

G-ZERO Lathe CAD Reader



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CAD Reader

G-ZERO CAD Import Interface

CAD Reader transfers coordinates from CAD drawing files directly into the G-ZERO source program. Just click on the elements of a CAD drawing to get their dimensions and coordinates instead of digging this information out of a blueprint and keying it in.

F9: Open/Close

Press the F9 function key to display the G-ZERO CAD import interface; press the F9 function key again to hide it.

If there is no CAD file defined in your source program, G-ZERO will guide you to open one. Make sure to select the correct file format in the "Files of type" box.

Source link

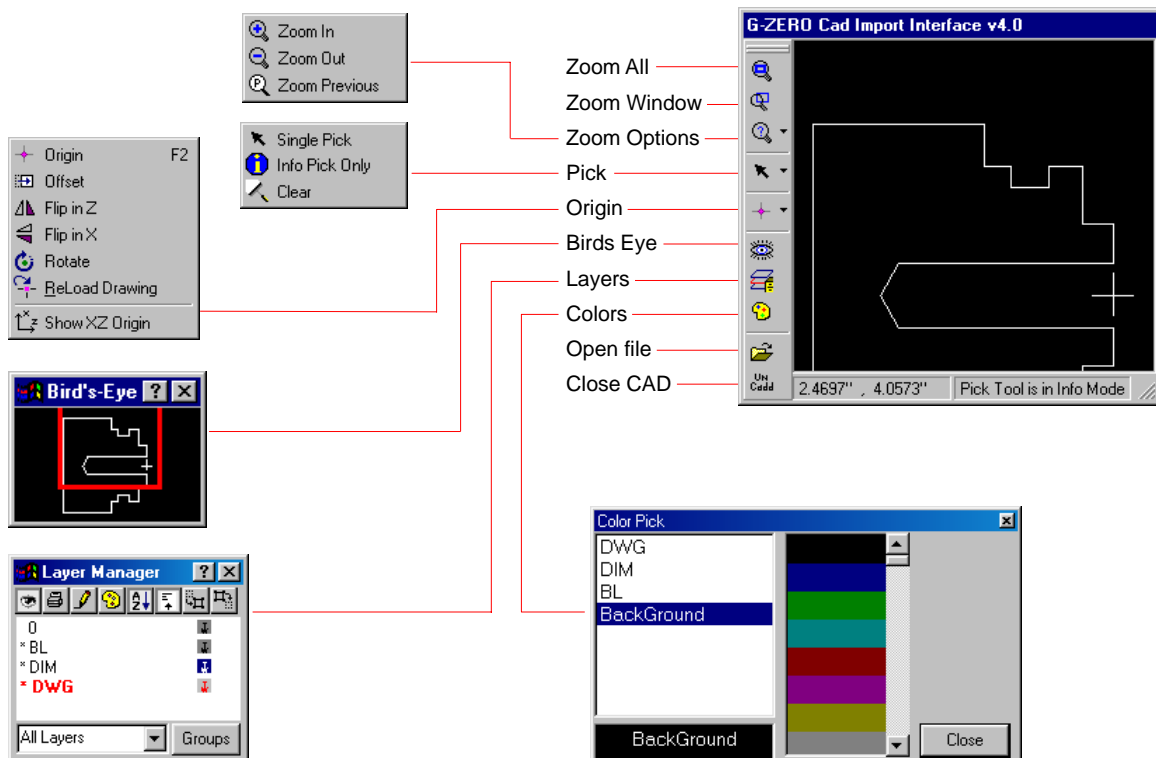
When you first activate CAD Reader, a comment line is added into your source program to establish a link with the selected CAD file. Example: CAD C:\LATHE\0002.dxf

File format

CAD Reader can directly import data from these file formats: AutoCAD DXF (.dxf), Visual CADD (.vcd), Generic CADD (.gcd), and AutoCAD DWG (.dwg).

Toolbar

By default, CAD Reader's toolbar is attached to the left border of the CAD Import Interface window. You can detach it (click and drag the double-lines at the top of the toolbar) and place it anywhere you want, or attach it to another side of the CAD Import Interface window.



