G-ZERO Lathe

Live Tooling

G·ZERO
CAM Software

RAPID OUTPUT
(888) 656-1945
www.g-zero.com

Contents

Lathe Live Tooling

Lathe Live Tooling is an add-on to the G-ZERO Lathe software that supports drilling and milling on the end of a workpiece as well as on a specific section of its OD (outside diameter) contour. Coordinates can be input in polar or rectangular form.

Cross-milling and cross drilling are displayed "unwrapped" on screen so you can see the inch and angular location of your cut. You can work with the ZC axes (polar) or the ZY' axes (rectangular). End-milling and end-drilling can also be displayed on the screen while you are working in the XC axes (polar) or XY' axes (rectangular). If your CNC machine supports Y axis movement, you can program the XY and ZY axes.

Live Tooling Commands

Function Keys

F1 CLEAR The <F1> function key redraws the first three lines of the source program. Since a MAT'L command is generally there, the graphics screen will be cleared.

F4 XC The <F4> function key redraws the entire source program and displays the end view of the workpiece. This view is automatically displayed when setting the axes XC, XY', and XY.

F5 ZC The <F5> function key redraws the entire source program and displays the unwrapped side view of the work-piece. This view is automatically displayed when setting the axes ZC, ZY', and ZY.

6) TOOL _ in/out_ xsafe_ zsafe_

WHICH TOOL STATION WOULD YOU LIKE?

IS THIS TOOL SET FOR: 1) ID OR 2) OD CUTTING?

- Select ID tool cutting when working with axes XC, XY', and XY. Z is the plunge axis.
- Select OD tool cutting when working with axes ZC, ZY', and ZY. X is the plunge axis.

WHAT SAFE X SHOULD THE TURRET RAPID TO BEFORE INDEXING? (g50)

- If you prefer, your G-ZERO system may not ask this question.

WHAT SAFE Z WOULD YOU LIKE TO INDEX AT?

- If you prefer, your G-ZERO system may not ask this question.

36) TYPE __ mat&__ tnr__ size&__ axes&__ workdia__

WHAT TYPE OF TOOL IS THIS?

WHAT IS THE TOOL (insert) MADE OF?

WHAT IS THE TOOL'S NOSE RADIUS: (0 for drills and mills)?

HOW BIG IS THIS TOOL: (ic, dia, width, tpi)?

WHICH AXES WILL YOU BE PROGRAMMING?

0=XZ	normal cross sectional view	normal programming	
1=XC	end view	polar coordinates	
2=ZC	unwrapped side view	polar coordinates	
3=XY'	end view	cartesian coordinates	G-ZERO Milling Continuous
4=ZY'	unwrapped side view	cartesian coordinates	G-ZERO Milling Continuous
5=XY	end view	cartesian coordinates	only for Y Axis Lathe
6=ZY	unwrapped side view	cartesian coordinates	only for Y Axis Lathe

Drilling or Cross Milling

ZC: polar input

ZY': rectangular input (cartesian coordinates)

ZY: rectangular input (cartesian coordinates)

End Drilling or End Milling

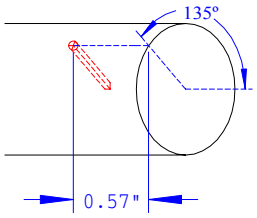
XC: polar input

XY': rectangular input (cartesian coordinates)

XY: rectangular input (cartesian coordinates)

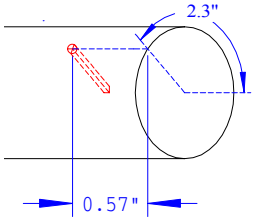
ZC

z=-57 c=135 (degrees)



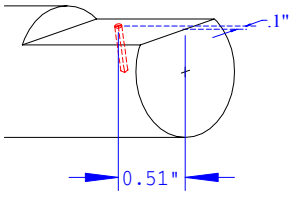
ZY'

z=-.57 y'=2.3 (inches)



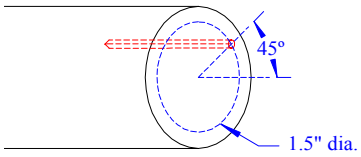
ZY

z=-.51 y=.1



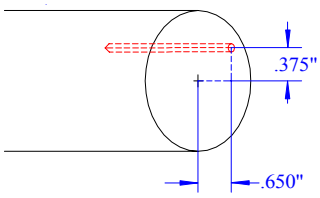
XC

x=1.5 c=45 (degrees)



XY' (or XY)

x=.65 y'=.375



WHAT IS THE DIAMETER OF THE SURFACE YOU WILL BE CUTTING?

