

G-ZERO Mill







































Reference - Commands



Contents

Commands	3
Fundamentals	4
1) POINT	5
2) RADIUS	6
3) LINE	7
4) COMP	8
5) UNCOMP	10
6) TOOL	11
7) MILL	12
8) DRILL	14
9) RECT	16
10) ROUND	17
11) BOLTCRC	18
12) GRID	19
13) COPY	20
14) STOCK	21
15) REPEAT	22
16) ROTATE	24
17) MAT'L	27
19) REDRAW	28
20) POST	29
21) PRINT	29
22) ALTER	30
23) INSERT	30
24) DELETE	31
25) MOVE	31
26) UNEDIT	31
27) BASIC	32
29) ROUGH	35
30) SHIFT	37
31) STATION	38
32) SCALE	39
33) Z MOVE	40
34) CURFIT	41
35) LETTER	42
36) MULTIPLY	43
37) FORTHAX	45
38) +VIEW	46
39) +TILT	47
40) +TRIM	49
41) +PROJ	50

Commands

		Programming	Non-Programming	Option
	1) Point	■		
	2) Radius	■		
	3) Line	■		
	4) Comp	■		
	5) Uncomp	■		
	6) Tool	■		
	7) Mill	■		
	8) Drill&Tap	■		
	9) Rect	■		
	10) Round	■		
	11) Boltrc	■		
	12) Grid	■		
	13) Copy	■		
	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		■	
	29) Rough	■		
	30) Shift	■		
	31) Station	■		■
	32) Scale	■		
	33) Z move	■		
	34) Curfit	■		
	35) Letter	■		
	36) Multiply	■		
	37) Fourth	■		
	38) +View	■		■
	39) +Tilt	■		■
	40) +Trim	■		■
	41) +Proj	■		■

Fundamentals

Calling commands

G-ZERO Mill v4.0 currently 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)

Availability of 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.

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: In the TOOL command, when G-ZERO asks you for the diameter of the tool, if you answer 30, it will not accept that value.

Override

If you are sure that the command you want to use is appropriate, or the values you want to enter are correct, and you want to override the computer, press the letter “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.

Example: If you want to use a 8” diameter tool and G-ZERO does not let you enter that value, press: number “8”, letter “O” (to override), and the Enter key.

The letter “O” key forces G-ZERO to accept your input.

“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:

- ♦ **%(JUMP CLAMPS HERE)** 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
- ♦ **POCKET THEN OD** Comments without a % sign and parentheses are for the programmer



1) POINT

Format: POINT x_ y_ f_

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

Use your keyboard (keypad) to enter the X dimension of the point you want to program. 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 might be negative.

Note 1: If you don't know this dimension, you can let G-ZERO calculate it for you: Treat it as an "unknown" RADIUS with radius = 0 (See: RADIUS).

Note 2: If this is just a plunge point, you can give the approximate location with a mouse (See: MouseRuf)

Note 3: If you would like to enter this dimension as an incremental distance from the last X dimension you gave, then use the @ key. For example, to move 1.5 inches from the last point given, enter @ + 1.5

Note 4: If you are in polar mode, then enter the radius instead of the X value. (See: Polar Coordinates)

WHERE, IN Y?

Use your keyboard (keypad) to enter the Y dimension of the point you want to program. 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 might be negative.

Note 1: If you don't know this dimension, you can let G-ZERO calculate it for you: Treat it as an "unknown" RADIUS with radius = 0 (See: RADIUS).

Note 2: If this is just a plunge point, you can give the approximate location with a mouse (See: MouseRuf)

Note 3: If you would like to enter this dimension as an incremental distance from the last X dimension you gave, then use the @ key. For example, to move 1.5 inches from the last point given, enter @ + 1.5

Note 4: If you are in polar mode, then enter the angle instead of the Y value. (See: Polar Coordinates)

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

How fast, in inches per minute (ipm), would you like the tool to feed toward this point? A typical answer would be 10.

Note 1: This question will NOT be asked when...

Note 2: If you would like the tool to move at rapid traverse, then give zero as your answer.

Note 3: If you would like the tool to continue feeding at whatever feedrate is currently active, then enter 99.99. This will kick out of rapid mode and leave the feedrate unchanged.

Programming Tips

Tip 1: Locates the XY-center of a Point to be drilled.

Tip 2: Programmed after MILL, POINT locates the XY-center of the location to begin cutting.

Tip 3: After COMP, a POINT is the finished part dimension that is wanted. G-ZERO will do any cutter radius compensation that is required.

Tip 4: When defining an OD profile, avoid leaving a dwell mark by programming (a) an "approach" Point before COMP and (b) a "retract" Point after UNCOMP. These two Points are usually the same if cutting a "closed loop" profile. Add the amount of the tool radius to these X and Y dimensions when they are programmed outside the COMP-UNCOMP group.

Tip 5: To break a corner, program a RADIUS instead of a Point. Don't forget to define as either positive or negative.



2) RADIUS

Format: RADIUS _ type_ x_ y_

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

Key in the radius of the circle or arc. If the tool will be travelling inside the circle instead of outside, your answer should be negative.

Note 1: Some blueprints show this dimension as a diameter, in that case, divide by 2.

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 a very small negative number such as -.0001 for an inside corner.



WHICH TYPE OF RADIUS? (1=center 2=corner 0=unknown)

Your answer should be one of these options: 1, 2, or 0

Center Radius: Key in 1 if the center of the circle is dimensioned or is easy to calculate.

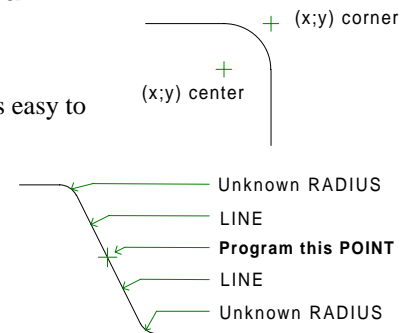
Corner Radius: Key in 2 if the circle is on an inside or outside corner that is dimensioned or easy to calculate. There MUST have a line defined before this radius and a line defined after this radius. G-ZERO will calculate the center of the radius and display it within parenthesis.

Unknown Radius: Key in 0 if you really don't know much about the

circle at all or if you only know one of the dimensions. G-ZERO will calculate the center of the radius and display it within parenthesis.

Note 1: G-ZERO does NOT allow 2 consecutive "unknown" radii even if they are separated by a line. If it is possible to find a point within the line, program that point before the second unknown radius.

Note 2: If you type a 0 and G-ZERO finds the wrong radius (arc drawn using the other end of the line), then you should enter a -1 to tell G-ZERO to find the other answer.



WHAT IS THE X DIMENSION OF THE RADIUS?

Enter the X value of the center (type=1) or corner (type=2) of the radius.

Note 1: This question will NOT be asked for unknown radii (type=0).

WHAT IS THE Y DIMENSION?

Enter the Y value of the center (type=1) or corner (type=2) of the radius.

Note 1: This question will NOT be asked for unknown radii (type=0).

Programming Tips

Tip 1: A RADIUS after COMP or before UNCOMP should always be a Center Radius (type=1)

Tip 2: To "break" corners, program a Corner Radius (type=2) instead of a POINT.

Tip 3: Program a Blend-on Radius after COMP and a Blend-off Radius before UNCOMP. These radii should:

- ♦ always be Inside Radii (negative radii)
- ♦ always be Center Radii (type=1)
- ♦ always be tangent to the surface to be cut.
- ♦ be at least twice the diameter of the cutting tool








3) LINE


Format: LINE angle_

MOVE ALONG THE LINE IN WHAT DIRECTION?

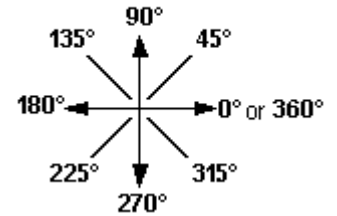
Give the angle that your tool will be moving as it travels along the line. Your answer should be in decimal degrees. If you don't know the line angle, have G-ZERO calculate it for you by pressing the (key.