Zoom All

Displays the complete CAD drawing in the current window.



Zoom Window

Displays the rectangular area selected (zoom in the selected area). After selecting this option, click the left button of your mouse over first corner of rectangle, drag the yellow box to the opposite corner, and release the mouse button.



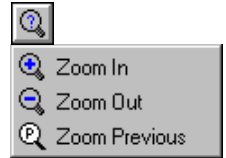
Zoom Options

Displays a list with three options:

ZOOM IN. Displays larger graphic (less area) in the same size viewport.

ZOOM OUT. Displays smaller graphic (more area) in the same size viewport.

ZOOM PREVIOUS. Displays the previous zoom setting.



Pick

Displays a list with three options:

SINGLE PICK. Position mouse cursor near an element (line, arc or circle) in the CAD graphic, and click the left button of your mouse. G-ZERO finds the CAD element closest to the cursor, changes its color, and inserts its information into the source program.

Pick each element in the order in which it is to be machined. Make sure to click on the outside of an arc to cut around the outside (+), or on the inside of the arc for an inside (-) cut.

INFO PICK ONLY. When this option is active, the status bar displays "Pick Tool is in Info Mode." Use your mouse to click on any element in your CAD file and see its information displayed on the status bar.

Example 1:

Arc -1.1000,1.3500 r=-0.1000 sa270.0000 ma90.0000

↓	↓	↓	↓	↓
z value of center of arc	x value of center of arc	radius of arc (negative if clicked inside)	start angle of arc	move angle of arc (degrees of arc sweep)

If you single picked this element, your source code would be:

POINT x2.7 z-1.1 f99.99 (x value is diameter)

If you single picked this element (clicked inside radius), and you are programming between *COMP/UNCOMP* commands, your source code would be:

RADIUS -1 type1 x2.7 z-1.1

Example 2:

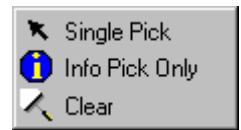
Line p1=-1.4000,1.2500 pt2=-1.1000,1.2500 Len=0.3000 np=p2

↓	↓	↓	↓	↓	↓
z value of 1 st point	x value of 1 st point (radius)	z value of 2 nd point	x value of 2 nd point (radius)	length of line	nearest endpoint to mouse click

If you single picked this element, your source code would be:

POINT x2.5 z-1.1 f99.99 (x value is diameter)

CLEAR. Clears any element selected.



Origin Options

Displays a list with seven choices:

ORIGIN - F2. Select this option (or press the F2 function key) to set the origin of coordinates of the CAD file to match it with the one the machinist is using. You will be asked to “Pick a line for Z-axis origin” (a line that contains the Z axis origin), and then, “Now pick a line for the X-axis” (a line that contains the X axis origin). The intersection of these two lines will be the new origin of coordinates from which all data are based on.

OFFSET. Offsets the origin of coordinates. You will be asked to “Enter the amount to offset the X-axis” and then “Enter the amount to offset the Z-axis.”

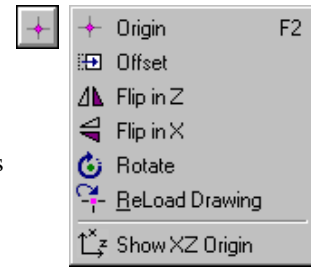
FLIP IN Z. Flips the entire CAD drawing horizontally in the Z axis.

FLIP IN X. Flips the entire CAD drawing vertically in the X axis.

ROTATE. Enter the angle of rotation (center of rotation is origin of coordinates) to rotate the entire CAD drawing.

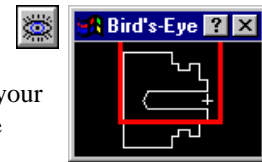
RELOAD DRAWING. Reads and displays all information from the original CAD file.

SHOW XZ ORIGIN / HIDE XZ ORIGIN. Displays or hides the XZ coordinates.



