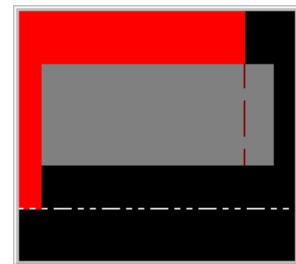
```
27 MAT'L od5 id1.5 zlim-.5 zface0 zclamp3.5 type4=ALUMINUM
```

Multiple Mat'l Command. Because we are going to flip the part to work on the other end, program a new Material command. You will need to press the "o" key for override after immediately typing 17 to invoke the Mat'l command since a Mat'l command has already been programmed in line 1.

Using multiple Material commands within one source program is useful only if using same jaws for both sides. If both sides do not use the same jaws, program two separate sources.



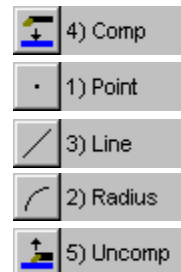
override if G-ZERO beeps



```
28 FLIP PART, MACHINE ID
```

To add a comment like FLIP PART any time during the source program, just start typing your comment. If you want a comment to be embedded in your tape for the operator, start with a and enclose it in parenthesis. For example: `%(FLIP PART HERE)`

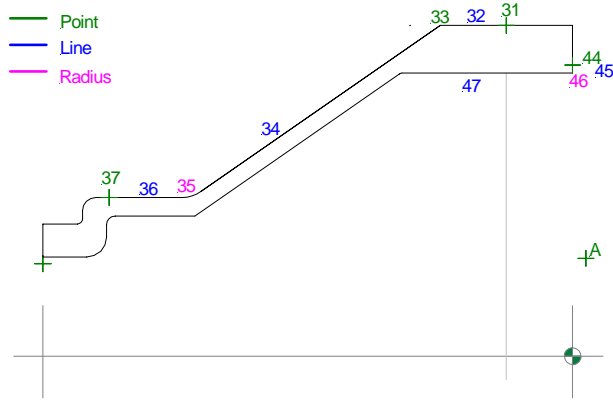
```
29 %OFF
30 COMP ang180 lft/rt1 f5
31 POINT x5 z-.5
32 LINE angle180
33 POINT x5 z-1
34 LINE angle215
35 RADIUS .02 type0 (xc3.04 zc-2.434454)
36 LINE angle180
37 POINT x3 z-3
38 UNCOMP angle270
39 %ON
```



%ON %OFF. To see what was programmed before you flipped the part, program a contour (or a portion of a contour) without generating any G-code.

All commands entered between the *comments* %OFF (disactivate G-code) and %ON (activate G-code) will not create any G-code.

Note: Reference-only information can also be shown on-screen if it is programmed before the first TOOL command.



```
40 TOOL -4 in/out1 *** ROUGH ID
41 TYPE 2 mat1=tin tnr.032 size.375
```

Negative tool number = program stop. Change tools to rough the ID. Program a negative (minus) tool number to trigger a M0 for a Program Stop to flip the material.

```
42 ROUGH xstk.01 zstk.005 depth.1 x1.48 z.1 safang90 f3
```

Start roughing from point A at x=1.48 and z=.1 (*See above illustration*)

```
43 COMP ang270 lft/rt1 f5
```

Define the contour to be roughed. Program Comp instead of calculating offsets for cutter radius. This Comp angle (270) was the compas direction between the Point (line 44) and the Radius (line 46). The cutter is on the right. The Comp feedrate is for the finish cut only. (Comp is always followed by a Point or a Radius type1).

```
44 POINT x4.4 z0
45 LINE angle270
46 RADIUS .01 type2 x4.28 z0 (xc4.3 zc-.01)
47 LINE angle180
```

POINT on line 44 is the first point on the contour.

RADIUS on line 46: You can enter the corner's x value as the calculation $5 - .36 - .36$.

```
48 RADIUS -.032 type0 (xc4.216 zc-1.288828)
49 ROTATE angle34.8065 xpiv5 zpiv-1
50 LINE angle180
51 POINT x-.25 z-1
52 LINE angle180
53 ROTATE angle0 xpiv0 zpiv0
```

We don't have much information on the angled line. All we know is that it is parallel (at 0.125") to the OD angled line programmed on line 18.