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 (   ) for 0°, 90°, 180°, or 270° combined with your calculation. Example: 100° could be entered as  + 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 T 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



Programming Tips

Tip 1: Programming a LINE is required between every POINT and RADIUS.

Tip 2: You can enter Lines as positive angles (273°) or negative angles (-87°).

Tip 3: 0° and 360° are the same!



4) COMP

Format: COMP angle_cl/con_lookahead_

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. See Programming Tips below.

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

IS CUT DIRECTION (4)1=CLIMB or (4)2=CONVENTIONAL?

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 your tool is on the LEFT side of the cutting path, then your answer will be 1 (climb cut); if your tool is on the RIGHT side of the cutting path, then your answer will be 2 (conventional cut).

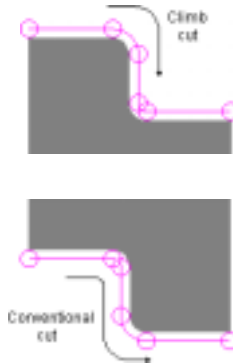
Note 1: To drive the cutter along the centerline of a contour, enter a 0 (zero). When entering radii (which must be inside or outside), answer as though you are climb cutting. G-ZERO does not offset for tool radius.

Note 2: To use your CNC's G41/G42 cycles, type 41 for climb cutting (instead of 1) and 42 for conventional cutting (instead of 2).

- ◆ climb 41 or 1
- ◆ conventional 42 or 2
- ◆ centerline 0

Note 3: To call up an unusual comp offset number, you must add a decimal offset. For example: to output G42D04, enter 42.04 here.

Note 4: 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. For example: -1 or -41, -2 or -42, -.0001 (centerline)



WOULD YOU LIKE TO LOOK AHEAD FOR GOUGE AVOIDANCE?

(0=no,1=yes)

When using an oversized endmill to rough a contour, you may wish to have G-ZERO check to see if the cutter fits into all the little nooks and crannies.

Note 1: On long contours, it takes quite a long time to process. To speed things up, limit the number of elements of lookahead by entering this number of elements here.

Note 2: To doublecheck your program for gouging, select F3 or F4 (redraw with tool) at the end of the program.

Programming Tips

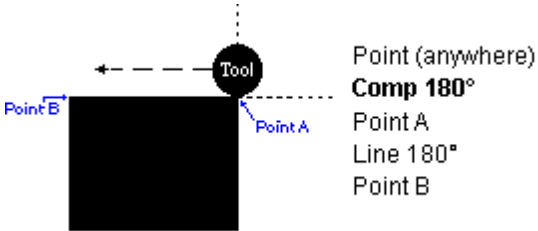
Tip 1: You must program either a POINT or RADIUS after the COMP command.

Tip 2: **Blend-on / Blend-off Radii.** The tool can automatically “blend” on the part by cutting further around the first or final RADIUS.

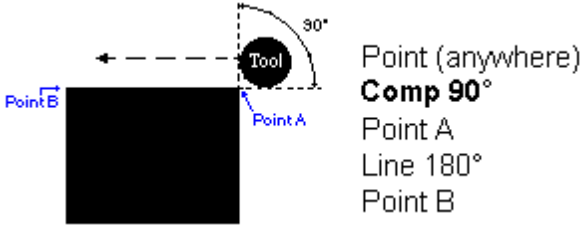
- ◆ The Blend Radius is always dimensioned as a center (type=1)
- ◆ The Blend Radius will always be negative (-) because the tool is on the inside.
- ◆ The Blend Radius should always be tangent to the part (touches in 1 place).
- ◆ The Blend Radius should be at least twice the diameter of the tool to avoid a gouge.
- ◆ The COMP angle determines exactly where the tool begins cutting the Blend Radius.
 - Mentally place the compass center on the center of the tool - not the middle of the Blend Radius.
 - The COMP angle is the tangent direction of the first cut, OR
 - The COMP angle is the direction the cutter will be facing as it starts cutting the Blend-on Radius.

Tip 3: **Examples of COMP starting on a POINT.** (Any POINT programmed before COMP has no effect on how the tool starts on the contour. Any POINT programmed after UNCOMP has no effect on how the tool stops on the contour.)

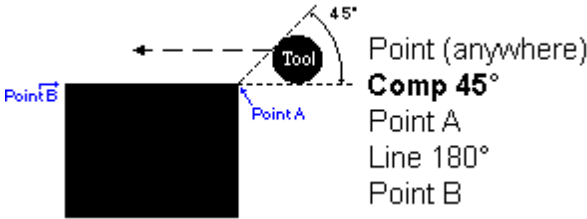
1. The tool is tangent to the first cut (Line 180°) and touches the first Point (A).



2. The tool is tangent to 90° and the first cut (Line 180°), and it is ready to touch the first Point (A).



3. The tool is tangent to 45° and the first cut (Line 180°), and it is ready to touch the first Point (A).





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.

Programming Tips

Tip 1: When programming a Blend-off Radius, the UNCOMP angle determines where the tool will stop.

Tip 2: Program either a POINT or RADIUS before UNCOMP.

Tip 3: **Blend-on / Blend-off Radii.** The tool can automatically “blend” off the part by cutting further around the first or final RADIUS.

- The Blend Radius is always dimensioned as a center (type=1)
- The Blend Radius is always negative (-) because the tool is on the inside.
- The Blend Radius should always be tangent to the part (touches in 1 place).
- The Blend Radius should be at least twice the diameter of the tool to avoid a gouge.
- The UNCOMP angle is the angle where you would like the tool to stop.

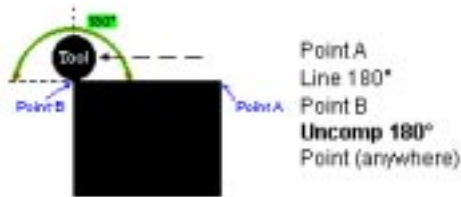
Mentally place the compass center on the center of the tool -not the middle of the Blend Radius.

The UNCOMP angle is the tangent direction of the last cut, *OR*

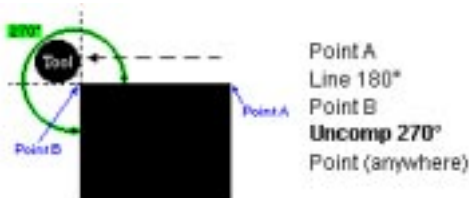
The UNCOMP angle is the direction the cutter will be facing as it stops cutting the Blend-off Radius.

Tip 4: **Examples of UNCOMP ending on a POINT.** (Any POINT programmed after UNCOMP has no effect on how the tool stops on the contour.)

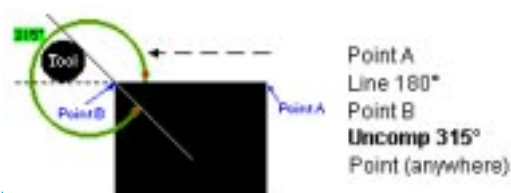
1. The tool remains tangent to the last cut (Line 180°) and touches the last Point (B).



2. The tool remains tangent to the last cut and is ready to cut in 270° crossing the last Point (B).



3. The tool remains tangent to the last cut and is ready to cut in 315° crossing the last Point (B).





6) TOOL

Format: TOOL _ dia_ flutes_ type_ rad_

WHICH TOOL NUMBER WOULD YOU LIKE?

Enter the number of the tool pocket that you would like to use.

Note 1: 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. For example: if you want H04 instead of H01 for tool 1, then enter 1.04.

Note 2: If you just want a program stop (M00) right now instead of a complete tool change, then enter a zero instead of a tool number.

WHAT IS THE DIAMETER OF THIS TOOL?

Enter the true full diameter of the tool you are about to use.

Note 1: To swap from “part edge” to “tool center” contouring (or vice versa), enter a minus tool diameter.

HOW MANY FLUTES (teeth) DOES THIS CUTTER HAVE?

In order for G-ZERO to calculate the correct feedrate for you, it may need to convert a chipload into a feedrate. Enter the number of teeth for this cutter. For a drill, your answer will almost always be 2.

WHAT TYPE OF CUTTER IS THIS?

Pick a tool type from the window on your screen.

This list comes from the file 6.4 located in the WinDat subdirectory within the Mill directory.