Toggle Birds Eye View

Displays (or hides) a small window that shows the entire CAD drawing with the currently displayed section marked with a red rectangle. Click the right button of your mouse (with cursor located on Birds Eye View), to move, resize or close Birds Eye View (system), display or hide a title bar, update view, and change the zoom of the area to be displayed in the CAD Import Interface window (size of red rectangle).



Toggle display of Layers

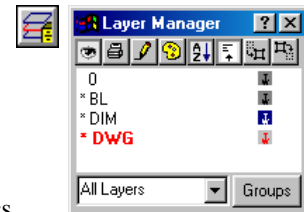
Displays (or hides) the Layer Manager window. From the Layer Manager, you can choose the layer(s) you want to view or hide in order to facilitate your importing information from the CAD file into your Source program. Toolbar Options:

Hide/Show selected layer. Names of hidden layers are shown in italics

Designate selected layer as Current layer (shown in red)

Sort layers in alphabetic order

Display Short/Long list of layers



Note: All other buttons are not active in G-ZERO.

To close the Layer Manager window, click the button in the upper right corner of the Layer Manager window, or click the option again from the CAD Import Interface window.

Toggle display of Colors

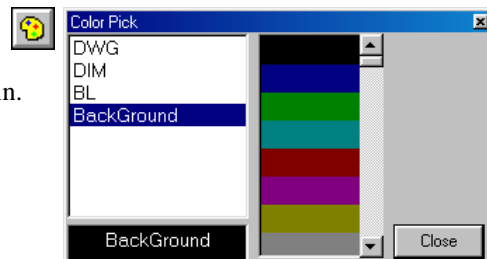
Select a layer and pick the color you want it to be displayed in.

If you want to change the color of the background, select

“BackGround” before picking its new color.

All changes take effect immediately.

Click the Close button to close the Color Pick window.



Open a different CAD file

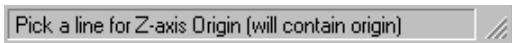
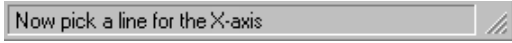
Displays the *CAD file to Open* window from where you can select a different CAD file to open. This will add one comment line (a link to the CAD file) to your source code.

Close CAD window

Closes the *CAD Import Interface* window. You can also close this window by pressing the F9 function key.

Setting Origin of Coordinates

CAD drawings usually have the origins of coordinates different from the one used by machinists. Setting the origin is the first thing that has to be done to make sure that all information taken from the CAD drawing are referred to the same point of origin. *Note:* since G-ZERO does not alter the content of the CAD file it opens, the new origin is not saved; therefore, a new origin needs to be set every time the CAD drawing is loaded.

1. Select the Origin icon from the Toolbar menu (or press the F2 function key when the CAD Reader window is active)
2. 
Select a horizontal line that contains the origin.
Note: If you select a circle or arc, G-ZERO will take the center of the circle (or radius) as the new origin (and will skip step 3)
3. 
Select a vertical line that contains the origin.
4. The intersection of the two lines will be the new origin. Your CAD drawing will be redrawn with new coordinates shown in yellow.

If you would like to change the origin, just repeat steps 1 to 4.

Getting Information from an Element

Answering G-ZERO questions

When CAD Reader is active, you can answer some G-ZERO questions by clicking on the elements from your CAD drawing. Once you picked an element, CAD Reader will read all its information (location relative to the origin, radius of a circle or an arc, etc.) and pick the appropriate data for your answer.

Note: You can always use the picked value as part of your input math calculation. *Note* that you need to pick the CAD value before adding or subtracting other values you key in. (See *Reference Basics, Input Math*)

Programming a POINT

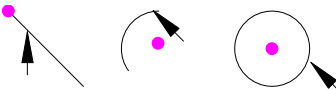
When you are NOT in the middle of programming a G-ZERO command (CAD Reader in Single Pick mode), clicking on ANY element of your CAD drawing will automatically program a 1)POINT command and add it to the end of your source program.

