

G-ZERO Lathe








































Reference - Commands



Contents

Commands	5
Fundamentals	6
1) POINT	7
2) RADIUS	8
3) LINE	9
4) COMP	10
5) UNCOMP	10
6) TOOL	11
7) BAR OP	11
8) FACE.....	12
9) ROUGH	13
10) GROOVE	17
11) THREAD	19
12) DRILL	20
13) BORE	21
14) STOCK	21
15) REPEAT	22
16) ROTATE	22
17) MAT'L	23
19) REDRAW	24
20) POST	24
21) PRINT	25
22) ALTER	25
23) INSERT	26
24) DELETE	26
25) MOVE.....	27
26) UNEDIT	27
27) BASIC	28
28) BOLTCRC	30
29) DEADTAP	30
30) SHIFT	31
31) PARTOFF	31
32) SCALE	32
33) TAIL.....	32
34) PIPE	33
35) LETTER	34
36) TYPE	35
37) MILL	36
39) FGROOVE.....	37
41) PECK	39

Commands

	Programming	Non-Programming	Option
 1) Point	■		
 2) Radius	■		
 3) Line	■		
 4) Comp	■		
 5) Uncomp	■		
 6) Tool	■		
 7) Bar op	■		
 8) Face	■		
 9) Rough	■		
 10) Groove	■		
 11) Thread	■		
 12) Drill	■		
 13) Bore	■		
 14) Stock	■		
 15) Repeat	■		
 16) Rotate	■		
 17) Mat'l	■		
 19) Redraw		■	
 20) Post		■	
 21) Print		■	
 22) Alter		■	
 23) Insert		■	
 24) Delete		■	
 25) Move		■	
 26) UnEdit		■	
 27) Basic		■	
 28) Boltcrc			■
 29) Deadtap	■		
 30) Shift	■		
 31) Partoff	■		
 32) Scale	■		
 33) Tail	■		
 34) Pipe	■		
 35) Letter			■
 36) Type	■		
 37) Mill			■
 39) FGroove	■		
 40) Takeover			■
 41) Peck	■		

Fundamentals

Calling commands

G-ZERO Lathe has all commands listed on the left side of the application window. These commands can be called using your mouse or keyboard.

- ♦ Mouse: Place your mouse cursor over the command you want to use. Click to select that command and answer questions displayed on the Dialog bar.
- ♦ Keyboard: Key in the command number that correspond to the command you want to call. Example: if you want to call the command Material, key in: 17 (numeric keypad is usually preferred)

Validation / Override

Commands. G-ZERO never makes all commands available at the same time because it automatically disables any command that is very unlikely to be called. Example: All geometry commands (POINT, RADIUS, LINE, etc.) are disable until a TOOL is selected.

If you are sure the command you want to use is appropriate, you can override the computer using the “O” (letter oh, not zero) key. Use the numeric key pad to enter the number(s) that correspond to the disabled command (shown in gray), and press the “O” key before pressing the Enter key. Example: If you want to call the command RADIUS, but it is disabled (gray), use the keyboard and press: the number “2” (command RADIUS), letter “O” (to override), and the Enter key.

Values. In the same way, if you answer a programming question with a value beyond the acceptable limits, G-ZERO will not let you enter that value. Example: drilling with an endmill.

However, if you are sure the value you want to enter is correct, override the computer using the “O” key. Use the numeric key pad to enter your value and press the “O” key before pressing the Enter key. Example: If you want to program a POINT command to move the tool to (x1.5; z-4.1) with the Z value beyond the zlim of the Material command (zlim-3.5), answer the POINT’s question “Where in Z?” by keying in -4.1, “O” key and Enter key.

Undo / “Esc”

The Escape key backs you up to the last question G-ZERO was asking you.

If you have just finished entering a command such as a Point and press the Esc key, G-ZERO will ask for your permission to delete that last Point (last source line). Key in “Y” or press the Enter key to confirm deletion, or key in “N” to leave everything unchanged.

“Comment” line

Each line of a G-ZERO source program should be either a command or a comment line. Use comments for your own information or instructions to the operator such as description of the part, operation or tool.

- ♦ To enter a comment line, just begin typing without choosing a command (start a comment line with any key except for numbers and the minus sign “-”)
- ♦ Comments do not appear in the G-code unless they begin with a % sign (and if the CNC control can read comments without ALARMIing)
- ♦ Comments for operators should be enclosed in parentheses. Ex: %(CHANGE CLAMPS HERE)
- ♦ The first six comments will appear in the WinLauncher “Launch any file” window when WinLauncher is set to use the Incremental Search method in the Setup | Options window
- ♦ Some commands, such as TOOL, and REPEAT, accept comments after the asterisks ***
- ♦ Comments can extend to 100 characters. As you type, the characters will run off the right side of the screen, but they will show up on Source and G-code printouts

There are three types of programmable comments:

- ♦ **% (FLIP PART)** Comments that begin with a % sign and are enclosed in parentheses go to the machine operator embedded in the G-code
- ♦ **%F25T0101** Comments that begin with a % sign become part of the G-code and are machine-readable
- ♦ **FACE THEN TURN OD** Comments without a % sign and parentheses are for the programmer



1) POINT

Format: POINT x_ z_ f_

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

In order to move the tool to a spot on or near your part, you must enter 2 dimensions: an X dimension (usually expressed in inch diameters) and a Z dimension (usually in inches from the face of the part). Enter the X dimension now. You may type your response as a calculation using the +, -, *, /, or = keys.

Note 1: If you want your tool to crossover the centerline of your part, then change the sign of X.

Note 2: If you don't know this dimension, then you can let G-ZERO calculate it for you. To do this, you must lie. Hit the ESCape key and call this POINT a very small RADIUS whose center location is unknown.

WHERE, IN Z?

In order to move the tool to a spot on or near your part, you must enter 2 dimensions: an X dimension (usually expressed in inch diameters) and a Z dimension (usually in inches from the face of the part). Enter the Z dimension now. You may type your response as a calculation using the +, -, *, /, or = keys. If you make a mistake, then just press the BACKSPACE key. Remember, your answer will generally be negative.

Note 1: If you don't know this dimension, then you can let G-ZERO calculate it for you. To do this, you must lie. Hit the ESCape key twice and call this POINT a very small RADIUS whose center is unknown.

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

How fast, in inches per revolution (ipr), would you like the tool to feed toward this point? A typical answer would be .01.

Note 1: If you would like the tool to move at rapid traverse, then give zero as your answer.

Note 2: You may enter the required finish instead of the feedrate. Just enter a number from 3 (finish) to 9 (rough).

Note 3: If you would like to use inches per minute (ipm) instead of ipr, then give a minus number instead of a plus number. For example, -20 means 20 ipm.



2) RADIUS

Format: RADIUS _ type_ x_ y_

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

When defining a circle or fillet, you must give the radius. Remember that some blueprints show this dimension as a diameter, so you must occasionally divide by 2.

Note 1: If the tip of the tool will be traveling inside the circle instead of outside, then your answer should be negative. Most filets will have a minus radius.

Note 2: To have G-ZERO find an unknown POINT, you must pretend the point is a circle. Give a radius of 0 for an outside corner. Enter -.0001 for an inside corner.

Note 3: If this is a CHAMFERED CORNER, you may enter the chamfer amount instead of a radius.
Remember: - for inside corners!