Use Windows Notepad to modify this file when necessary.



WHAT IS THE BULL NOSE RADIUS? (usually 0 or half dia)

When machining 3-axis shapes you will generally use a ball endmill. Of course, 2 or 2-1/2 axis machining will require a sharp cornered endmill. For the best of both worlds, you may wish to use a bull-nosed endmill — one with radiused corners. If so, then enter the corner radius of the cutter here. If your cutter is a plain sharp-cornered endmill, then enter a 0. And if this is a ball endmill, then enter the diameter / 2.

Programming Tips

- Tip 1: G-ZERO is programmed like a CNC; pick a TOOL, define the Z information with a MILL or DRILL command, then define the contour / locations.
- Tip 2: TOOL cancels the current DRILL, MILL, or STOCK and retracts the spindle to the tool-change position for the new tool.
- Tip 3: To program non-machine-readable contours such as macros, clamps, and fixtures, program these part definitions before the very first TOOL in that source program.



7) MILL

Format: MILL zrapid_zcut_passes_zret_zf_xyf_

RAPID TO WHAT Z DIMENSION BEFORE KICKING TO FEED MODE?

The MILL command will automatically wait until an XY location is given before moving the cutter at rapid to that location. Enter the Z value to rapid to. Since the top of the part makes a convenient Z0, you will generally enter a value like .1 now.

Note 1: Your answer **MUST** be .1 above part surface when using passes! Example: if you are cutting a secondary pocket and the first pocket is -1, the rapid must be -.9

FEED TO WHAT Z DIMENSION? (usually minus)

Once the tool has rapided to the Z value you just gave, it will feed into the part to this Z dimension. For a .5 deep cut, you would enter -.5 here.

HOW MANY PASSES TO FULL DEPTH? (usually just 1)

When cutting harder metals, you may not want to take it all in one pass. G-ZERO can automatically make repetitive passes at deeper and deeper Z levels. G-ZERO automatically divides the total depth of the cut (zcut) equally by the number of passes. Enter 1 to disable this feature.

Your Z rapid dimension **MUST** be .1 above part surface if you use this feature! Everything from this command to the next Z command (MILL, DRILL, ZMOVE, FOURTHAXIS, TOOL) will be stored in a CNC subroutine.

Note 1: If you are roughing a pocket and want a light finish pass, program an integer and a decimal amount (ex: 3.005) that tells G-ZERO to generate the defined number of passes (3) leaving the decimal amount of zstock (.005). Example: to make 4 passes and then a .003 clean-up pass, enter 4.003. G-ZERO makes an automatic final pass to remove the specified zstock.

Note 2: When roughing a pocket, a negative pass tells G-ZERO to start from the last point in the Mill cycle and ramp to the first point in the Mill cycle. Example: answering -3 tells Mill to ramp down into each one of the 3 levels of the cut.

Note 3: To avoid lifting the tool to the Z rapid dimension between each pass, enter your answer as negative (minus).

RETRACT TO WHAT Z DIMENSION? (usually = zrap)

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 Z dimension to retract to at the end of the cut you are about to make. If you have to jump over clamps or from one cavity to another, give the clearance dimension.

PLUNGE INTO THE PART AT WHAT Z FEEDRATE?

Enter the Z feedrate in inches per minute. The answer you see in default is based on the part material and your tool type. The calculated feedrate is displayed on the status bar. For example: Computer ZF=136.1

Note 1: If the number G-ZERO offers seems too fast, then check the value at the lower left corner of the G-ZERO Mill window (in the status bar); the spindle speed may be too fast. If so, then 23) INSERT a max rpm just before your first tool. For example, %MAXS6000

Note 2: If you would rather not “lock in” the feedrate, then you may give a feed factor instead. Just enter it as a minus number. Example: -2 is 2 times the rate in the feed & speed chart.

MOVE IN XY AT WHAT FEEDRATE?

Enter the XY feedrate in inches per minute. The answer you see in default is based on the part material and your tool and the depth of cut per pass.

The calculated feedrate is displayed in the status bar. For example: **Computer XYF=43.5 x ff1.29 =56.1**
 The XYF is the feedrate pulled from the Speed & Feed chart. The feedrate is based upon the Material type and Tool diameter, # of flutes, and Tool type.

If you want the actual feedrate from the Speed & Feed chart, type in the XYF value or click the left mouse button on the XYF=value. The recommended feedrate is the base feedrate x feed factor (ff).

ff<1 gives you a slower feedrate

ff=1 gives you the base feedrate

ff>1 gives you a faster feedrate

Note 1: To have G-ZERO recalculate the feedrate if there is a change to the TOOL or MILL commands, or a change of Material type, then type this answer as -1.

Note 2: If the number G-ZERO offers seems too fast, then check the value at the lower left corner of the G-ZERO Mill window (in the status bar); the spindle speed may be too fast. If so, then 23) INSERT a max rpm just before your first tool. For example, %MAXS6000

Note 3: If you would rather not “lock in” the feedrate, then you may give a feed factor instead. Just enter it as a minus number.

Programming Tips

Tip 1: General Mill Programming Formats:

TOOL	TOOL	TOOL
MILL	STOCK	MILL
COMP	MILL	ROUGH
(contour)	COMP	COMP
UNCOMP	(contour)	(pocket)
	UNCOMP	UNCOMP
	STOCK	ROUGH
	REPEAT	

Tip 2: MILL brings the spindle down to the next defined location. At the end of the Mill cycle, the spindle will rapid up to the retract plane.

Tip 3: If you do not program a MILL command between locations cut with the same tool, the tool will NOT lift out of the material.

Tip 4: The spindle stays DOWN at the cutting plane until one of three commands is programmed:

1. TOOL makes the spindle retract to the tool-change position.
2. MILL makes the spindle move to the retract plane and then come down again at the next location.
3. ZMOVE makes the spindle move up or down as commanded.

Tip 5: STOCK is the only command that can be programmed between TOOL and MILL.

Tip 6: You can select your feedrate in either 3 ways:

- Use your last answer (default) that is located at the end of the prompt. Just press <Enter>.
- Type in a new feedrate and press <Enter>.
- Click the mouse on the base feedrate or recommended feedrate displayed on the status bar (lower left corner of your G-ZERO window). The base feedrate comes from the modifiable Feed & Speed charts. The ff is a multiplier based on the depth of cut.

Tip 7: Ramping into a pocket:

<u>Circular</u>	<u>Straight</u>	<u>Plunge</u>
MILL	MILL	MILL
ZMOVE	POINT	(rough cycle)
ROUND	ZMOVE	
(rough cycle)	(rough cycle)	

When the tool ramps into a pocket (using ZMOVE), the depth of cut (zcut) is equal to zero (zcut=0). The tool will feed from the zrapid plane to the surface of the part. The command ZMOVE will control the depth of cut.



8) DRILL

Format: DRILL g_ zrap_ zcut_ pecks_ tip_ zret_ f_

WHAT G-CODE WOULD YOU LIKE FOR THIS DRILL CYCLE?

Most machining centers have built-in canned cycles for drilling various types and depths of holes.

G-ZERO shows you a list of the standard drill cycles. You can either key in the cycle number you want, or select it from the list.

Note 1: Some drill G-codes recognized by G-ZERO:

G81: simple drill/countersink (answer remaining DRILL prompts as asked)

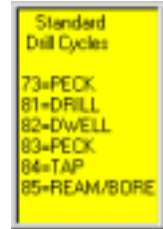
G82: simple drill with dwell (enter the dwell-time in seconds instead of # of pecks)

G83: peck drill for deeper holes (can enter depth of peck instead of # of pecks)

G84: tapping (enter the TPI instead of # of pecks)

G85: boring/reaming (answer remaining DRILL prompts as asked)

Note 2: For high speed tapping, you may wish to use your machine's G85 (boring) cycle. To have G-ZERO calculate the correct feed, enter -85.



BEGIN THE DRILLING OPERATION AT WHAT Z DIMENSION?