The X and Z values of the new POINT command will be:

Picking a LINE: X (diameter) and Z values of the closest end point of that line

Picking an ARC: X (diameter) and Z values of the center of the arc

Picking a CIRCLE: X (diameter) and Z values of the center of the circle



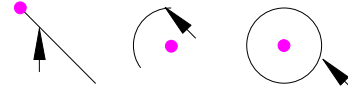
Using CAD Reader with G-ZERO Commands

1) POINT x_y_f_

x_

WHERE IN X WOULD YOU LIKE TO POSITION THE TOOL?

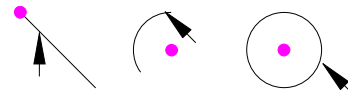
<u>Click on...</u>	<u>You'll get...</u>
line	X value of closest end (diameter)
arc	X value of center (diameter)
circle	X value of center (diameter)



z_

WHERE, IN Z?

<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end
arc	Z value of center
circle	Z value of center



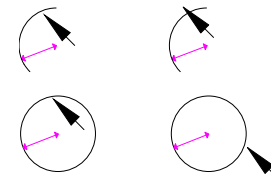
2) RADIUS _ type _ x_ y_

(Note: only available within COMP/UNCOMP commands)

-

WHAT IS THE RADIUS OF THE CIRCLE? (- for inside rad/chamfer)

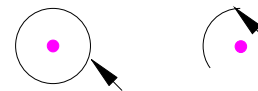
<u>Click on...</u>	<u>You'll get...</u>
arc (inside)	radius of arc (inside cut: -)
arc (outside)	radius of arc (outside cut: +)
circle (inside)	radius of circle (inside cut: -)
circle (outside)	radius of circle (outside cut: +)



x_

WHAT IS THE X DIMENSION OF THE RADIUS?

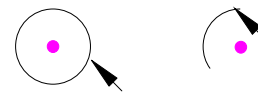
<u>Click on...</u>	<u>You'll get...</u>
arc	X value of the center of the arc (diameter)
circle	X value of the center of the circle (diameter)



z_

WHAT IS THE Z DIMENSION?

<u>Click on...</u>	<u>You'll get...</u>
arc	Z value of the center of the arc
circle	Z value of the center of the circle

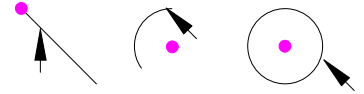


8) FACE x_ z_ xcut_ zcut_ depth_ f_ ***

x_

RAPID TO WHAT SAFE X DIMENSION?

<u>Click on...</u>	<u>You'll get...</u>
line	X value of closest end (diameter)
arc	X value of center (diameter)
circle	X value of center (diameter)



z_

START FACING AT WHAT Z DIMENSION?

<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end
arc	Z value of center
circle	Z value of center

xcut_

FEED TO WHAT X DIMENSION? (usually minus to go past cl)

<u>Click on...</u>	<u>You'll get...</u>
line	X value of closest end (diameter)
arc	X value of center (diameter)
circle	X value of center (diameter)

zcut_

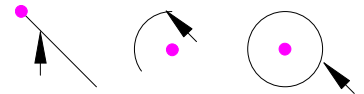
CUT TO WHAT FINAL Z? (usually 0)

<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end
arc	Z value of center
circle	Z value of center

9) ROUGH xstk_ zstk_ depth_ x_ z_ safang_ f

x_

<u>Click on...</u>	<u>You'll get...</u>
line	X value of closest end (diameter)
arc	X value of center (diameter)
circle	X value of center (diameter)



z_

CUT TO WHAT FINAL Z? (usually 0)

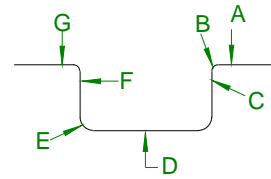
<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end
arc	Z value of center
circle	Z value of center

10) GROOVE x_ z_ dp_ w_ skip_ r_ ir_ a_ f_

x_

WHAT IS THE CURRENT X DIMENSION OF THE GROOVE SURFACE?

1. Select Single Pick
2. Click line A (or line G) close to groove
3. CAD Reader picks up the X value (diameter) of nearest endpoint



z_

WHAT IS THE Z DIMENSION OF THE RIGHT-MOST EDGE?

1. Click line C
2. CAD Reader picks up the Z value of nearest endpoint

dp_

HOW DEEP IS THE GROOVE?