WHICH TYPE OF RADIUS? (1=center 2=corner 0=neither)

If the center of the circle is dimensioned or is easy to calculate, then type a 1. If the circle is on an inside or outside corner that is dimensioned or easy to calculate, then type a 2. If you really don't know much about the circle at all or if you only know one of the dimensions, then type a 0.

Note 1: When G-ZERO does not let you type a 0, you must back up and give a point along the line first. Then continue along the line. NOW you may call up Radius and type a 0.

Note 2: If you type a 0 and G-ZERO finds the wrong radius, then you should enter a -1, this tells it to find the other answer.

Note 3: If this is a CHAMFERED CORNER, then G-ZERO will radius BOTH corners of the chamfer for you! Just enter the break radius. Remember: 0, 1 and 2 inches are not legal break radius sizes.

WHAT IS THE X DIMENSION OF THE RADIUS?

If you don't know, then back up and re-read the last question. Depending on your last answer, give the X value of the center or the corner of the circle. Remember that X is usually in diameters, so you may have to double some values when calculating.

WHAT IS THE Z DIMENSION?

If you don't know, then back up twice and re-read the question. Depending on your answer, give the Z value of the center or the corner of the circle.



3) LINE

Format: LINE angle_

MOVE ALONG THE LINE IN WHAT DIRECTION?

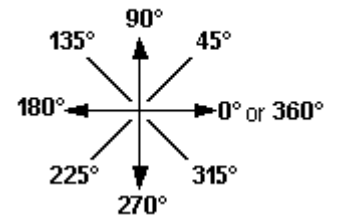
Give the angle that your tool will be moving as it travels along the line. Your answer should be in decimal degrees. If you don't know the line angle, have G-ZERO calculate it for you by pressing the (key.

Note 1: Always imagine the G-ZERO on-screen compass to be directly centered on the tool. The mouse may be used by clicking on the compass including the lines at 45° quadrants.

Note 2: You may also use the arrow keys (\leftarrow \uparrow \rightarrow \downarrow) for 0°, 90°, 180°, or 270° combined with your calculation. Example: 100° could be entered as \leftarrow + 10

Note 3: If you know the angle in degrees, minutes, and seconds, you may convert it by using the quote " key or the apostrophe ` key. For example, if you want to convert 22° 30' 15" into decimal degrees, type 22.3015" and G-ZERO will convert it to 22.50417 degrees.

Line angle compass





4) COMP

Format: COMP ang_ lft/rt_

WHAT ANGLE WILL YOUR TOOL FIRST BE TRAVELLING?

Give the angle that your tool will be moving as you first begin compensating for the radius of your cutter. Your answer will often be the same angle as that of the first line.

Note 1: If the first of this contour is for definition only, enter a dimension that is totally opposite (180 degrees) from the correct value. Your tool will start at the 2nd feature.

Note 1: If you have no idea whatsoever, then force G-ZERO to calculate it for you by entering 99.99.

WOULD YOU LIKE COMPENSATION FOR THE CUTTER: 1)LEFT or 2)RIGHT?

As you describe the path that your cutter will be making, you must stay to one side of the path or the other. For example, if you are cutting the outside diameter of your part and the tool is moving gradually toward the chuck, then your tool is on the right side of the path.

Note 1: If you would like G-ZERO to generate a smooth set of arcs through a bunch of points, then give your answer as a minus number (eg: -1 or -41).



5) UNCOMP

Format: UNCOMP angle_

WHAT ANGLE WAS YOUR TOOL LAST TRAVELLING?

To exit the tool compensation mode, you must tell G-ZERO exactly where you would like to quit. Give the angle that your tool will be moving at the very end of the contour. Your answer will often equal your last line angle.

Note 1: If the last point of this contour is for definition only, then enter a dimension that is totally opposite (180°) from the correct value. G-ZERO will halt the tool at the NEXT-TO-THE-LAST feature on this contour.

Note 2: If you have no idea whatsoever, then let G-ZERO calculate it for you by entering 99.99.



6) TOOL

Format: TOOL _ in/out_ xsafe_ zsafe_ ***

WHICH TOOL NUMBER WOULD YOU LIKE?

Your lathe's turret probably has 8 to 12 stations. Just enter the number of the turret position you want to use next.

Note 1: If you would like to reverse spindle direction, then enter this as a minus tool number.

Note 2: If you want to use an offset register other than the one usually used for a tool, then give a decimal of the register to use. (eg: to get T0104 instead of T0101, then enter 1.04)

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

If this tool will be used on the inside diameter (ID) of your part, or for drilling, then enter 1. If it will be used on the outside diameter (OD) or the face of your part, then type 2.

Note 1: If you are setting this tool to use the "away" side (a backroughing tool), then add 2 to your answer: ID/backset=3 OD/backset=4.

Note 2: To mill on the end with the Live Tool Option, choose ID.

Note 3: To use the front turret instead of the back, enter your answer as a minus number. (eg: -1 or -2)

WHAT SAFE X SHOULD THE TURRET RAPID TO BEFORE INDEXING?

Enter the X dimension that the turret must clear so that it can index without any part interference. 7 inches is common.

WHAT SAFE Z WOULD YOU LIKE TO INDEX AT?

Enter the Z dimension that the turret must clear so that it can index without any part interference. 7 is a common answer.

NOTE: After programming TOOL, G-ZERO will automatically call the command 36) TYPE.



7) BAR OP

Format: BAR OP xstart_ zstart_ xfin_ zfin_

RAPID TO WHAT X DIMENSION? (0 if not pulling)

For side-grip PULLers, enter a safe X. Most other PULLers will rapid to X0. If you are FEEDing the bar, then enter a safe X dimension to rapid the stop to.

RAPID TO WHAT Z DIMENSION?

Enter the Z dimension where this cycle begins.

FEED TO WHAT X DIMENSION?

Most PULLers will feed to X0. If you are FEEDing the bar, then enter the final X dimension to feed the stop to before opening the collet.

FEED THE BAR TO WHAT FINAL Z DIMENSION?

Give the Z dimension that you would like the part fed to before closing the collet. A typical value would be .02



8) FACE

Format: FACE x_z_xcut_zcut_depth_f_

RAPID TO WHAT SAFE X DIMENSION?

Give an X dimension that is safely away from your part.

Note 1: If you are facing from ID to OD, then you should enter a value that is safely within the ID.

START FACING AT WHAT Z DIMENSION?

Enter a Z dimension just in front of your unfinished part.

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

As the tool feeds toward the center of the part, it should almost always go a little past centerline. This will remove the burr at the center. A typical value is -.04

CUT TO WHAT FINAL Z? (usually 0)

This is the finished Z dimension of the face.

DEPTH OF CUT PER PASS? (0 for single pass)

Enter the amount of material you would like removed during each stroke of the tool. A typical value might be .1 To take it all in one pass, enter 0 here. Or maybe just use choice 1) POINT instead of FACE (to drive the tool yourself).

Note 1: To suppress your machine's canned cycle (G75), enter a minus depth.

WHAT FEEDRATE (ipr) WOULD YOU LIKE?

Enter the feedrate in inches per revolution for this operation.

Note 1: 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.

Note 2: 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.



9) ROUGH

Format: ROUGH xstk_zstk_depth_x_z_safang_f_

HOW MUCH STOCK WOULD YOU LIKE LEFT IN X?