The DRILL command will automatically wait until an XY location is given before moving the cutter at rapid to that location. Enter the Z value to rapid to. If your last command was a TOOL command, then your tool will rapid to the next XY then rapid DOWN to the Z value you enter here. If you were last drilling, then your tool will rapid UP to this Z value before rapid to the next XY.

DRILL TO WHAT FINAL Z DIMENSION? (usually minus)

Once the tool has rapid to the Z value you just gave, it will begin performing the canned cycle. Enter the final Z depth you would like to drill, peck, or tap to.

Note 1: If this tool is a center drill or countersink, then you can have G-ZERO calculate this final Z depth for you. Enter now the Z dimension of the TOP of the countersink. Your answer will often be 0.

HOW MANY PECKS / TPI / DEGREES / SECONDS?

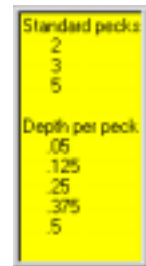
Give the total number of pecks you would like the tool to make as it reaches the final depth. If the cycle you selected does not peck, then just enter a 1.

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

Note 2: If this is a tapping operation, then enter the threads per inch here. G-ZERO will calculate the correct feedrate.

Note 3: If this is a G82 type dwell drill cycle, then enter the number of seconds of dwell here.

Note 4: If you are using a countersink and this cycle does not peck, 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.



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

Often, when drilling through holes, you may wish to have a drill go a bit deeper than usual to break through the bottom of your part. If you enter 1=yes, then G-ZERO will add 1/3 of the current tool diameter to the final Z depth. For example: if you are drilling with a 1 inch drill, and you give a final Z of -2 inches, then your tape will show a final Z of -2.3333 inches.

Enter 0=no if your final Z depth should remain unchanged.

Note 1: To set the amount of extra depth yourself, enter the depth here as a minus number.

Note 2: For drills that have non-standard tip angles, just enter the angle instead of a 1 here.

Note 3: 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.

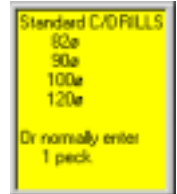
RETRACT TO WHAT Z DIMENSION BETWEEN HOLES? (usually = zrap)

Occasionally, you may want the tool to rapid up to a higher plane between holes. If you are jumping clamps or drilling into separate cavities, this will be the case. Enter the Z value to retract back to. If you do not want this feature, then give the SAME value you gave for the Z rapid dimension.

DRILL AT WHAT FEEDRATE?

Enter the Z feedrate in inches per minute. The default value given is calculated based on the material and tool used.

Note 1: You can ignore this question when tapping.



Programming Tips

Tip 1: DRILL is always programmed after TOOL and before the locations to be drilled (POINT, BOLTCRC, GRID)

Tip 2: DRILL automatically retracts the TOOL to the z rapid position between each location.

Tip 3: Feedrate is based upon the material selected (Material type) and Tool. You can select in either 3 ways:

- Use your last answer (default) that is located at the end of the prompt. Just press <Enter>.
- Type in a new feedrate and press <Enter>.
- Click the mouse on the base feedrate or recommended feedrate displayed on the status bar (lower left corner of your G-ZERO window). The base feedrate comes from the modifiable Feed & Speed charts. The ff is a multiplier based on the depth of cut.

Tip 4: The tool automatically comes up to the z retract position and down to the zrapid position (usually the same) at each location while in DRILL mode.

Tip 5: To counterbore, use MILL with ROUND commands instead of DRILL and repeated POINTS.

Tip 6: Countersink:

In the DRILL command, there are four “flags” that tell G-ZERO that this is a countersink and to calculate the depth automatically:

- G82 drilling cycle
- zcut: enter top surface of countersink
- # of pecks: enter countersink angle
- thru: enter countersink diameter



9) RECT

Format: RECT xmin_ xmax_ ymin_ ymax_ thru_

WHAT IS THE X OF THE LEFT-MOST EDGE OF THE RECTANGLE?

A rectangle can be either a pocket or a window in your part. Enter the X dimension of the left wall of your rectangle. Remember, your answer might be minus.

Note 1: If you would rather conventional-cut instead of the normal climb-cut, then swap xmin for xmax. In other words, give the right-most X value here.

WHAT IS THE X OF THE RIGHT-MOST EDGE?

Enter the X value of the right wall of your rectangle. This is the maximum X dimension.

WHAT IS THE Y OF THE NEAR EDGE OF THE RECTANGLE?

Enter the Y value of the near wall of your rectangle. This is the minimum Y dimension.

Note 1: By swapping the ymin with the ymax, you can force the tool to start in a different corner. You may also be required to swap the xmin and xmax to obtain the desired results.

WHAT IS THE Y OF THE FAR EDGE?

Enter the Y value of the far wall of your rectangle. This is the maximum Y dimension.

WILL THE TOOL BE CUTTING ALL THE WAY THRU YOUR PART?

(1=yes,0=no)

If you are milling a rectangular window, then enter 1 for “Yes” — the cutter will make one pass around the edge.

If you are milling a pocket, then enter 0 for “No” — the cutter will spiral out from the center of the pocket, stepping-over 80% of the tool on each pass.

Note 1: If you are using this cycle to FACE the part, then enter 2 here. This will bring the cutter all the way to the edge.

Note 2: Cutter comp engages automatically, so to override the default D number, give YOUR D number after the decimal. (Example: for D52 you could enter 1.52 or 0.52)

Programming Tips

Tip 1: RECT has cutter compensation built-in and feeds from the center and spirals outward.

Tip 2: The corner radii of RECT are always equal to the radius of the current tool.



10) ROUND

Function: ROUND dia_ x_ y_ thru_

WHAT DIAMETER ROUND ARE YOU MILLING? (- for inside)

For an inside case such as a round pocket, enter this number as a minus diameter.

Note 1: If you are on the inside, and your tool is the same diameter as your hole, then use DRILL instead of ROUND.

WHERE IS THE CENTER IN X?

Enter the X dimension of the center of the ROUND. Your answer may be negative.

WHERE IS THE Y CENTER?

Enter the Y dimension of the center of the ROUND. Your answer may be negative.

WILL THE TOOL BE CUTTING ALL THE WAY THRU YOUR PART?

(1=yes,0=no)

Enter 1 and the cutter will start at the center and make a single spiral pass to the edge. Enter 0 and the cutter will make several overlapping spirals out from the center.

Note 1: To avoid feeding all the way from the center of the ROUND, enter the angle to start at on the edge. (Example: to cut a full circle starting at the top of the circle, enter 90. Conventional cut by entering -270).

Note 2: To conventional-cut instead of the normal climb-cut, enter a -1. (or -.0001)

Note 3: To add a cleanup pass, enter the number of loops in the thousandths place. (Example: 1.002 will loop twice)

Note 4: When using your CNC canned cycle, you may wish to set the D offset register. If so, then give the D number after the decimal. (Example: for D52, enter 1.52)

Note 5: For helixing, just give the start angle. Remember, there must be a ZMOVE with ramping just before this ROUND.

Programming Tips

Tip 1: ROUND has cutter compensation built-in and usually starts feeding from the center and works its way outward.

Tip 2: A negative (-) radius value cleans the material out of the inside of the ROUND.

Tip 3: A positive (+) radius cuts around the ROUND (standing boss).



11) BOLTCRC

Function: BOLTCRC dia_x_y_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.

WHERE IS THE CENTER OF THE BOLT-CIRCLE IN X?

Enter the X dimension of the bolt-circle center.

WHERE IS THE CENTER IN Y?

Enter the Y dimension of the bolt-circle center.

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.



12) GRID

Function: GRID num_ xnum_ x_ xstp_ y_ ystp_

HOW MANY HOLES WOULD YOU LIKE?

Enter the total number of holes to be drilled. Example: In figure 1, your answer will be 8.

Note 1: If you enter this number as a minus number, G-ZERO will only drill the holes around the outside of the grid. This is called a FRAME. You must still calculate this number as if the middle holes were included.

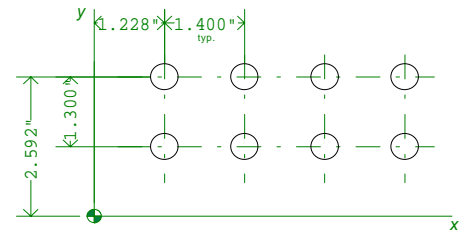


Figure 1

HOW MANY HOLES IN X?