1. Click line D
2. CAD Reader calculates the depth of the groove

w_

HOW WIDE IS IT?

1. Click line F
2. CAD Reader picks up the Z value of nearest endpoint (will be used by G-ZERO to calculate the width of the groove)

z_

WHAT IS THE Z DIMENSION OF THE RIGHT-MOST EDGE?

1. Click line C
2. CAD Reader picks up Z value of nearest endpoint

r_

BREAK OUTSIDE CORNERS WITH WHAT RADIUS?

1. Click arc B if there is a radius
2. CAD Reader picks up the radius of the arc

ir_

WHAT SIZE RADIUS IN BOTTOM OF GROOVE?

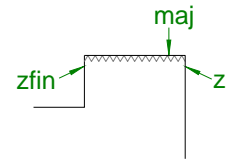
1. Click arc E (click outside of radius for a positive radius; inside for negative radius)
2. CAD Reader picks up the radius of the arc

11) THREAD maj_ tpi_ min_ s_ z_ zfin_ num_

maj_

WHAT IS THE MAJOR DIAMETER OF THE THREAD?

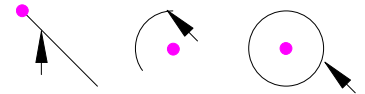
<u>Click on...</u>	<u>You'll get...</u>
line	X value of closest end (diameter)
arc	X value of center (diameter)
circle	X value of center (diameter)



z_

WHERE, IN Z WOULD YOU LIKE TO START?

<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end
arc	Z value of center
circle	Z value of center



zfin_

WHERE, IN Z DOES THE THREAD END?

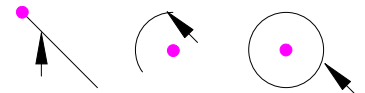
<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end
arc	Z value of center
circle	Z value of center

12) DRILL g_ rap_ cut_ peck_ tip_ ret_ f_

cut_

DRILL TO WHAT FINAL DIMENSION?

<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end
arc	Z value of center
circle	Z value of center



17) MAT'L od_ id_ zlim_ zface_ zclamp_ type_

od_

WHAT IS THE MAXIMUM DIAMETER OF THIS PART?

<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end (diameter)
arc	Z value of center (diameter)

id_

WHAT IS THE MINIMUM DIAMETER OF THIS PART? (often 0)

<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end (diameter)
arc	Z value of center (diameter)

zlim_

WHAT IS THE FURTHEST Z OF THE PART? (usually minus)

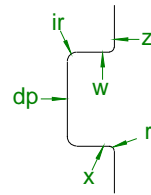
<u>Click on...</u>	<u>You'll get...</u>
line	Z value of closest end
arc	Z value of center

39) FGROOVE x_z_dp_w_skip_r_ir_a_f_

x_

WHAT IS THE X DIMENSION OF THE GROOVE ID?

1. Select Single Pick
2. Click line x (see illustration) close to groove surface (face)
3. CAD Reader picks up the Z value (diameter) of nearest endpoint



z_

WHAT IS THE Z DIMENSION OF THE GROOVE'S FACE? (usually 0)

1. Click line z
2. CAD Reader picks up the Z value of nearest endpoint

dp_

HOW DEEP IS THE GROOVE?

1. Click line dp
2. CAD Reader calculates the depth of the groove

w_

HOW WIDE IS IT?

1. Click line w
2. CAD Reader picks up the X value of nearest endpoint and calculates the width of the groove

r_

BREAK OUTSIDE CORNERS WITH WHAT RADIUS?

1. Click arc r if there is a radius
2. CAD Reader picks up the radius of the arc

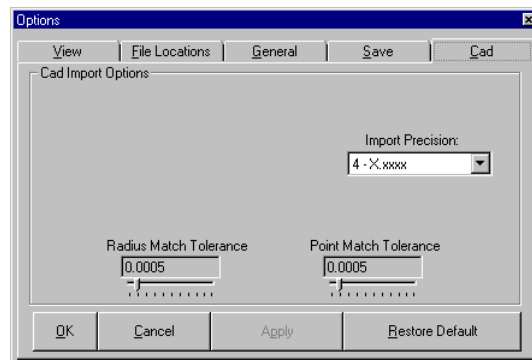
ir_

WHAT SIZE RADIUSES IN BOTTOM OF GROOVE?