In order to rough an area or surface using G-ZERO you must follow 3 simple steps: 1) call up the ROUGH command and answer all the questions, 2) using COMP, POINT, RADIUS, LINE, and UNCOMP, define the surface you wish to rough, and 3) call up ROUGH again with all the same answers. When roughing, you will generally want to leave material on your part to clean up later with a finishing tool. Enter the amount of material you would like left on all diameters. A typical value would be .01

Note 1: If using a groove tool, then you probably wish to suppress the normal cleanup move that ROUGH makes before rapiding to each new cut. If so, give a minus number here.

HOW MUCH STOCK WOULD YOU LIKE IN Z?

When roughing, you will want to leave material on your part to clean up later with a finishing tool. Enter the amount of material to leave on all faces. A typical value is .005

Note 1: When roughing away from the chuck you will want Z stock added in the opposite direction, so enter this as a minus number.

DEPTH OF CUT PER PASS?

Enter the amount of material you would like removed during each stroke of the tool. A typical value might be .1

Note 1: If most of the material is gone, you may wish to make a semifinish clean up pass (G73) that still leaves material for a finishing tool. If so, then enter a zero here.

Note 2: To suppress your machine's canned cycle (G71), enter a minus depth. For undercuts, G-ZERO may work better. G-ZERO Apprentice always uses its own canned cycle.

WHERE IN X WOULD YOU LIKE TO START ROUGHING FROM?

You will usually want to start roughing from just beyond the outermost X dimension of your part.

STARTING ROUGHING WHERE, IN Z?

Almost always start roughing from just in front of the part. A typical answer would be .02

WHAT IS THE STEEPEST SAFE ANGLE WHEN PLUNGING? (usually 270 or 90)

If you are using your machine's G71 type canned cycle, then ignore this question. Your answer will correlate with the relief angle of your tool. For example, if your tool has a 40 degree relief, then the steepest plunge on the OD would be 180+40 or 220 degrees (ID would be 180-40 or 140). This lets G-ZERO skip certain unreachable undercut edges. Another tool (one with sufficient clearance) can be used later to reach the skipped area.

WHAT FEEDRATE (ipr) WOULD YOU LIKE?

Enter the feedrate in inches per revolution for this operation.

Note 1: 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.

Note 2: 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.

Programming Tips

The Rough command is always programmed in pairs. The second Rough command must be identical to the first Rough command (of the pair). Comp thru Uncomp describes the finished part geometry without lead-on or lead-off Points.

Optional: Program a lead-on (approach) Point before the first Rough command. Program a lead-off (retract) Point after the second Rough command (of the pair).

More programming tips:

- 1) Programming format
- 2) Rough with several clean up passes
- 3) SuperRough
 - 3a) Multiple undercut roughing
 - 3b) Rest area roughing
 - 3c) Example

1) Programming format

The most common Source programming format for a lathe roughing operation is:

1. MAT'L
2. TOOL
3. TYPE
4. FACE (optional)
5. ROUGH
6. COMP
7. (finish part geometry)
8. UNCOMP
9. UnROUGH

2) Rough with several clean up passes

To rough and then take several clean up passes:

1. MAT'L
2. TOOL
3. TYPE
4. FACE (optional)
5. ROUGH
6. COMP
7. (finish part geometry)
8. UNCOMP
9. UnROUGH
10. TOOL
11. STOCK (toggle on)
12. REPEAT (Comp thru Uncomp)
13. STOCK (toggle off)
14. REPEAT (Comp thru Uncomp)

3) SuperRough

Lathe v3.1 introduced two new roughing capabilities:

- a) Multiple Undercut Roughing
- b) Rest Area Roughing
- c) Example

3a) Multiple undercut roughing

Using SuperRough, G-ZERO can break complex contours (containing up to 99 sub-pocketed areas) into separate rough commands automatically. Just place a simple comment right before you call the ROUGH command:

SUPERG71 (for X direction roughing), OR
SUPERG72 (for Z direction roughing)

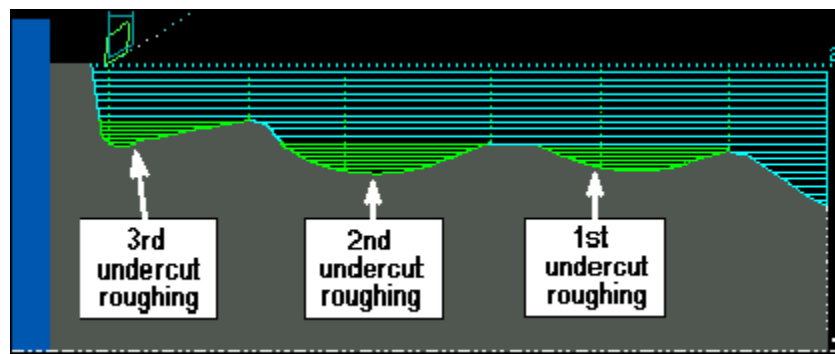
G-ZERO will rough the main area first, then retract the tool to your ROUGH's safe X and Z locations. Next, it will rapid to a point near the "earliest" (as defined by the order in which you entered the geometry) undercut and begin roughing that area. It then rapids to the next undercut, and so on.

OPTIONS:

Depth: To reduce the depth of cut when in undercuts, just add Depth followed by the desired depth of cut (eg.: SUPERG71 Depth.05).

Face: When using SuperRough on the face of your part, be sure to add the word Face to your comment. This tells G-ZERO to rapid away from the part in Z, not in X (eg.: SUPERG72 Depth.055 Face).

Example:



3a) Rest area roughing

Deeper undercuts may be unreachable by one tool. ROUGH asks you for a Safang so that you can "hide" unreachable areas. SuperRough remembers the hidden areas, then comes back later to polish off the "Rest of the Areas" (Rest Area for short) with another tool.

To tell SuperRough you want it to grab another tool, you must "stack" the rest area tool right after the main roughing tool. That is, you must call out 2 tool commands in a row. An example of the most common source programming format for a lathe "rest area" roughing is:

```
TOOL      (1st Roughing Tool)
TYPE
TOOL      (2nd Roughing Tool)
TYPE
SUPERG71 (SuperRough comment line)
ROUGH
COMP      (Roughing boundary)
...
UNCOMP
UnROUGH
```

Example:

Step 1: Original material



Step 2: 1st roughing tool at work



Step 3: 2nd roughing tool at work



Step 4: Material after roughing



3c) Example

Here is an example using SuperRough and 2 roughing tools (one ID and one OD) to rough out the face of a wheel:

```
1  MAT'L      od6 id0 zlim-1 zface0 zclamp.5 type1=303
2  TOOL 1    in/out1 *** PILOT HOLE
3  TYPE 29   mat0=M2 tnr0 size.437 axes0=XZ
4  DRILL     g74 rap.1 cut-.375 peck3 tip0 ret1 f.005
5  TOOL 2    in/out1 *** ROUGH OUTWARD
6  TYPE 6    mat0=ALUM/OXY tnr.0312 size.375 axes0=XZ
7  TOOL 3    in/out2 *** ROUGH REST AREA
8  TYPE 6    mat0=ALUM/OXY tnr.0312 size.5 axes0=XZ
9           SUPERG72 DEPTH.05 FACE
10 ROUGH     xstk.01 zstk.01 depth.1 x.45 z.05 safang115 f.009
11 COMP      ang92 lft/rt2 f.003
12 RADIUS    .5 type1 x0 z-.9
13 RADIUS    -.05 type0 (xc1.024695 zc-.7)
14 LINE      angle90
15 RADIUS    -.75 type1 x3.5 z0
16 UNCOMP    angle0
17 UnROUGH
```

Note 1: Tool 1 clears a start hole, so the next can feed straight in.