Enter the number of holes to be drilled in a row. Example: In figure 1, your answer will be 4.

Note 1: For a single vertical row, your answer would be 1.

Note 2: If you enter this number as a minus number, G-ZERO will step along the Y-axis first instead of the X-axis.

WHERE IS THE FIRST HOLE IN X?

Enter the X dimension of the first hole to be drilled. Your answer may be minus. Example: In figure 1, your answer will be 1.228.

HOW FAR APART ARE THE HOLES IN X?

Enter the INCREMENTAL distance between each row in X.

If the rest of the holes in the grid are off to the left, then enter this as a minus number.

If you only have 1 row in X, then just ignore this question.

Example: In figure 1, your answer will be 1.4.

WHERE IS THE FIRST HOLE IN Y?

Enter the Y dimension of the first hole to be drilled. Your answer may be negative. Example: In figure 1, your answer will be 2.592.

HOW FAR APART ARE THE HOLES IN Y? (usually minus)

Enter the INCREMENTAL distance between each row in Y.

If the rest of the holes in the grid are off to the Y minus direction, then enter this as a negative number.

If you have only 1 row in Y, then just ignore this question, or answer 0 (zero) for NO stepover.

Example: In figure 1, your answer will be -1.3.

Programming Tips

Tip 1: The GRID command can also drill a single line of evenly-spaced holes.

Tip 2: The Y stepover is negative if the first dimensioned hole of a GRID is the uppermost point.

Tip 3: The Y stepover is a 0 (zero) when there is no stepover in Y.

Tip 4: The X stepover is negative if the first dimensioned hole of a GRID is the right-most point.

Tip 5: The X stepover is 0 (zero) when there is no stepover in X.



13) COPY

Function: COPY x_ y_ z_ exclude_

WHAT IS THE PRIMARY X DIMENSION OF THE FIRST COPY?

Enter the X dimension of your point of reference original.

WHAT IS THE PRIMARY Y DIMENSION?

Enter the Y dimension of your point of reference on the original.

WHAT IS THE PRIMARY Z?

Enter the Z dimension of your point of reference on the original.

WOULD YOU LIKE THE ORIGINAL EXCLUDED? (1=yes,0=no)

If you have defined your original at some convenient location (like X0 Y0), then you may wish to have it deleted from your G-code. If so, answer 1 for yes. In most cases you will want the original, so answer 0 for no.

Note 1: If you plan to use the special rotation feature (COPY with BOLTCIRCLE), then enter 360 here to let the post know.

Programming Tips

Tip 1: Sample Usage 1:

COPY... (defines point of reference for geometry between the 2 COPY commands)

MILL...
RECT... } (geometry to be copied)

COPY... (program this COPY with same answers as first COPY command)

POINT... (repeats the MILL and RECT commands with new base point defined here)

GRID... (repeats again the MILL and RECT commands with new base points)

BOLTCRC... (repeats again the MILL and RECT commands with new base points)

Tip 2: Sample Usage 2:

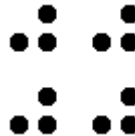
DRILL


COPY x0 y0 z0 exclude1

POINT x0 y0
POINT x2 y0
POINT x2 y1 } (geometry to be copied)

COPY x0 y0 z0 exclude1

GRID num4 xnum2 x-4 xstp4 y-3 ystp5



Tip 3: If G-ZERO doesn't seem to allow you to enter a GRID or BOLTCRC command, press the O key (to override) before you press the  key.

Tip 4: COPY's "repeat" mode is cancelled by: TOOL, MILL, DRILL, COPY, or End of Program.

Tip 5: To make several identical parts, use the MULTIPLY command. MULTIPLY and COPY cannot be used together.



14) STOCK

Function: STOCK xystk_ zstk_

LEAVE HOW MUCH MATERIAL ON PART WALLS FOR CLEANUP?

Enter the amount you would like the tool to stay away from the finished walls of your part. This only leaves material on the sides.

LEAVE HOW MUCH MATERIAL ON PART FLOOR?

If you are cutting with the bottom of your endmill, then you may wish to leave some material on the floor of your part for later cleanup. If so, then enter the amount to stay away.

Programming Tips

- Tip 1: STOCK is usually programmed after TOOL and before MILL.
- Tip 2: STOCK must be programmed before describing the contour's cutting path.
- Tip 3: STOCK is usually programmed for OD contours (use ROUGH for pockets). It may also be used for pockets when another tool is to be used to finish.
- Tip 4: STOCK remains in effect and highlighted on the menu until either of 2 commands are programmed:
 - Another STOCK command (xystk=0, and zstk=0)
 - TOOL command (automatically resets all STOCK to zero)



15) REPEAT

Function: 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 reuse.

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, MILL, DRILL, or ZMOVE source lines when using REPEAT to reverse cut a contour.

Note 2: To mirror your cutter path in X, make this a negative (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. Example you could repeat FROM 12 THRU 20.

If you only want to repeat one single line, then give the same answer you gave for the previous question. For example, FROM 14 THRU 14.

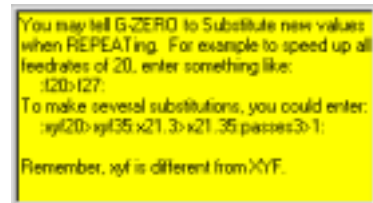
Note 1: To reverse the direction your cutter takes along a surface, you may use REPEAT backwards. For example, FROM 14 THRU 12. Be sure not to repeat COMP, UNCOMP, ROTATE, MILL, DRILL, or ZMOVE source lines when using REPEAT to reverse cut a contour.

Note 2: To mirror your cutter path in Y, make this a negative (minus) number.

Please describe this Repeat:

You can enter a few words here to describe the REPEAT procedure. This information will be treated as a comment and will not disrupt your G-codes. Example: POCKET B.

Another thing you can do in this section is to tell G-ZERO to replace one or more values with new values when the REPEAT is taking effect. For example, if you have included feedrates f20 in the original lines, you can change those feedrates to another value such as f27 in the repeated source code lines. This is done by enclosing the original and new values between “colons” and separated by a “greater than” sign: **:f20>f27:**



Programming Tips

Tip 1: Example of how to Mirror in X:

```

2  MAT'L      xmin-4 xmax4 ymin-4ymax4 thk.5 typ0=ALUMALOY
3  TOOL 1     dia.25 flutes2 type0=HSS MILL rpm5000 *** POCKET
4  MILL      zrapid.1 zcut-.25 passes1 zret.1zf6.1 xyf12.2
5  ROUGH     xtk.01 stp60 angle180 cleanup-1
6  COMP      angle180 cl/con1 lookahead0
7  RADIUS    -.5 type1 x.75 y2
8  LINE      angle270
9  RADIUS    -.5 type2 x.25 y.25 (xc.75 yc.75)
10 LINE      angle0
11 RADIUS    -.5 type2 x3 y.25 (xc2.5 yc.75)
12 LINE      angle90
13 RADIUS    -.5 type2 x3 y1.5 (xc2.5 yc.9999999)
14 RADIUS    .125 type2 x1.5 y1.5 (xc1.625 yc1.625)
15 RADIUS    -.25 type2 x1.5 y3.5 (xc1.25 yc3.25)
16 RADIUS    -.25 type2 x.25 y3.5 (xc.5 yc3.25)
17 RADIUS    -.5 type1 x.75 y1.95

```

```
18 UNCOMP angle0
19 ROUGH xtk.01 stp60 angle180 cleanup-1
20 MILL zrapid.1 zcut-.25 passes1 zret.1zf6.1 xyf12.2
21 ROUGH xtk.01 stp60 angle0 cleanup-1
22 COMP angle0 cl/con1 lookahead0
23 REPEAT from-17 thru7
24 UNCOMP angle180
21 ROUGH xtk.01 stp60 angle0 cleanup-1
```

Tip 2: Repeating REPEAT commands (nesting) makes a shorter Source program, but it can be more difficult to read and understand.