1. Click arc ir (click outside of radius for a positive radius; inside for negative radius)
2. CAD Reader picks up the radius of the arc


Default Settings

To modify the default settings of the CAD Import Interface, make the appropriate selections in the CAD section of the *Options* window (*Tools* pull-down menu from the G-ZERO main screen).



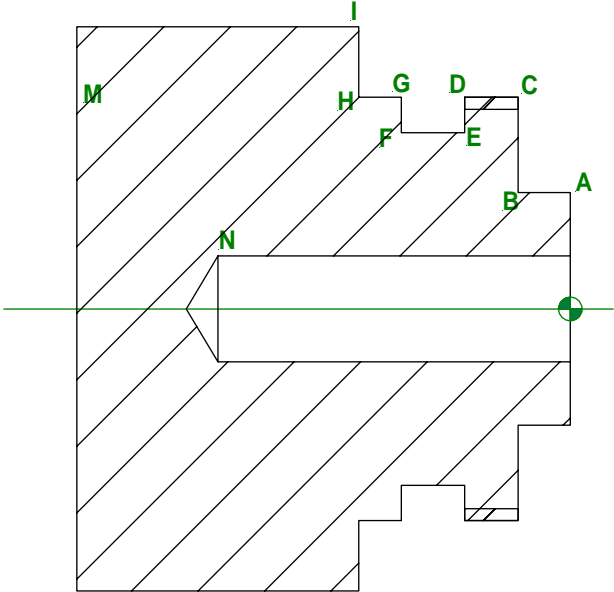
Sample Programming


This section covers a step-by-step example of programming source code files using G-ZERO Conversational Lathe through CAD Reader. DXF files used in this example is located in the Lathe\Tutorial directory.

Note: Use Zoom Window  or other Zoom options to setup the best view for you to make selections.

Sample 1: Hub

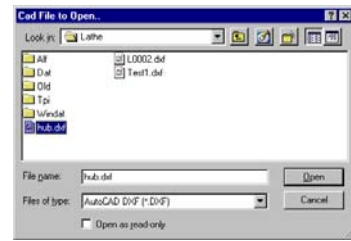
- ◆ Load CAD Reader and open DXF file
- ◆ Hide layers
- ◆ Set origin for DXF file
- ◆ Define material using values from DXF file
- ◆ Face and turn OD
- ◆ Drill
- ◆ OD Groove
- ◆ Thread OD
- ◆ Exit CAD Reader
- ◆ Proofread and edit source program








	TITLE: Hub		NOTES:	RECOMMENDED OP & TOOL LIST:	
	MATERIAL: 316 Stainless Steel				1. All measurements in inches (not to scale)
	DRAWING: L0002	INITIALS: MC			2. Deburr all sharp corners
			3. Material amount in chuck = 0.5"	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)	

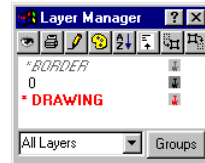
Load CAD Reader and open DXF file

1. Start G-ZERO Conversational Lathe
2. Open a new file named Hub-DXF (File | Open)
3. Press F9 (Tools | CAD) to load CAD Reader
4. Locate and open file hub.dxf
Note: Select drive and directory where the file is located. Select files of type = AutoCAD.DXF to display only DXF files.
5. The hub.dxf file is opened in the G-ZERO CAD Import Interface window.
6. The first line of source code is added to your program to establish a link to the DXF file:
1 CAD C:\LatheTutorial\hub.dxf