The diameter of the surface you will be cutting is used by G-ZERO when working in the unwrapped side view. G-ZERO only asks this question when programming in ZC or ZY'.

12) DRILL g__ rap__ cut__ peck__ tip__ ret__ f__

WHAT G-CODE WOULD YOU LIKE TO DRILL WITH?

You will normally drill with the 74 cycle.

- When cross-drilling or end-drilling, you will use 81 or 83 instead.

START FEEDING FROM (rapid to) WHAT DIMENSION?

You will generally want to start from just in front of your part. A typical value would be .02.

- When cross-drilling, give a value larger than your OD.

DRILL TO WHAT FINAL Z DIMENSION?

Your answer will usually be negative.

- When cross-drilling, the final Z dimension is the value of the diameter minus twice the depth of the hole to drill. Example: if we want to drill a hole 0.25" deep on a 2" diameter bar, the final Z dimension will be: 2 - .25 - .25 (or 1.5")
- When cross-drilling, give a positive value near your ID.

PECK HOW MANY TIMES? (- for full retract)

WOULD YOU LIKE TO DRILL TIP ADDED? (1=yes, 0=no)

RETRACT TO WHAT DIMENSON? (usually = zrap)

37) MILL rap__ cut__ passes__ ret__ plungef__

RAPID TO WHAT DIMENSION BEFORE KICKING TO FEED MODE?

The MILL command will automatically wait until a location is given before moving the tool at rapid to that location. You will generally want to start from just in front of your part. A typical value would be .05.

- When cross-milling, give a value larger than your OD.

FEED IN TO WHAT FINAL DIMENSION?

Once the tool has rapided to the proper location, it will feed straight into the part to this dimension. Your answer will often be negative.

- When cross-milling, generally give a positive (+) value near your ID.

RETRACT TO WHAT DIMENSION? (usually=rap)

At the end of this cut, you will probably wish to have the tool pull out of the hole you just milled so that it can rapid over to the next cut. Enter the dimension to retract to at the end of the cut.

PLUNGE INTO THE PART AT WHAT FEEDRATE? (ipr)

Enter the plunge feedrate in inches per revolution. The answer you see in default brackets is based on both the part material and your tool. This information is from the .F file.

WHAT FEEDRATE (ipr) WOULD YOU LIKE?

Give the feedrate in inches per revolution for this operation.

- When contouring (COMP, FACE, BORE or POINT) you may enter the required finish instead of the feedrate. Just enter a number from 3 (finish) to 9 (rough) now.
- When ROUGHing, GROOVing or MILLing, you may enter the required load instead of the feedrate. Just enter a number from 1 (light load) to 4 (heavy load) now.

14) STOCK x__ z__

LEAVE HOW MUCH MATERIAL ON TURNED SURFACES FOR CLEANUP?

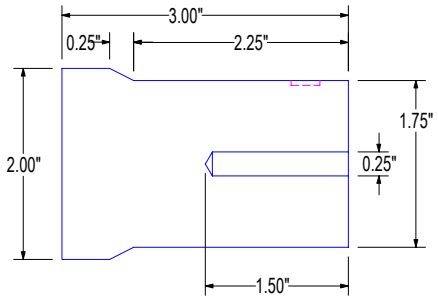
For milling with live tools, enter the amount of stock to leave on part walls.

HOW MUCH STOCK WOULD YOU LIKE TO LEAVE IN Z?

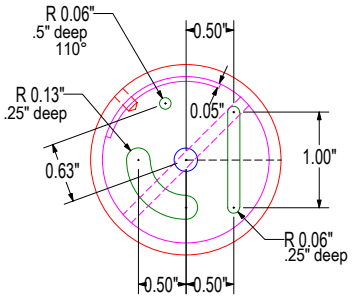
For milling with live tools, enter the amount of stock to leave on pocket floors.