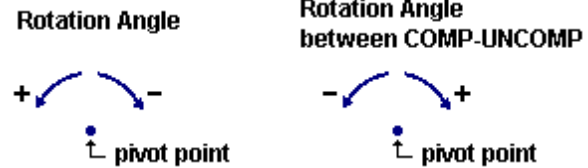
16) ROTATE

Function: ROTATE angle_ xpiv_ ypiv_

ROTATE HOW MANY DEGREES?

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

Rotation angle is positive if the rotation is counter-clockwise is positive, and negative if it is clockwise.



However, if you are using ROTATE between COMP and UNCOMP, the sign of the rotation angle is opposite: positive if rotation is clockwise, and negative if rotation is counterclockwise.

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

Note 2: If you know the angle in degrees, minutes, and seconds, then you may convert by using the quote key. For example, if you want to enter 22° 30' 15" in decimal degrees (dd.mmss), type 22.3015" and G-ZERO will convert it 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. The pivot point becomes a temporary origin. Typically, the pivot point is the obvious point that all the angled dimensions are referenced from.

Note 1: If you are just translating and not rotating, then give the amount that you would like to displace in X. The most common use of translation is to shift an identical feature to one or more locations on the part.

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

This will generally be the Y dimension of the point about which some angled dimensions are given. Quite often, your answer will be zero.

Note 1: If you are just translating and not rotating, then give the amount that you would like to displace in Y. The most common use of translation is to shift an identical feature to one or more locations on the part.

Cancel ROTATE

There are 3 ways to cancel the effect of ROTATE:

- ◆ Program a new TOOL command.
- ◆ Program a new ROTATE and set all answers equal to zero.
Example: ROTATE angle30 x5 y-1.987
 (contour)
 ROTATE angle0 x0 y0
- ◆ Program another ROTATE command which cancels the current ROTATE and activates the new ROTATE.
Example: ROTATE angle30 x5 y-1.987
 (contour)
 ROTATE angle-19.6 x5.2 y0

In this example, the angle -19.6 is an absolute value (NOT 19.6° from 30°).

Types

There are 2 types of ROTATE depending on whether you have already programmed COMP or not.

ROTATE before COMP: ROTATE always pivots around the origin (x0;y0), then translates by xpiv and ypiv amounts.

```
1    MAT'L
2    COMP
    ....
25   UNCOMP
26   TOOL
27   MILL
28   ROTATE
29   REPEAT (COMP thru UNCOMP)
30   MILL
31   ROTATE
32   REPEAT (COMP thru UNCOMP)
```

ROTATE after COMP: After programming COMP then ROTATE, program all following dimensions from the new axes orientation (0, 90, 180, 270). The dimensions are automatically rotated around the specified pivot point to their proper placement.

```
1    MAT'L
2    TOOL
3    MILL
4    COMP
    ....
10   ROTATE
    ....
20   ROTATE angle0 xpiv0 ypiv0
    ....
30   UNCOMP
```

Troubleshoot

When rotating an entire feature outside of COMP, there may be part geometries that include RADIUS type0 (neither) or RADIUS type2 (corner). If a contour includes these radii, then use the 25) MOVE command to move the contour before the first TOOL of the program. Then, use ROTATE and REPEAT to cut the contour in its correct position.

Note: Dimensions before the first TOOL do not generate any G-code.

```
Example:  COMP
          (rotated contour programmed before first tool)
          UNCOMP
          TOOL
          MILL
          ROTATE
          REPEAT (COMP-UNCOMP)
          MILL
          ROTATE
          REPEAT (COMP-UNCOMP)
          ROTATE angle0 x0 y0
```

Programming Tips

Tip 1: Any command following ROTATE (POINT, RADIUS, LINE, RECT, etc.) will be affected by the ROTATE until it's effect is canceled.

Tip 2: Any ROTATE following another ROTATE first cancels the previous ROTATE (always absolute).

```
Example:  ROTATE angle30
          ROTATE angle25  (rotate angle is 25°, NOT 55°)
```

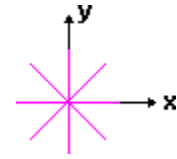
Tip 3: If you program ROTATE after COMP, turn ROTATE off before UNCOMP

Example: COMP

```
.....  
ROTATE  
.....  
ROTATE angle0 xpiv0 ypiv0  
.....  
UNCOMP
```

Tip 4: If confused about what ROTATE angle to program, give it your best guess. Keep programming. If incorrect, just <Esc> to back ups and correct.

Tip 5: The ROTATE angle should be less than 90°. Imagine placing rotated coordinates on top of the original coordinates to assist you determine the angle of rotation.



Tip 6: If you program ROTATE before COMP, turn ROTATE off after programming UNCOMP.

Example: ROTATE

COMP

```
.....  
UNCOMP  
ROTATE angle0 xpiv0 ypiv0
```



17) MAT'L

Function: MAT'L xmin_ xmax_ ymin_ ymax_ thk_ type_

WHAT IS THE LEFT-MOST X DIMENSION OF YOUR MATERIAL? (often -)

In order for G-ZERO to correctly display your part, you must describe the length and width of your material. Start by selecting a zero point and then entering the minimum X dimension on your blueprint. If you are unsure, you may calculate by using the +, -, *, and / keys to add, subtract, multiply, and divide, right on the computer, or just give your best guess.

WHAT IS THE RIGHT-MOST X DIMENSION?

Enter the maximum X value on your part. If this dimension is difficult to figure, then just give your best guess.

WHAT IS THE NEAREST Y DIMENSION OF YOUR MATERIAL? (often -)

Enter the minimum Y value on your part. If this dimension is difficult to figure, then just give your best guess.

WHAT IS THE FURTHEST Y DIMENSION?

Enter the maximum Y value on your part. If this dimension is difficult to figure, then just give your best guess.

WHAT IS THE THICKNESS OF YOUR MATERIAL?

Enter the depth of the part. If your part is round, and zero is in the middle, then enter your answer as a negative number.

WHAT IS THE PART MADE OF?

Pick a material type from the window currently on your screen. If there is no window, then you must create it by exiting to DOS and typing WS DAT\17.6 then enter several materials:

```
0=ALUMALOY
1=303
2=304
3=4140
4=1018 etc.
```

These names must match exactly the names of the .S files.

Programming Tips

- Tip 1: MAT'L tells G-ZERO the "general" size of the part: left most X, right most X, nearest Y, furthest Y, thickness and material type.
- Tip 2: MAT'L types can be customized to accurately calculated feeds and speeds for each tool type.
- Tip 3: Once you program MAT'L and your first TOOL, watch the SFM (s) and RPM (S) change (lower left corner) of your Source screen as you program.
- Tip 4: Select F1 at any time to see the MAT'L boundaries again or to reset the scale after zooming.
- Tip 5: If you are programming multiple parts, define the MAT'L command for the first part ONLY.



19) REDRAW

Function: REDRAW from_ thru_ with tool_ plot_ scale_

WHAT IS THE FIRST LINE YOU WOULD LIKE TO REDRAW?

You may redraw any or all of the lines in your program. Give the line number that you would like to see drawn on screen.

Note 1: To redraw your entire program, try ' or " instead of REDRAW.

LAST LINE NUMBER TO REDRAW?

Many times you may wish to redraw a block of several lines at a time. For example, you could redraw FROM 12 THRU 14.

SHOULD THE DRAWING SHOW TOOL PATH? (1=yes,0=no,2=iso)

If you enter 1 for yes, G-ZERO will calculate your tool diameter offset and plot it (like F3 or F4). If you press 0 for no, then the plot will show the part boundary only (like F2).

Note 1: To plot an isometric view of your part, just enter a 2 here.

Note 2: 3 will display an isometric with tool path (like clicking the left mouse button on Iso).

Note 3: If you enter 4, then a cleaner isometric view will be displayed. (Option 4 works on the screen only)

WOULD YOU LIKE THIS PLOTTED ON YOUR PLOTTER? (0=no,1=yes)

(Optional) If you want to see the picture on the screen, then enter 0. To see it on your plotter, enter 1.