Hide layers

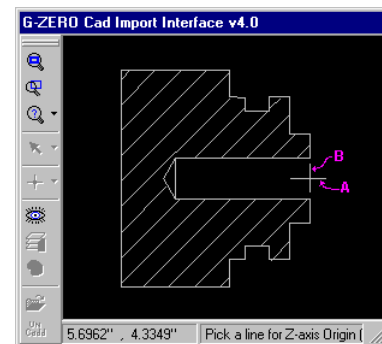
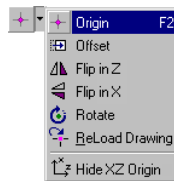
7. Click the “Toggle Display of Layers”  button to open the “Layer Manager”
 8. Select the DRAWING layer and set it as current layer 
 9. Select the DIMENSION layer and hide it (non-visible) 
 10. Click the “Toggle Display of Layers” button again to close the “Layer Manager”
- Note: use  Zoom All or  Zoom Window whenever you need to get a better display of the section you are working on.





Set origin for DXF file

It's very important to make sure that the coordinates of the DXF file match the coordinates of the source program. In this example, the origin is already shown with two crossed lines.

11. Select Origin from the *Origin Options* button  (or press F2 with G-ZERO Cad Import Interface window active)
12. Pick a line for Z-axis Origin (will contain origin)
Click line A (horizontal line)
13. Now pick a line for the X-axis
Click line B (vertical line)

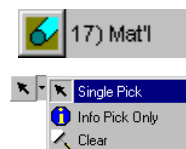


The XZ Origin of coordinates will appear in that corner; you can hide it by selecting  Hide XZ Origin from  Origin in the toolbar.

Define material using values from DXF file

We can use values from the DXF file to define the material size and thickness. Call the MAT'L command the usual way, and instead of keying in the min and max values, select them from the DXF file.

14. Start the *I7)MAT'L* command
15. Select *Single Pick* from the CAD Reader's toolbar
 od click top side of line M. “4” will be picked and entered as od
 id type 0 since the material is not pre-drilled
 zlim click line M. “-3.5” will be picked and entered as zlim
 zface type .07
 zclamp type .5
 type type 2
2 MAT'L od4 id0 zlim-3.5 zface.07 zclamp.5 type2=316



Face and Turn OD

16. Program the 6)TOOL and 36)TYPE commands the normal way.

```
3 TOOL 1 in/out2 *** FACE AND TURN OD
4 TYPE 1 mat0=ALUM/OXY trn.032 size.375
```

17. For the 8)FACE command, we can get the x and z_{cut} values from the DXF file. (Program the rest of the FACE command as you normally would)

Rapid to what safe x dimension?

- Select Single Pick
- Click line M, and CAD Reader will pick up the value 4
- Type +.1 (you will see 4+.1)
- Press <Enter>, and G-ZERO will enter the result of that calculation

Cut to what final z ? (usually 0)

- Select Single Pick
- click line A

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

18. For the 9)ROUGH command, we can get the x value from the DXF file. (Program the rest of the ROUGH command as you normally would)

Where in X would you like to start roughing from?

- Select Single Pick
- Click line M, and CAD Reader will pick up the value 4
- Type +.2 (you will see 4+.2)
- Press <Enter>, and G-ZERO will enter the result of that calculation

```
6 ROUGH xstk.01 zstk.002 depth.1 x4.2 z.05 safang270 f2
```

19. Program the 4)COMP command as you normally would.

```
7 COMP ang90 lft/rt2 f5
```

20. To define the contour, select points A, B, C, H, and I from the DXF file to get the x ; z values needed to program the POINT commands.

- Select Single Pick
- Click point A, and CAD Reader will enter the POINT command at line 8 in your source file

```
8 POINT x1.64 z0
```

- Click point B

```
9 POINT x1.64 z-.37
```

- Click point C

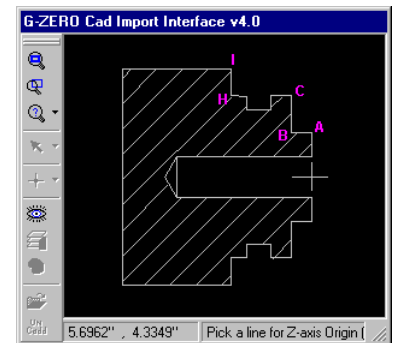
```
10 POINT x3 z-.37
```

- Click point H

```
11 POINT x3 z-1.5
```

- Click point I

```
12 POINT x4 z-1.5
```



21. Program the 5)UNCOMP and 9)ROUGH (unRough) commands as you normally would.

```
13 UNCOMP ang90
14 unROUGH
```

22. Program a 15)REPEAT command from Comp to Uncomp to remove remaining stock in a finish pass.

```
15 REPEAT from7 thru13
```