Sample 1: LT001

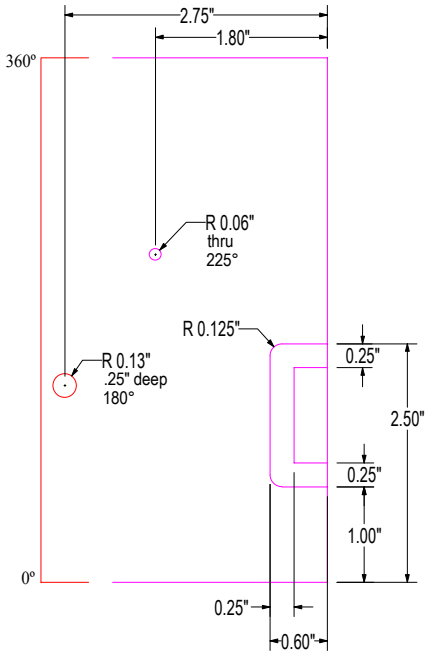
This example shows you the way to set up the correct axes and the input method to use according to the information provided on a blueprint.



NORMAL CROSS SECTIONAL VIEW



END VIEW



UNWRAPPED SIDE VIEW

NOTE:
All measurements in inches
(Not to scale)

RAPID OUTPUT CO.

G·ZERO
CAM Software

TITLE: LIVETOOL - LT001

MATERIAL: Stainless Steel 303

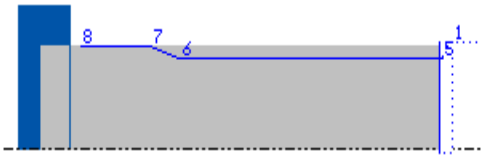
Working on Normal Cross Sectional View (axes 0=XZ)

1 MAT'L od2 id0 zlim-3.2 zface.02 zclamp.25 type1=303
Use the blueprint to get information for the MAT'L command.

2 TOOL 1 in/out2 ***FACE AND TURN OD
3 TYPE 1 mat0=ALUM/OXY tnr.032 size.5 axes0=XZ

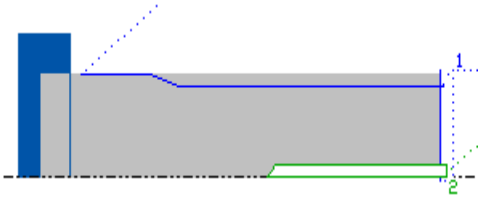
In the TYPE command, when G-ZERO asks "Which axes will you be programming?" type 0 to select the XZ axes since we are going to program in the traditional XZ coordinates.

4 FACE x2.05 z.1 xcut-.064 zcut0 depth0 f.005
5 POINT x1.75 z.02 f0
6 POINT x1.75 z-2.25 f.005
7 POINT x2 z-2.5 f.005
8 POINT x2 z-3.1 f.005
Since we are programming in the XZ coordinates, all points are entered in inches in the traditional way. X values are diameter values, and Z values are usually negative values since they are usually on the left side of the Z0 plane (face of the part).
Press F2 to redraw the source program displaying the part in the xz view.



9 TOOL 2 in/out1 ***DRILL ON CENTER
10 TYPE 29 mat0=M2 tnr0 size.25 axes0=XZ
11 DRILL g74 rap.05 cut-1.5 peck2 tip0 ret.05 f.003

Again, select axes 0=XZ and enter all dimensions in inches to program in the normal traditional way.

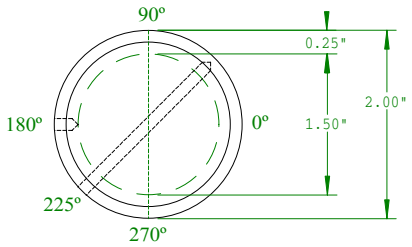


Working on Unwrapped Side View (axes 2=ZC with polar input)

12 TOOL 3 in/out2 ***CROSS DRILL
13 TYPE 31 mat0=M2 tnr0 size.125 axes2=ZC workdia2

Since our blueprint shows the two side holes in the unwrapped side view, choose axes 2=ZC. Enter all points in polar mode: z values in inches, and c values in degrees.

When working in the ZC axes, G-ZERO needs to know the diameter of the surface you will be cutting, so enter a 2 for *workdia*.



Depth of cut

14 DRILL g83 rap2.1 cut1.5 peck2 tip0 ret2.1 f.003
rap2.1: When cross drilling, use a rap value larger than the OD.

cut1.5: When cross drilling, use a positive (+) cut value near the ID. The hole is .25" deep, therefore *cut* is 1.5.

15 POINT z-2.75 c180

The first hole (left hole in unwrapped side view of blueprint) is located at -2.75” in the z axis and 180º in the 4th axis (c value).