Note 2: Tool 2 is an ID rougher.

Note 3: Tool 3 is an OD "rest area" rougher, but it is placed in a very odd source location. Since SuperRough needs to know about both tools before it begins, we must "stack" two tools together when performing Rest Area Roughing.

Note 4: Comment just before ROUGH command must start with SUPERG72 (SUPERG71 will also work in some cases).

Note 5: Optional Depth value sets depth of cut in undercut area (area that is in shadow when 90 degree light is shined on contour).

Note 6: Optional Face comment MUST be present in this case to prevent tool from rapping in X before rapping out of hole in Z.



10) GROOVE

Format: GROOVE x_z_dp_w_skip_r_ir_a_

WHAT IS THE CURRENT X DIMENSION OF THE GROOVE SURFACE?

Give the X of the top of the groove. For an OD groove this is the maximum X, and for an ID groove this will be the minimum X.

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

Your groove has 2 walls. Enter the Z dimension for the wall that is closest to the face of your part. This will generally be a negative number.

HOW DEEP IS THE GROOVE?

Enter the distance from the top of the groove to the bottom.

Note 1: If you would like to give the full absolute diameter of the groove bottom, then enter it here as a negative number.

HOW WIDE IS IT?

Enter the distance from one side of the groove to the other. Measure this across the groove bottom, and ignore radii and chamfers for now.

Note 1: To enter the absolute Z dimension of the left wall of the groove, just enter it here as a negative number.

IS THE LEFT EDGE OF GROOVE AGAINST A SHOULDER: (0=no,1=yes)?

If you would like to avoid radiusing the left outside corner, then enter a 1 here to radius only the right side of the groove. Normally, it will be safe to radius (or chamfer) both outside corners so you can enter 0.

Note 1: Enter a 2 here if the RIGHT edge is against a shoulder.

Note 2: To suppress the finish pass (ROUGH ONLY) add 10 to your answer.

Note 3: FINISH ONLY by adding 20 to your answer (eg.: 21=right+finish)

BREAK OUTSIDE CORNERS WITH WHAT RADIUS?

If you wish to radius the outer corners of the groove, then enter the desired corner radius. If you don't want the corners broken, then enter 0. A typical value would be .010

Note 1: To put a 45 degree chamfer on the corners instead, just give the size of the chamfer as a negative here.

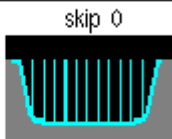


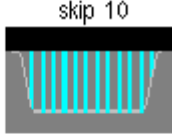
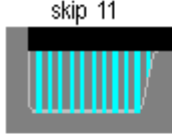
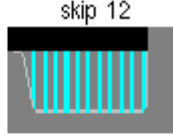
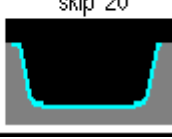
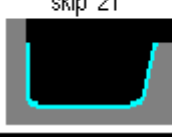
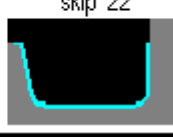
WHAT SIZE RADIUSSES IN BOTTOM OF GROOVE?

To generate 2 small radiusses in the bottom left and bottom right corners of your groove, enter the radius here.

ANGLE WALLS HOW MANY DEGREES?

Enter the taper angle for each wall of the groove. Note that if only the right wall tapers then you must back up 3 questions and give your tool corner radius as a minus number. Most groove walls have no taper, so your answer will usually be 0. For angles greater than 15 degrees, use ROUGH to rough the walls.

Programming Tips

	No Shoulder	Left Shoulder	Right Shoulder
ROUGH and FINISH	skip 0 	skip 1 	skip 2 
ROUGH ONLY	skip 10 	skip 11 	skip 12 
FINISH ONLY	skip 20 	skip 21 	skip 22 

NOTE: In the Groove cycle, when you are roughing without a finish pass, G-ZERO's picture does not actually show the gray material being removed. This is because the roughing process does not really define a cutting boundary.

Groove + Partoff

If you would like to have a radius or chamfer on your material before parting off, you can use GROOVE with your desired radius right before using the command PARTOFF. In this way, the cutting tool will cut the chamfer and then separate the material. Example:

```

1  MAT'L      od1.6 id0 zlim-1.63 zface.005 zclamp0 type4
2  TOOL 1     in/out2 ***
3  TYPE 10    mat1=C-5 tnr0 size/125
4  GROOVE     x1.6 z-.5 dp.25 w.125 skip0 r.05 ir0 a0 f0
5  PARTOFF    x1.7 z-.5 xcut0 catch1 f0

```



11) THREAD

Format: THREAD maj_tpi_min_S&_z_zfin_num&_a_

WHAT IS THE MAJOR DIAMETER OF THE THREAD?

This is usually the first dimension given. For example, a 1/4-20 thread has a major diameter of nearly .25

HOW MANY THREADS PER INCH?

Threads per inch (tpi) is usually the second dimension given. For example, a 1/4-20 thread has 20 tpi. A double lead 1/4-20 will have 10 tpi.

Note 1: To trigger the long-hand (G92) thread cycle, give a minus TPI.

WHAT IS THE MINOR DIAMETER OF THE THREAD?

In general, you will have to look this number up from a table, or in the Machinery's Handbook.

Note 1: G-ZERO offers a precise 60 degree minor diameter.

WHAT RPM WOULD YOU LIKE?

How fast do you want the spindle to turn as you cut this?

Note 1: To cut a square-form (acme) thread, enter this as a minus.

WHERE, IN Z WOULD YOU LIKE TO START?

You will usually want to start threading from just in front of your part. A typical value would be twice the thread pitch.

WHERE, IN Z DOES THE THREAD END?

In general, your answer will be negative.

HOW MANY PASSES WOULD YOU LIKE TO MAKE?

Give the total number of passes to cut this thread.

Note 1: Or just enter the D value for the G76 (depth of first pass).

Note 2: When G92 threading, if you would like a few spring passes, then enter the number of spring passes as a decimal. For example, to make 4 passes followed by 2 spring passes, enter 4.2. Note: You can only go up to 9 spring passes.

WHAT ANGLE WOULD YOU LIKE FOR THE INFEEED? (usually 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. If so, then enter the angle of the thread here. For most standard threads your answer will be 60 degrees.



12) DRILL

Format: DRILL g rap cut peck tip ret

WHAT G-CODE WOULD YOU LIKE TO DRILL WITH?

Note 1: You will normally drill with the 74 cycle.

Note 2: When cross or end drilling, you will use 81 or 83 instead.

Note 3: When tapping with the live head, you will normally use 84. Apprentice ignores this question.

START FEEDING FROM (rapid to) WHAT DIMENSION?

You will generally want to start from just in front of your part. A typical value might be .05

Note 1: When cross drilling, give a value larger than your OD.

DRILL TO WHAT FINAL DIMENSION?

Your answer will usually be negative.

Note 1: If this tool is a center drill or countersink, then you can have G-ZERO calculate this final Z depth for you. Be sure to read the next two help screens for help! Right now enter the Z dimension of the TOP of the countersink. Your answer will often be 0.

Note 2: When cross drilling, give a + value near your ID unless you actually wish to go ALL the way through the part.