Note 1: If the current part makes extensive use of the 4th axis, then you may wish to redraw only when the 4th is at certain angles. To do this, just tack a decimal point and the desired angle onto this answer. For example, to see only the 90 degree position on the screen, enter 0.090. To plot all 0 degree operations, enter 1.360.

PLOT AT WHAT SCALE?






(Optional) This is FOR PLOTTER USE ONLY. Enter a value that is equal to or smaller than the one G-ZERO is offering.

Programming Tips

Tip 1: Instead of REDRAW, you can redraw the entire Source on the viewport by selecting any of these:

- F2 Part
- F3 Tool
- F4 Slinky
- F5 Isometric
- F6 Solid

Tip 2: From the toolbar on the viewport, you can also have the following redraw options:

-  Left view
-  Front view
-  Top view
-  Isometric view
-  Single step simulation (use space bar to simulate next motion)



20) POST

Since your CNC does not understand G-ZERO, this source program must be processed and converted into a language that 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 with all the Post Processors available in your computer to create a G-code customized to a particular CNC.

Note: If you select NC Verify, G-ZERO will create a file with extension .CL and display it through NC Verify (if available).

This command does not add any line code to the source program.



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. With a mouse, you can click on the number 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. If you wish to change one command to another, then backup and use the special function in 23) INSERT to make your change.

REPLACE WITH?

Enter the value that you would like to put in place of the incorrect value.

G-ZERO searches the Source program for any occurrences of the incorrect value. When one is found, G-ZERO displays the entire line with the wrong value highlighted and asks:

CHANGE (Yes, No, All, Done)? Y

- Y Yes G-ZERO will change the incorrect value and look down the Source program for another occurrence of the same incorrect value.
- N No G-ZERO will not change the incorrect value but will look down the Source program for another occurrence of the same incorrect value.
- D Done G-ZERO will not change the incorrect value and will not look for more occurrences. Note: G-ZERO will display each occurrence throughout the program until you press D for "Done."
- A All G-ZERO will change the incorrect value and EVERY number in the source that also matches the incorrect value — without double-checking.
W A R N I N G: Using A for "All" can be very dangerous.



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.

You will be asked (Y/N/I). Type N (NO) aborts the DELETE command. Type I (INVERT) to restore lines that were previously deleted. To bring back already erased lines, you delete the lines a second time and choose I.

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 and updates the current source program with any changes made. 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.

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: ellipse.b)
3. The Basic system will prompt you several questions. 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.

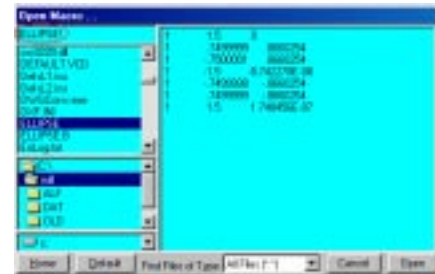
G-ZERO has created a file with the same name as the basic file you selected, but without any file extension. (Example: Ellipse)

4. In the Source program, type # (pound sign) to open the Open Macro window.

In the upper left corner, type the name of the file just created (without extension) followed by ! (exclamation mark) to import new source lines. (Example: ellipse!)

Note: You can also select the new file from the Mill directory (make sure you are displaying "All Files" in the *Find Files of Type* box in the bottom of the window since the new file does not have extension). Place the cursor at the end of the file name and type an exclamation mark to import codes.

5. G-ZERO will automatically display the contents of the file in a yellow window. Press the Enter key to accept and add new lines to your source program.



Note: If you want to create a new file, select and open the NEW.B file (or any Basic file) to start QBasic. From the File menu, select New (to create a new blank Basic file) or Save As (to create a new Basic file with the current information).

Create a new BASIC program

1. Start qbasic.exe

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

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 a parabolic formula:

```

15 OPEN "O", 1, "STUFF"
20 FOR X = -3 TO 3
30 Y = X * X
40 PRINT "X="; X, "Y="; Y
45 PRINT #1, 1; X; Y
50 NEXT

```


Examples

ELLIPSE.B

Because ellipses are so common, your G-ZERO system includes a sample BASIC program to make them. The program is named ELLIPSE.B and it looks like this:

```
5 OPEN "O",1,"ELLIPSE"
7 INPUT "WHAT IS THE MAJOR DIAMETER";MAJOR:MAJOR=MAJOR/2
8 INPUT "WHAT IS THE MINOR 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 360+.0001 STEP STP
20   A=ANG*3.1415926536#/180:X=COS(A)*MAJOR:Y=SIN(A)*MINOR
30   PRINT X;Y
35   PRINT #1,1,X,Y' ;:PRINT #1,USING "##.##### ";X,Y
40 NEXT
50 SYSTEM
```

Note that BASIC programs look a little like G-code. Each line starts with a line number (without the N).

Here's what BASIC program ELLIPSE.B is doing, line-by-line:

Line 5 tells the computer that you would like to create a file called ELLIPSE.

Line 7, the computer asks the question: WHAT IS THE MAJOR DIAMETER? and waits for your answer. To simplify the math, this program converts the diameter into a radius.

Line 8, the computer asks the question: WHAT IS THE MINOR DIAMETER? and waits for your answer. To simplify the math, this program converts the diameter into a radius.

Line 9, the computer asks the question: STEP ANGLE? and waits for your answer. This program will check if it is a valid angle. If it is not valid, it will display "MUST DIVIDE EVENLY INTO 360" and wait for your new answer until a valid angle is given.

Line 10 begins a looping process that continues all the way around a 360° path.

Line 20 converts the angle from degrees to radians, and computes the X and Y values for the ellipse.

The formula for an ellipse is: $x = \cos(\text{angle}) \times \text{majorradius}$
 $y = \sin(\text{angle}) \times \text{minorradius}$.

Line 30 prints the answer on-screen.

Line 35 prints X and Y to the file so it can be grabbed later from inside G-ZERO.

Line 40 works with line 10 to loop over and over until ANG is 360°.

Line 50 tells BASIC to exit and return to DOS.

CONE.B

```
5 OPEN "O", 1, "CONE"
7 INPUT "WHAT IS THE BOTTOM DIAMETER"; DIA
8 INPUT "WHAT IS THE BOTTOM Z VALUE (-) (HEIGHT OF CONE)"; HT: IF HT>0
  THEN HT = -HT
9 INPUT "STEP AMOUNT IN Z"; STP:IF STP < 0 THEN STP = -STP
10 TANangle = ABS(HT) / ABS(DIA / 2)
20 FOR Z = 0 TO HT STEP -STP
35   DIA = ABS(Z) / TANangle * 2
36   PRINT #1, 33, Z, 1
37   PRINT #1, 10, DIA, 0, 0, 90
40 NEXT
50 SYSTEM
```

DRAIN.B

```
10 INPUT "PART DIAMETER"; DIA
20 INPUT "X SPACING OF GRID"; xstp
30 INPUT "Y SPACING "; YSTP
32 INPUT "X CENTER OF NO HIT ZONE"; XC
34 INPUT "Y CENTER IF ZONE"; YC
36 INPUT "DIAMETER OF NO HIT ZONE"; rz: rz = rz / 2
40 OPEN "O", 1, "DRAIN"
50 RAD = DIA / 2: X = -RAD: Y = RAD
60 YSP = -ABS(YSTP): xstp = ABS(xstp): X = X - xstp
70 XNUM = INT(DIA / xstp) + 3: YNUM = INT(DIA / -YSTP) + 3
80 FOR J% = 1 TO YNUM
90 FOR I% = 1 TO XNUM
100 X = X + xstp:
105 R = SQR(X ^ 2 + Y ^ 2): rr = SQR((X - XC) ^ 2 + (Y - YC) ^ 2)
106 IF R < RAD AND rr > rz THEN PRINT X; Y: PRINT #1, 1, X, Y
110 NEXT I%
120 xstp = -xstp: X = X - xstp / 2: Y = Y + YSTP
130 NEXT J%
140 CLOSE
```



29) ROUGH

Format: ROUGH stk_ stp_ angle_ cleanup_

HOW MUCH MATERIAL WOULD YOU LIKE LEFT ON POCKET WALLS?

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
- 3) call up ROUGH again with all the same answers.

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 the part for a cleanup pass.