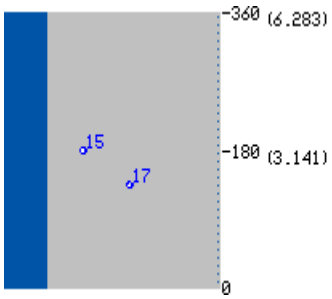
16 DRILL g83 rap1.85 cut-1.75 peck3 tip1 ret1.85 f.003

rap1.85: When cross drilling, use a rap value larger than the OD. Here the OD is 1.75, so we are using rap1.85.

cut-1.75: When cross drilling and cutting through the material, the cut value is negative. In this case, we are drilling *through* on diameter 1.75.

17 POINT z-1.8 c225

Similarly, enter the point for the second hole, programming a z value of -1.8 and c value of 225 (degrees).

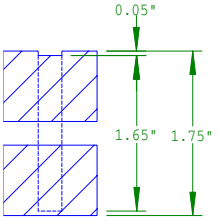


Working on Unwrapped Side View (axes 4=ZY’ with rectangular input)

18 TOOL 4 in/out2 ***CROSS MILL
19 TYPE 27 mat0=M2-2F tnr0 size.25 axes4=ZY’ workdia1.75
20 MILL rap1.95 cut1.65 passes1 ret1.95 plungef.0005 f.003

rap1.95: When cross milling, use a rap value larger than the OD. Here the OD is 1.75, so we are using rap1.95.

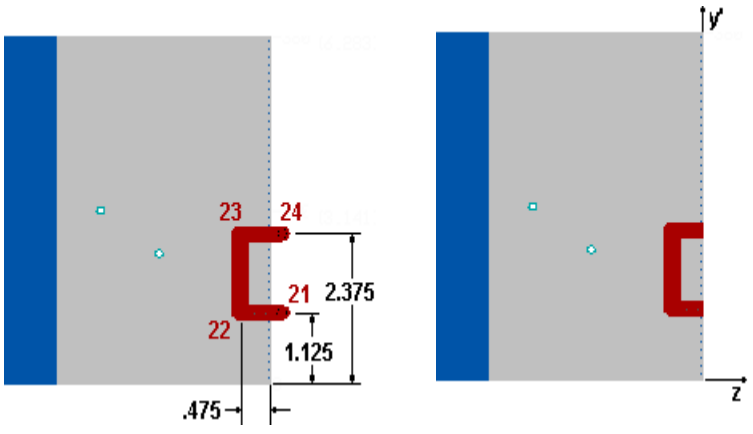
cut1.65: When cross milling, use a positive (+) cut value near the ID. In this case, our cut is .05” deep and OD is 1.75”, therefore we can use a *cut* value of 1.65.



Cross Sectional View showing milling cut

21 POINT z.2 y’1.125
22 POINT z-.475 y’1.125 f.003
23 POINT z-.475 y’2.375 f.003
24 POINT z.2 y’2.375 f.003

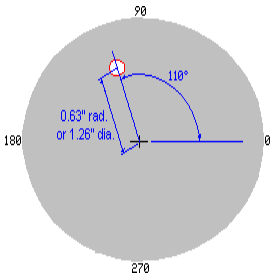
Enter these points using rectangular input with coordinates (z;y’)



Working on End View (axes 1=XC with polar input)

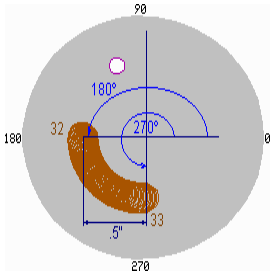
```
25    TOOL  5  in/out1 ***END DRILL
26    TYPE     31 mat0=M2 tnr0 size.125 axes1=XC
27  DRILL      g83 rap.05 cut-.5 peck2 tip0 ret.05 f.003
28  POINT      x1.26 c110
```

To work on the End View, select axes 1=XC. Enter all geometry using polar input: x values represent diameter, and c values represent degrees.



```
29    TOOL  6  in/out1 ***END MILL ON XC
30    TYPE     27 mat0=M2-2F tnr0 size.25 axes 1=XC
31  MILL       rap.1 cut-.25 passes1 ret.1 plungef.0006 f.003
32  POINT      x1 c180
33  POINT      x1 c270 f.003
```

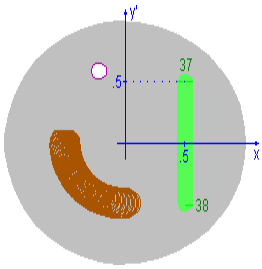
To mill the arc shown below through the spindle control motion, define two points:
- Point in line 32, use x=1 (2 times .5, or its diameter value), and c=180 (degrees)
- Point in line 33, use x=1 (same as in line 32), but c=270 (degrees)