PECK HOW MANY TIMES? (- for full retract)

Give the total number of pecks you would like the tool to make as it reaches the final depth. If the cycle you selected does not peck, then just enter a 1. A typical answer would be 4. If you must give a number bigger than 45, then give peck distance.

Note 1: You may enter a peck distance instead of the number of pecks.

Note 2: To have the drill back all the way out of the hole each time, just give a minus number like -4.

Note 3: If you are using a countersink, then you may enter the full countersink angle here, and G-ZERO will calculate how far to push the point of your tool into the hole. Read the next question's help page!

Note 4: When live head tapping, enter the threads per inch (TPI) here.

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

Note 1: You may wish to have the drill go far enough to break thru the part. If you enter 1 for yes, then G-ZERO will add 1/3 of the current tool diameter to the total cut distance.

Note 2: To set the amount of extra depth yourself, just enter the depth here as a minus number.

Note 3: For drills that have non-standard tip angles, enter the angle.

Note 4: If you are using the special countersink calculation then give the diameter that you would like to have left on your part. G-ZERO will figure out how far to feed into the hole. Apprentice ignores this question.

RETRACT TO WHAT DIMENSION? (usually = rap)

Occasionally you may want the tool to rapid further away between pecks/holes. Enter the value to retract back to. If you do not want this feature, then you MUST give the same value you gave for the rapid dimension.



13) BORE

Format: BORE x_ z_ xcut_ zcut_ depth_

RAPID TO WHAT SAFE X DIMENSION?

This routine is normally used to finish bore a hole. If, however, you call BORE with an OD tool, then it may be used as a crude roughing cycle. Give an X dimension that is safely away from your part.

START BORING AT WHAT Z?

Enter a Z dimension just in front of your unfinished part. A typical value might be .05 inches.

WHAT IS THE FINAL X DIMENSION OF THE BORE?

Give the desired finished diameter of this bore.

BORE TO WHAT FINAL Z DIMENSION?

Give the Z depth that you would like your boring tool to plunge. Your answer will usually be negative.

DEPTH OF CUT PER PASS? (0 for single pass)

Enter the depth of cut you would like removed during each stroke of the tool. A typical value might be .1. To take it all in one pass, enter 0 here. Or maybe just use choice 1) POINT instead of BORE (to drive the tool yourself).

Note 1: To suppress your machine's canned cycle (G74), enter a minus depth.



14) STOCK

Format: STOCK x_ z_

LEAVE HOW MUCH MATERIAL ON TURNED SURFACES FOR CLEANUP?

When you make a semi-rough pass you may wish to leave material on your part to clean up later with a finish pass. Enter the amount of material you would like left on all diameters. A typical value would be .005

Note 1: When milling with live tools, this is the amount of stock to leave on part walls.

HOW MUCH STOCK WOULD YOU LIKE TO LEAVE IN Z?

Enter the amount of material to leave on all faces. A typical value is .002

Note 1: When you cut a face that points toward the chuck, you will want Z stock added in the opposite direction, so enter this as a minus number.

Note 2: When milling with live tools, this is the amount of stock to leave on your pocket floors.



15) REPEAT

Format: REPEAT from_ thru_ ***

WHAT IS THE FIRST LINE NUMBER YOU WOULD LIKE TO REPEAT?

Enter the line number in this source program of the line that you already defined and would now like to re-use. Please remember that if you are repeating a surface definition, you may wish to call up COMP before calling up this command. If you can't remember the line number, Press PgUp to look back.

Note 1: To reverse the direction your cutter takes along a surface you may use repeat backwards. For example, FROM 14 THRU 12. Be sure not to repeat COMP, UNCOMP, ROTATE or MILL this way.

Note 2: To mirror your cutter path in Z, make this line number a minus number.

WHAT IS THE LAST LINE NUMBER OF THE GROUP YOU WISH TO REPEAT?

Many times you may wish to repeat a block of several lines at a time. For example you could repeat FROM 12 THRU 14. If you only want to repeat one single line, then give the same answer you gave for the previous question.

Note 1: To reverse the direction your cutter takes along a surface you may use REPEAT backwards. For example, FROM 14 THRU 12. Be sure not to repeat COMP, UNCOMP, ROTATE or MILL this way.

Note 2: To mirror your cutter path in X, make this line number a minus number.



16) ROTATE

Format: ROTATE angle_ xpiv_ zpiv_

ROTATE HOW MANY DEGREES?

Give the angle that you would like to have everything from here on rotated. Your answer should be in decimal degrees. Be sure to cancel this when you're done by ROTATEing 0 degrees. A TOOL command will also cancel ROTATE.

Note 1: You may use this command to translate as well as rotate. If you wish to move something, but not revolve it, enter 0.

Note 2: If you know the angle in degrees, minutes, and seconds, then you may convert by using the " key. For example, if you type 22 degrees, 30 minutes, and 15 seconds (22.3015) and the " key G-ZERO will convert to 22.5042 degrees.

WHERE IS THE PIVOT POINT IN X? (or translation amount)

This will generally be the X dimension of the point about which some angled dimensions are given. Often, your answer is zero.

Note 1: If you are just shifting, then give the amount that you would like to displace in X.

WHERE IS THE PIVOT IN Z? (or translation amount)

This will generally be the Z dimension of the point about which some angled dimensions are given. Your answer is often zero.

Note 1: If you are just shifting, then give the amount that you would like to displace in Z.



17) MAT'L

Format: MAT'L od_id_zlim_zface_zclamp_type&_

WHAT IS THE MAXIMUM DIAMETER OF THIS PART?

In order to make optimum use of your computer screen, you must define your part's size. Give the major outside diameter (OD) of your part material. If you are unsure, you may calculate the answer using the +, -, *, and / keys to add, subtract, multiply, and divide.

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

For normal solid round stock, your answer is zero. If the inside of the part has already been removed, then enter the inside diameter (ID) now.

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

You will generally want to program with Z0 at the face of your part. Give the left-most Z dimension that you would like to see on your display. Quite often, this will be the length of your part material. Your answer will probably be -.

Note 1: If you wish to program with Z0 near your lathe's chuck and not at the face of your part, give your answer as a + value.

HOW MUCH EXTRA MATERIAL WILL YOUR PART FACE FIRST HAVE?

When you first load the part(or barfeed/pull it), you will probably have a little extra sticking out in front. Enter that amount now. A common value might be .05 inches.

HOW MUCH EXTRA MATERIAL WILL BE CLAMPED IN THE CHUCK?

A typical answer might be .5 inches.

WHAT IS THE PART MADE OF?

Pick a material type from the window currently on your screen.

Note 1: To add material types, exit to DOS and type WS DAT\17.6 Then type FEEDS to add SFM and chipload information to each tool.

Programming Tips

Lathe v3.1 incorporated an Enhanced Material! The original MAT'L picture can now be graphically represented with an elaborate shape. In this way, if you want to work on an already processed material, it can be represented with its real shape rather than a rectangle.

In order to have a shaped material, you need to program a COMP/UNCOMP right after the command MAT'L. Example:

```

1  MAT'L          od4 id0 zlim-5 zface0 zclamp0 type4=ALUMINUM
2  COMP          ang0 lft/rt2 f.01
3  POINT         x4 z-4
4  RADIUS        .5 type2 x2.5 z-3 (xc3.585582 xc-2.890388)
5  RADIUS        -.5 type2 x3 z-2 (xc1.969224 zc-2)
6  POINT         x2 z0
7  UNCOMP        angle0

```



Note 1: To avoid (disable) shaped material, insert a comment right after MAT'L command.