Drill

23. Program the 6)TOOL and 36)TYPE commands the normal way.

```
16 TOOL 2 in/out1 *** DRILL
17 TYPE 29 mat0=M2 tnr0 size.75
```

24. For the 12)DRILL command, we can get the *cut* value from the DXF file. (Program the rest of the FACE command as you normally would)

Drill to what final dimension?

- Select Single Pick
- Click point N, and CAD Reader will pick up the value -2.5

```
18 DRILL g74 rap.05 cut-2.5 peck-4 tip1 ret.1 f.0085
```

OD Groove

25. Program the 6)TOOL and 36)TYPE commands the normal way.

```
19 TOOL 3 in/out2 *** OD GROOVE
20 TYPE 9 mat0=TIN tnr0 size.125
```

26. For the 10)GROOVE command, we can get the *x*, *z*, *dp*, and *w* values from the DXF file. (Program the rest of the GROOVE command as you normally would)

What is the current X dimension of the groove surface?

- Select Single Pick
- Click point D or G, and CAD Reader will pick up value 3 and prompt for the next parameter

What is the Z dimension of the right-most edge?

- Select Single Pick
- click point D, and CAD Reader picks up the value -.75

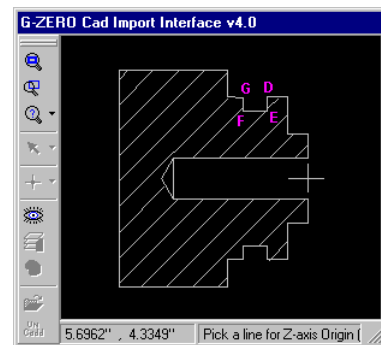
How deep is the groove?

- Select Single Pick
- click point E, and CAD Reader picks up the value -2.5

How wide is it?

- Select Single Pick
- click point G or F, and CAD Reader picks up the value .45

```
21 GROOVE x3 z-.75 dp-2.5 w.45 skip0 r.005 ir0 a0 f2
```



Thread OD

27. Program the 6)TOOL and 36)TYPE commands the normal way.

```
22 TOOL 4 in/out2 *** THREAD OD
23 TYPE 15 mat1=C-5 tnr0 size20
```

28. For the 11)THREAD command, we can get the *maj*, *z*, and *zfin* values from the DXF file. (Program the rest of the THREAD command as you normally would)

What is the major diameter of the thread?

- Select Single Pick
- Click point C, and CAD Reader will pick up value 3 and prompt for the next parameter

What is the minor diameter of the thread?

- G-ZERO makes the calculations and suggests 2.938655. Press *Enter* to accept this number

Where, in Z would you like to start?

- Select Single Pick
- Click point C, and CAD Reader will pick up value -.37


- A pop-up box suggests a minimum start distance of .04 in., so type +.04
- G-ZERO makes the calculation and enter the value -.33 (press “o” for overwrite if needed)

Where, in Z does the thread end?

- Select Single Pick
- Click point D, and CAD Reader will pick up value -.75
- A pop-up box suggests a minimum stop distance of .012 in., so type -.012
- G-ZERO makes the calculation and enter the value -.762

24 THREAD maj3 tpi20 min2.938655 X318 z-.33 zfin-.762 num8 a60

Exit CAD Reader

Exit CAD Reader by pressing F9 again, *or* selecting  in the toolbar, *or* selecting UnCad from the Tools drop-down menu.

Proofread and edit source program

Proofread source program using the different redraw function keys: F2 Part, F3 Tool, or F6 Solid.

Make any changes if necessary using G-ZERO editing features.

Return to CAD Reader to get data if needed by pressing F9 again. Remember to set a new origin in your DXF file any time you reload it.

Save source file and exit G-ZERO when done.