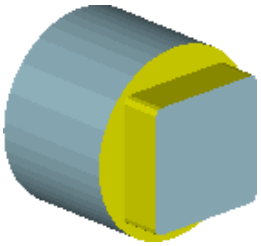
Working on End View (axes 3=XY' with rectangular input)

```
34    TOOL  7  in/out1 ***END MILL ON XY'
35    TYPE     27 mat0=M2-2F tnr0 size.125 axes3=XY'
36  MILL       rap.1 cut-.25 passes1 ret.1 plungef.0004 f.003
37  POINT      x.5 y'.5
38  POINT      x.5 y'-.5 f.003
```

When the blueprint shows dimensions in cartesian coordinates for a work to be done in the end view, set the axes to 3=XY' and enter the x and y (or y') values of the geometry.



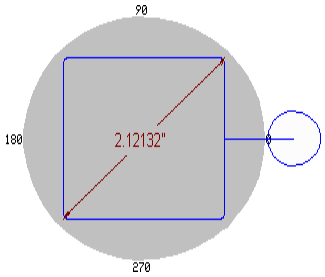
Sample 2: Square.l



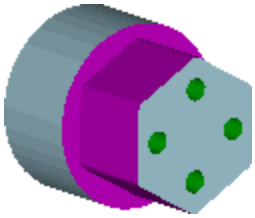
Cut an square into the top half inch of a part that is 2.25” in diameter, using a tool half an inch in diameter.

Using axes XC makes it easier to program the points (or radii) of the corners of the square: Lines 8-11 use the same x value (2.12132) and the c values represent the angle in which they are located.

```
1   MAT’L       od2.25 id0 zlim-1 zface0 zclamp.1 type4=ALUMINUM
2     TOOL  1   in/out1 ***
3     TYPE      27 mat0=M2-2F tnr0 size.5 axes1=XC
4   MILL        rap.1 cut-.5 passes1 ret.1 plungef.003 f.004
5   POINT       x2.8 c0
6   COMP        ang270 lft/rt1
7   POINT       x1.5 c0
8   RADIUS      .1 type2 x2.12132 c-45 (xc1.838477 cc-45)
9   RADIUS      .1 type2 x2.12132 c-135 (xc1.838477 cc225)
10  RADIUS      .1 type2 x2.12132 c-225 (xc1.838477 cc135)
11  RADIUS      .1 type2 x2.12132 c-315 (xc1.838477 cc45)
12  POINT       x1.5 c0
13  UNCOMP      angle270
14  POINT       x2.8 c0 f.004
```



Sample 3: Hexagon.l



Cut a hexagon into the top one inch of a part that is 2” in diameter, using a tool half an inch in diameter.

```
1   MAT’L      od2 id0 zlim-1 zface0 zclamp.1 type4=ALUMINUM
2     TOOL  1  in/out1 ***
3     TYPE     27=em mat0=M2-2F tnr0 size.5 axes1=XC
4   MILL       rap.1 cut-1 passes1 ret.1 plungef.003 f.004
5   POINT      x2.7 c0
6   COMP       ang270 lft/rt1
7   POINT      x1.5 c0
8   RADIUS     .005 type2 x1.732 c-30 (xc1.720453 cc-30)
9   RADIUS     .005 type2 x1.732 c-90 (xc1.720453 cc-90)
10  RADIUS     .005 type2 x1.732 c-150 (xc1.720453 cc210)
11  RADIUS     .005 type2 x1.732 c-210 (xc1.720453 cc150)
12  RADIUS     .005 type2 x1.732 c-270 (xc1.720453 cc90)
13  RADIUS     .005 type2 x1.732 c-330 (xc1.720453 cc30)
14  RADIUS     .005 type2 x1.5 c-360 (xc1.49 cc-.00001990945)
15  UNCOMP     angle270
```

Drill 4 holes in the End View

```
16    TOOL  2  in/out1 ***
17    TYPE     29=drill mat0=M2 tnr0 size.25 axes1=XC
18  DRILL      g74 rap.1 cut-.5 peck-.25 tip1 ret.1 f.004
19  POINT      x1 c0
20  POINT      x1 c90
21  POINT      x1 c180
22  POINT      x1 c270
```