19) REDRAW

Format: REDRAW from_ thru_ type_

WHAT IS THE FIRST LINE YOU WOULD LIKE TO REDRAW?

To see your entire program, you will generally enter 1 here. If you only want to see one section, then enter the first line number in that section. **SHORT-CUT:** The F2-F5 keys provide a simpler redraw.

Note 1: When plotting, enter a -1 here to skip plotting the frame.

LAST LINE NUMBER TO REDRAW?

To see your entire program, just enter 999 here. If you only want to see one section, enter the last line in that section.

WHAT TYPE OF PICTURE? (0=part,1=tool,10=full part)

Enter 0 to see the part boundary. If you enter 1, G-ZERO will display your profiling tool so you can check for clearance. Enter 10 to see the entire part (+X and -X). **SHORT-CUT:** Press F2 or F3

Note 1: To redraw the end view of your part, enter -1. **SHORT-CUT:** Press F4

Note 2: To redraw the unwrapped side view, enter -2. **SHORT-CUT:** Press F5



20) POST

Since your CNC does not understand G-ZERO, this source program must be processed and converted into a language your machine will recognize. This machine tool language (G-code or tape file) is stored in a separate file with extension .T.

Once you press 20) POST, G-ZERO displays a window from where you can choose the Post Processor to create a G-code customized to a particular CNC.



21) PRINT

Format: PRINT from_ thru_

WHAT IS THE FIRST LINE NUMBER YOU WISH TO PRINT?

Whenever you make changes to your source program, it is helpful to get a printout (listing) from your printer. Enter the line number you would like to see listed on the printer. For a listing of the entire program, give 1 as your answer.

Note 1: To suppress the preliminary tool printout, enter 0.

WHAT IS THE LAST LINE NUMBER TO PRINT?

Many times you may wish to list a block of several lines at a time. For example, you could list FROM 12 THRU 14. If you only want to list one single line, then give the same answer you gave for the previous question.

Note 1: Enter a 0 here to print all the way to the end.



22) ALTER

Format: ALTER line_ from_ to_

WHAT LINE WOULD YOU LIKE TO ALTER?

Enter the line number that needs to be changed.

WHAT VALUE DO YOU WANT TO CHANGE?

Enter the value that is incorrect in your source program.

Note 1: This command will only change numeric values. To change one command to another, back-up and use the special function in INSERT to make your change.

REPLACE WITH:

Enter the value that you would like to put in place of the incorrect value. G-ZERO will then search the program for any occurrences of the incorrect value. When one is found, you will be asked if it should be replaced. At that time, answer Y for Yes or N for No or A for All or D for Done



23) INSERT

Format: INSERT after_

WHAT LINE NUMBER WOULD YOU LIKE TO INSERT AFTER?

Give the line number that is just in front of the spot you would like to add one single command at.

Note 1: If you need to change from one command to another (e.g.: POINT to RADIUS), then you may use INSERT as a sort of super-ALTER. Just enter the line number to change as a minus number.

Programming Tips

Tip 1: Instead of inserting a group of lines, program them at the bottom of the Source and use MOVE to insert the group at the correct location in the Source.

Tip 2: G-ZERO automatically renumbers the Source program.



24) DELETE

Format: DELETE from_ thru_

START DELETING AT WHAT LINE NUMBER?

Give the line number that you would like to erase from your program. If you wish to delete a group of lines, enter the first line number of the group.

LAST LINE TO DELETE?

If you only want to delete one single line, then give the same answer you gave for the previous question. However, if you wish to delete a block of several lines at a time, give the last line number of the group. For example, you could delete lines FROM 12 THRU 14.

Note 1: You will be asked (Y/N/I). Type N (NO) to abort the DELETE command. Type I (INVERT) to restore lines that were previously deleted.

Programming Tip

Tip 1: Instead of deleting the last line of a Source, press <Esc> then Y to delete.

Tip 2: To undelete a line or group of lines, DELETE again but press I for Invert at the Y/N/I prompt.

Tip 3: Lines deleted at the very end of your program are always permanently erased. There must be at least one “live” line at the end of your source program to take advantage of the restore (Invert) capability.



25) MOVE

Format: MOVE from_ thru_ after_

WHAT IS THE FIRST LINE NUMBER OF THE GROUP YOU WISH TO MOVE?

When you need to change the order of events in your program, use this command to rearrange things. Enter the line number that you would like to shift somewhere else.

WHAT IS THE LAST LINE NUMBER OF THE MOVE GROUP?

Many times you may wish to move a block of several lines at a time. For example, you could move FROM 12 THRU 14. If you only want to move one single line, then give the same answer you gave for the previous question.

WHAT LINE NUMBER WOULD YOU LIKE TO MOVE THE GROUP AFTER?

Give the line number that is just in front of the spot you would like to shift the misplaced lines to.

Programming Tip

Tip 1: If you want to delete some lines and keep the Source lines numbered perfectly, use MOVE instead of DELETE. Use MOVE to transfer the lines to the end of your source program, and then press <Esc> to delete them one at a time.



26) UNEDIT

Recalculates all unknown values (such as type0 RADs and unknown LINE angles) and updates the current source program with any changes made. Call this command after any major edit, or after a whole source program is copied. This command does not add any line code to the source program.



27) BASIC

Use the command 27) BASIC to run or create BASIC programs. This command uses the program QBASIC.EXE located in your C:\WINDOWS\COMMAND directory. If you don't have the QBASIC.EXE file, copy it from your Windows CD (OTHER\OLDMSDOS directory)

Using a BASIC program

1. Select 27) BASIC when you are in any G-ZERO Source program.
2. In the "Basic File to Open" window, select the Basic file you want to open (files with extension .B) and click the Open button. (Example: Lellipse.b)
3. The Basic system will prompt you several questions.

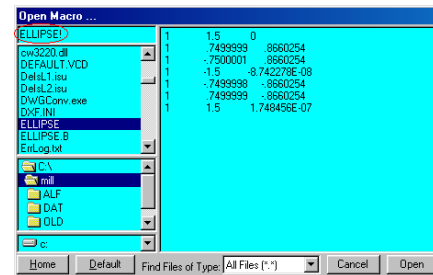
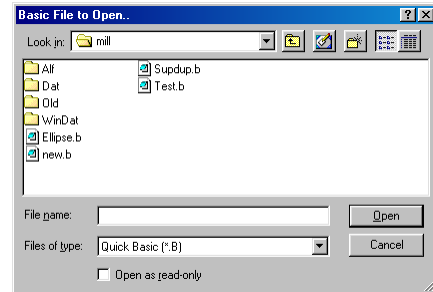
Example:

```
WHAT IS THE MAJOR (Z) DIAMETER? 6
WHAT IS THE MINOR (X) DIAMETER? 2
STEP ANGLE? 10
```

After you answer the questions, G-ZERO will bring you back to your source code. Note: If your system did not automatically close the DOS window, click the "X" box in the upper right corner of the window to close it.

G-ZERO should have created a macro file in the same directory (Example: Ellipse.u). File name depends on the way the Basic file was programmed.