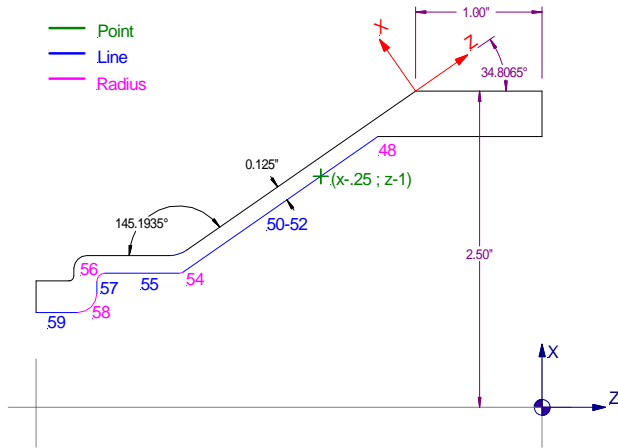
override if G-ZERO beeps

	6) Tool
	36) Type
	9) Rough
	4) Comp
	1) Point
	3) Line
	2) Radius
	16) Rotate

Rotate. We can shift and rotate the center of coordinates (*blue coordinates - see following illustration*) to the new temporary origin (*red coordinates*), and program a line with angle 180°.

To calculate the Rotate angle, use the unknown Line angle calculated on line 18 of this source (145.1935) and subtract it from 180 (you can enter the calculation 180-145.1935 and let G-ZERO do the calculation). The new center of coordinates is located at 5" (dia.) in X and 1" in Z, so xpiv=5 and zpiv=-1.

This Rotate command (line 49) rotates all later dimensions 34.8065° counterclockwise (+) around the pivot xpiv5 and zpiv-1 until G-ZERO encounters another Rotate command with zero values (line 53).



Line-Point-Line. Radii on lines 48 and 54 are unknown (type=0) because the blueprint does not provide the (x;z) locations of these radii. We cannot program two consecutive unknown radii even if they are separated by a line.

However, we can easily find the (x;z) values of a point within the angled line and insert it as a “fake” point. This point could be x=-.25 and z=-1. (Note that the x value of -.25 is a diameter value)

LINE commands from lines 50 and 52 should be identicals because they are in fact the same line.

This technique of programming a Point between two unknown Radii on a Line is called Line-Point-Line. The format is:

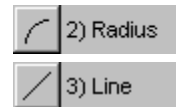
```

Radius A ..... unknown
Line X..... direction from unknown Radius A to unknown Radius B
Point ..... pick any point on Line X between Radii A and B
Line X..... remaining half of Line X; therefore, same angle
Radius B ..... unknown
  
```

G-ZERO automatically calculates the locations of the two Radii using the Line-Point-Line method.

54 RADIUS .05 type0 (xc2.22 zc-2.868071)

To break this sharp corner, program a radius .05. This is an unknown radius (type0) since the blue print does not show its x;z location.



55 LINE angle180

56 RADIUS -.065 type2 x2.12 z-3.52 (xc1.99 zc-3.455)

57 LINE angle270

58 RADIUS .15 type2 x1.5 z-3.52 (xc1.8 zc-3.67)

59 LINE angle180

60 POINT x1.5 z-4.1

This point moves the tool to the final Z-dimension of the bore.

61 UNCOMP angle180

Stop compensating (like G40) for cutter radius. The Uncomp angle (180°) is determined by the direction between the Radius (line 58) and Point (line 60).

62 unROUGH

Turn the roughing definition off by programming another Rough command. After selecting Rough from the menu, it becomes unRough in the Source. Now, G-ZERO roughs the profile given between Rough (line 42) and UnRough (line 62) leaving the defined stock allowance.

63 TOOL 5 in/out1 *** FINISH ID

64 TYPE 2 mat1=TIN tnr.015 size.375

65 REPEAT from43 thru61

Finish the ID contour by repeating the cutter path defined by the Comp and UnComp commands between the Rough/UnRough commands. Always Repeat Source Program lines instead of retyping!

66 POINT x1.25 z.1 f0

Insert a “retract” Point to ensure that the tool will not hit the part on its way to the tool-change position. A “retract” Point is usually needed when doing ID work. A feedrate of 0 moves the tool to the Point at rapid. ■