4. In the Source program, type # (pound sign) to open the *Open Macro* window.
5. Select the macro file just created (and type an exclamation mark right after its name to import source lines). Note: You can also select the new file from the Mill directory (make sure to select the correct file extension or simply set it to display *All Files* in the *Find Files of Types* box in the bottom of the window).
6. Click *Open* to accept the file selected and return to the main programming window. G-ZERO will add a link to the macro (plus all the new source codes if an exclamation mark was included right after the file name)



Creating a new BASIC program

1. Start qbasic.exe

Note: You need to have the file qbasic.exe in your C:\DOS directory. If you don't have it, copy the qbasic.exe file from your Windows CD to C:\DOS:

If you have Windows 95 CD: qbasic.exe is located in OTHER\OLDMSDOS directory of CD.

If you have Windows 98 CD: qbasic.exe is located in TOOLS\OLDMSDOS directory of CD.

2. You can either make a copy of an existing basic file (e.g.: ellipse.b) and modify it, or create a basic file from scratch. Here is an example of a simple basic program for an ellipse:

```
5 OPEN "ELLIPSE.U" FOR OUTPUT AS #1
7 INPUT "WHAT IS THE MAJOR (Z) DIAMETER";MAJOR:MAJOR=MAJOR/2
8 INPUT "WHAT IS THE MINOR (X) DIAMETER";MINOR:MINOR=MINOR/2
9 INPUT "STEP ANGLE";STP:IF 360/STP<>INT(360/STP) THEN PRINT "MUST DIVIDE
EVENLY INTO 360":GOTO 9
10 FOR ANG=0 TO 90+.0001 STEP STP
20 A=ANG*3.1415926536#/180:X=COS(A)*MAJOR:Y=SIN(A)*MINOR
25 Z=X-MAJOR
```

```
26 X=Y*2
30 PRINT X;Z
35 PRINT #1,1,X,Z
40 NEXT
50 SYSTEM
```

Note: You can use any text editor to create the Basic file; make sure to save the file with extension .b.



28) BOLTCRC

Function: BOLTCRC dia_num_st_qu_

WHAT DIAMETER ARE THE BOLT-CIRCLE HOLES ON?

Enter the diameter of the bolt-circle. Remember, the dimension may be given on the blueprint as a radius, in which case you need to multiply by 2.

Note 1: Enter your number as a minus to go around clockwise.

HOW MANY HOLES WOULD YOU LIKE?

Enter the total number of holes to be drilled.

WHAT ANGLE IS THE FIRST HOLE AT?

Give the angle (in degrees) from the center of the bolt-circle to the first hole you want drilled.

WHAT ANGLE IS THE LAST HOLE AT? (usually same as first)

Give the angle (in degrees) from the center of the bolt-circle to the last hole you want drilled. You may occasionally be required to add or subtract 360 from this number in order to obtain the desired results.

Note 1: G-ZERO will calculate the angle for the last hole on a FULL bolt-circle automatically if you just give the same angle you gave for the start (first hole will not be drilled again).

Programming Tips

Tip 1: If you can't figure out the start angle of the bolt-circle, look at the compass in the lower left corner of the G-ZERO screen. Mentally place the compass on top of the bolt-circle with the compass center on top of the bolt-circle center. Which axis (0, 45, 90, 135, 180, 225, 270, 315) lines up with a hole on the boltcircle?

Tip 2: Program single bolt-circles the same as full bolt-circles.

Tip 3: Program partial bolt-circles the same as full bolt-circles except that you need to give the angles of both first and last holes.



29) DEADTAP

Format: DEADTAP tpi_zstart_zdepth_

HOW MANY THREADS PER INCH FOR THIS TAP?

Threads per inch(tpi) is usually the second dimension given. For example, a 1/4-20 thread has 20 tpi.

START FEEDING FROM WHAT Z DIMENSION?

Generally, you will start feeding from just in front of the part. A typical answer would be 2 times the pitch.

TAP TO WHAT FINAL Z DIMENSION?

Give the depth that you would like your tap to plunge. Be sure to allow for overtravel due to deceleration of the spindle.



30) SHIFT

Format: SHIFT x_ z_

WHAT NUMBER WOULD YOU LIKE ADDED TO EVERY X DIMENSION?

It is often helpful to have all the X dimensions on your tape offset by a certain amount. This moves the zero point of the tape. Enter the amount you would like added to all X dimensions. To subtract, give a minus number.

WHAT NUMBER SHOULD BE ADDED TO EVERY Z DIMENSION?

It is often helpful to have all the Z dimensions on your tape offset by a certain amount. This moves the zero point of the tape. Enter the amount you would like added to all Z dimensions. To subtract, give a minus number.



31) PARTOFF

Format: PARTOFF x_ z_ xcut_ catch_

RAPID TO WHAT SAFE X BEFORE STARTING TO PARTOFF?

Enter the OD of your part plus a little extra.

CUT OFF AT WHAT Z DIMENSION?

This is almost always the length of your part, except that Z is usually a minus number.

FEED TO WHAT FINAL X DIMENSION? (often 0)

For a solid part, you will generally feed all the way to X0. For tubes you may wish to feed to the ID of your part.

WOULD YOU LIKE THE PARTS CATCHER ACTIVATED? (1=yes,0=no)

If your lathe has a parts catcher and you would like to engage it to catch the part during PARTOFF, then enter a 1 here.



32) SCALE

Format: SCALE x_ z_

WHAT SCALE WOULD YOU LIKE IN X?

Enter the number that you would like all X dimensions to be multiplied by. Examples: (a) for a half scale model, enter .5; (b) to offset for a 6% shrinkage, enter 1.006.

Note 1: You must be certain to lie about your cutter diameter so that the scaling effect is neutralized. (eg.: if you are using a 1/2 inch cutter at a scale of 1.006, then give your tool diameter as $.5/1.006=.497$)

WHAT SCALE WOULD YOU LIKE IN Z? (usually same as x)

Enter the number that you would like all Z dimensions to be multiplied by.

Note 1: Giving a number different than the X scale will produce undefined results when machining circles.



33) TAIL

Format: TAIL in/out _ zmax_ zmin_

WOULD YOU LIKE TAILSTOCK IN OR OUT? (1=in,0=out)

If your lathe has a tailstock and you would like to engage it (press it against the part), then enter a 1 here. To pull the tailstock away from the part, enter a 0.

WHAT IS THE MAXIMUM (FARTHEST FROM CHUCK) Z FOR THIS OPERATION?

WHAT IS THE MIN Z? (NEAREST CHUCK)



34) PIPE

Format: PIPE ret_tpf_X_Z_clr_leads_

The PIPE command works in conjunction with 11)THREAD. You must place PIPE just before THREAD in your source program.

HOW WOULD YOU LIKE TO RETRACT? (0=straight,1=chamfer)

Enter 0 if you would like a straight retraction (M24 on most machines). Enter 1 for 45 degree retract (M23). **WARNING:** Do not enter 1 if you are threading to a square shoulder!

TAPER HOW MANY INCHES PER FOOT? (usually .75)

The angle of a thread is usually defined as the difference in diameter (in inches) for every foot of thread length.

WHAT IS THE PITCH DIAMETER?

Normally, enter the major diameter at the front or back of your thread. If at the front, then give the GAUGELEN (next question) as 0. If at the back, then GAUGLEN equals Finish Z of thread.

WHERE IS THE DIA MEASURED IN Z? (aka: gauge len)

Z zero is a common place to measure threads. Or sometimes the Z finish dimension of your thread. You must enter a Pitch Dia (previous question) to activate this calculation.

HOW MUCH CLEARANCE AS TOOL RAPIDS?

As tool pulls out, how far away (in X radius) should it pull away from the thread?

WARNING: On some cycles, this causes Z to finish much deeper.

HOW MANY LEADS (usually 1)?

For a multi-lead thread, give the number of leads. For a regular thread, just enter 1.



35) LETTER

Format: LETTER size_x_y_sp_lr_r_a_***_

HOW TALL WOULD YOU LIKE THE LETTERS TO BE?

This lettering routine is an optional program that will help you engrave numbers, letters, and other characters into your part. Right now enter the character's height.

Note 1: To engrave a mold or die with mirrored characters, just enter this value as a minus number.

WHAT IS THE X DIMENSION OF THE FIRST CHARACTER CELL?

Each number and letter has its own personal "space" called a cell. There is always extra room in the right side of each cell to help keep a little space between characters. Enter the X value of the left-most edge of the first character's cell.

Note 1: If you are lettering around a radius, give the X of the radius' center.

WHAT IS THE Y DIMENSION?

Enter the Y value of the bottom of your line of characters.

Note 1: If lettering on a rad, then give the Y of the rad's center.

ENTER THE LETTER SPACING: (normally=height)?

Half inch tall letters will normally have half inch spacing (from startpoint to startpoint). To stretch letters, give a value larger than the height. A smaller number squashes.

Note 1: To set the spacing only, WITHOUT STRETCHING, give a minus value. (Often needed when inside a radius)

WOULD YOU LIKE LETTERING JUSTIFIED: (1=left, -1=right, 0=centered)?

Most straight line lettering is left justified (text flows toward the right from the start dimension). To make the text end at the given point, use -1 to push text left. Enter 0 if you would like text centered about the given point.

Note 1: Enter the slant angle (italics) as a 3 place decimal (eg: for a 15 degree slant of left justified text, enter 1.015)

LETTER AROUND WHAT SIZE RADIUS: (0 for straight line)?

To letter along the outside of a circle, enter its radius. Give a minus radius to work inside.

PUT AT WHAT ANGLE ON THE RADIUS?

Most outside work will be at nearly 90 degrees. Most inside work at 270. Unlike straight line lettering, this angle controls the CENTER of the letter cell, not the edge.

WHAT WOULD YOU LIKE IT TO SAY?

Enter letter(s) or word(s) you want to mill.



36) TYPE

Format: TYPE _ mat&_ tnr_ size&_ axes_

WHAT TYPE OF TOOL IS THIS?

Enter the number that most closely matches.

WHAT IS THE TOOL(insert) MADE OF?

Pick a material type from the window currently on your screen.

Note 1: If you don't see the desired material, then you must add it to this tool's data file. For example, to edit ID tool type 24, exit to DOS and type: FEEDS. Then edit IDTOOL24.F

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

Most inserts have a tnr of .03 or .015 or even .008

Note 1: For sharp tools or tools without a tnr, enter 0.

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

For inserts, enter the inscribed circle of the tool. For drills and endmills, enter the tool diameter. For grooving and partoff tools, enter the tool width. For threading tools, enter the thread's tpi (threads per inch). For TAPPING tools, enter the major diameter.

WHICH AXES WILL YOU BE PROGRAMMING)?

Normally, you will be programming in X diameters and Z. However, when live tooling is active, you probably will think polar. That means when you cut on the face of your part, you will program in X and C. when you cut on the side, you will most likely program in Z and C.

Note 1: Types 5 and 6 are only for lathes with a Y axis.

Note 2: If you have the WRAP option, then yo may program in inches instead of degrees. Activate by adding the arc break stepover to your answer. For example, 3.005 will translate XY' contours to XC output with arcs broken every .005 inches.

0=XZ
1=XC
2=ZC
3=XY'
4=ZY'
=====
Y AXIS
5=XY
6=ZY



37) MILL

Format: MILL rap_cut_passes_ret_plungef_f_

RAPID TO WHAT DIMENSION BEFORE KICKING TO FEED MODE?

The mill command will automatically wait until a location is given before moving the cutter at rapid to that location. You will generally want to start from just in front of your part. A typical value would be .02

Note 1: 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.

Note 1: When cross milling, give a + 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 rev. The answer you see in default is based on the part material and your tool.

WHAT FEEDRATE (ipr) WOULD YOU LIKE?

Enter the feedrate in inches per revolution for this operation.

Note 1: 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.

Note 2: 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.



39) FGROOVE

Format: FGROOVE x_z_dp_w_skip_r_ir_a_

WHAT IS THE X DIMENSION OF THE GROOVE ID?

Your face groove has 2 walls. Give the X dimension of the wall closest to the centerline of your part.

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

Enter the Z dimension of the face that the groove is cut into.

HOW DEEP IS THE GROOVE?

Enter the distance from the face of the groove to the bottom. Your answer **MUST** be positive.

HOW WIDE IS IT?

Enter the distance (radial, not diameter) from the ID of the groove to the OD. Your answer **MUST** be positive. Measure this across the groove bottom, and ignore radii and chamfers for now.

WOULD YOU LIKE TO AVOID RADIUSING THE OD EDGE: (0=no,1=yes)?

If your groove is pushed up against a shoulder, then enter a 1 here to radius only the ID edge of the groove. Normally, it will be safe to radius (or chamfer) both outside corners, so you can enter 0.

Enter a 2 here if the inner edge is against a shoulder. To suppress the finish pass (ROUGH ONLY) add 10 to your answer. FINISH ONLY by adding 20 to your answer (eg.: 21=inner+finish)

BREAK OUTSIDE CORNERS WITH WHAT RADIUS?

If you wish to radius the outer corners of the groove, then enter the desired corner radius. If you don't want the corners broken, then enter 0. A typical value would be .010

Note 1: To put a 45 degree chamfer on the corners instead, just give the size of the chamfer as a negative here.










WHAT SIZE RADIUS IN BOTTOM OF GROOVE?

To generate 2 small radiuses in the bottom ID and bottom OD corners of your groove, enter the radius here.

ANGLE WALLS HOW MANY DEGREES?

Enter the taper angle for each wall of the groove. Note that if only the ID wall tapers then you must back up 3 questions and give your tool corner radius as a minus number. Most groove walls have no taper, so your answer will usually be 0.

Programming Tips

	No Shoulder	Left Shoulder	Right Shoulder
ROUGH and FINISH	skip 0 	skip 1 	skip 2 
ROUGH ONLY	skip 10 	skip 11 	skip 12 
FINISH ONLY	skip 20 	skip 21 	skip 22 

Note: In the FGroove cycle, when you are roughing without a finish pass, G-ZERO's picture does not actually show the gray material being removed. This is because the roughing process does not really define a cutting boundary.



41) PECK

Format: PECK amt_ liftup_ groovestep_

WHAT IS THE PECK DISTANCE?

When you are FACING, ROUGHING, BORING, GROOVING, or PARTING OFF, you might have problems with long, stringy chips. To break the chips, call this PECK command after selecting a tool but BEFORE the main command. PECK stays in effect until you cancel it with a PECK amount of 0, or until you change tools.

Note 1: To disable canned cycle pecking (G75 or G74 type) enter this as a minus number.

LIFT UP HOW FAR BETWEEN PECKS?

At the end of each peck, the tool will stop and backup a bit. Enter the distance to lift before continuing its cut. A typical answer might be .005

STEPOVER HOW FAR BETWEEN EACH PASS WHEN GROOVING?

If you would like to override GROOVE's calculated stepover, then enter the step amount here. Normally just enter 0.