Note 1: Remember that even if you enter 0 here, some material will still be left on your part.

Note 2: To define islands, just add a MILL command and the contour (or a ROUND), just before the closing ROUGH. Add more islands by adding more MILLS and contours before closing.

STEPOVER HOW FAR PER PASS?

Enter the distance that the tool should step on each pass.

Note 1: If you enter a number larger than 2, then G-ZERO assumes it is a percentage of the tool. (eg.: 80 will step a 1 inch tool .8 inches per pass).

WHAT ANGLE WOULD YOU LIKE THE CUTTER TO ROUGH AT?

Give the angle that you would like the tool to step as it marches across your pocket.

Ex: If you give 0 degrees, then the tool will start at a Point near the left side of the pocket and will gradually step toward the right.

WOULD YOU LIKE A CLEANUP PASS?

Your tool leaves little bumps on the part walls as it zigzags back and forth, so you may want to make a final pass all the way around the perimeter to clean things up. If so, then enter 1 for yes. This saves you the trouble of REPEATing to clean the walls.

Note 1: To avoid feeding right through a wall, you may want to lift the tool. If so, then enter your answer as a negative number. Warning: some machines have trouble engaging cutter comp when you do this.

Programming Tips

- Tip 1: Includes Stock (wall) and an automatic cleanup pass so the STOCK and REPEAT commands (finish cut) are unnecessary (unless you need to change tools for the finish cut).
- Tip 2: You can define the amount of stepover per pass as a percentage of the tool (ex: 80) or as an absolute amount (ex: .8).
- Tip 3: When defining the original ROUGH command, program the first ROUGH command before turning on COMP and program the identical ROUGH command after turning COMP off with UNCOMP.
- Tip 4: The first ROUGH and last ROUGH must be identical. If you don't program the second (identical in every way) ROUGH command, G-ZERO does not calculate a roughing cycle in your post or on-screen!
- Tip 5: G-ZERO automatically roughs the contour given between the first and last ROUGH commands.
- Tip 6: Actual roughing does not occur until the second ROUGH command is programmed.

- Tip 7: The roughing angle is the general direction of material removal, not the back and forth motion of the Tool. Example: a roughing angle 270 means that the tool begins at the top and finishes at the bottom.
- Tip 8: When programming ROUGH, make sure you have your roughing angle meet the beginning of your pocket definition!
- Tip 9: Automatically avoid islands while roughing. Before "closing" your ROUGH, you may program a MILL command and then the island geometry (POINT, COMP, RADIUS or even ROUND commands). Each new island starts with another MILL command.)
- Tip 10: Steps to ROUGH an area or surface:
1. Rough command.
 2. Define contour/surface.
 3. Rough command (program the exact same answers as the first Rough).
- Tip 11: ROUGH includes an automatic clean-up option. By answering YES (1), the tool will, after Roughing, automatically cut the defined contour to finished dimensions. Use this in place of extra programming to generate a cleanup pass (i.e. instead of REPEAT).
- Tip 12: Doublecheck your roughing pattern by redrawing with the tool: F3, F4 or 19) REDRAW. Watch out for roughing angles and cleanup passes that feed right through the part!
- Tip 13: Program the middle pocket before the first TOOL because any locations programmed before the first TOOL command will NOT generate any NC code. After programming the TOOL, use REPEAT to cut the pockets in an efficient left to right order. ROUGH using the center pocket dimensions and finish using an automatic cleanup pass (i.e. cleanup=1).
- Tip 14: When defining the first ROUGH command, cleanup pass=1 will generate the Roughing cycle first and then travel around the defined perimeter for a cleanup pass.



30) SHIFT

Format: SHIFT x_ y_ 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 WOULD YOU LIKE ADDED TO EACH Y DIMENSION?

It is often helpful to have all the Y 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 Y 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.

Programming Tips

Note 1: SHIFT will stay in effect throughout the rest of your program or until you change it or cancel it with another SHIFT.

Note 2: SHIFT affects your post processor only, so it will not interfere with your plot.



31) STATION

Format: STATION #_ x_ y_ z_ b_

WHAT STATION WOULD YOU LIKE TO WORK ON NEXT?

Enter a number 1 or greater. On most CNCs this corresponds to G54 thru G59. If you have more than 6 stations, then G-ZERO automatically uses G52 or G10 or G92 or advanced work offset registers.

To pre-load the stations place Station commands just before the first tool command. Examples:

```
3 STATION #1 x0 y0 z0 b0
4 STATION #2 x10.1 y.02 z.11 b0
5 STATION #3 x21.2 y0 z0 b0
6 TOOL 1 ...
```

WHAT IS THE X OFFSET FOR THIS STATION?

Enter the distance in X from your program zero to this Station. Your answer will typically be 0 for Station #1 and something like X10 for Station #2, etc.

Note 1: If this is Station #0, then this X dimension is the master Station offset. That means the X value will be added to every other Station X. Station #0 shifts all other stations.

WHAT IS THE Y OFFSET FOR THIS STATION?

Enter the distance in Y from your program zero to this Station. Your answer will typically be 0 for Station #1 and something like Y.1 for Station #2, etc.

If this is Station #0, then this Y dimension is the master Station offset. This Y will be added to every other Station.

WHAT IS THE Z OFFSET FOR THIS STATION?

Enter the distance in Z from your program zero to this Station. Your answer will typically be 0 for Station #1 and something like Z.1 for Station #2, etc.

Note: If this is Station #0, then this Z dimension is the master Station offset. This Z will be added to every other Station.

WHAT IS THE 4th AXIS POSITION FOR THIS STATION?

Enter the 4th axis rotation angle for this Station. If your machine's 4th axis is in the wrong position, then when you call Station, G-ZERO automatically rotates the 4th axis for you.

WHAT IS THE X DIMENSION OF THE 4th AXIS CENTER POINT?

WHAT IS THE Z OF THE 4th AXIS CENTER POINT? (usually -)

GIVE AN X VALUE ON THE ANGLED FACE OF THE PART:

GIVE A Z ON THE ANGLED FACE: (usually -)

Programming Tips

Tip 1: G-ZERO's Station Software is IDEAL for:

1. Mills with G54 type work offsets
2. 4th axis work
3. Tombstone work

Tip 2: The G-ZERO CAM System for Mills comes standard with 2 commands that make complex multi-part (and multi-position) setups easier. Choice 36)Multiply asks how many parts are on your machine, and how far apart they are. Everything programmed after Multiply is automatically repeated over and over until all parts are machined. Choice 37)B Axis controls positioning of the machine's 4th axis, making short work of most simple 4-axis work as well as tombstone work.

Tip 3: Optional choice 31) STATION amplifies these 2 commands by allowing them to work in concert, eliminating long, complicated programs. STATION also improves programmer control of work offsets (also called work coordinates).



32) SCALE

Format: SCALE x_ y_ z_

WHAT SCALE WOULD YOU LIKE IN X?

Enter the number that you would like all X dimensions to be multiplied by. (eg.: for a half scale model, enter .5) (eg.: 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 Y? (usually same as x)

Enter the number that you would like all Y dimensions to be multiplied by. (eg.: for a half scale model, enter .5) (eg.: 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 $.5/1.006=.497$)

Note 2: Giving a number different than the X scale will produce undefined results when milling circles.

WHAT SCALE WOULD YOU LIKE IN Z?

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

Note 1: Quite often you will not want scaling in Z. Just enter a 1 to maintain normal scale.



33) Z MOVE

Format: ZMOVE z_ ramp_ f_

MOVE TOOL TO WHAT Z DIMENSION? (usually -)

Enter the Z value that you would like the bottom of the cutter to feed to. If Z0 is at the top of your part, then your answer will probably be negative. To get the cutter to rapid, try using the MILL command.

RAMP TO THE NEXT POINT? (0=no, 1=yes, #=TPI)

If you would like the cutter to immediately feed up or down in Z, then enter 0. To have an endmill feed in all 3 axes at the same time (ramp), then enter 1.

Note 1: To add a Z dimension to the end of a drill XY, enter 1 here.

Note 2: To trigger optional HELICAL ramping, enter the amount of arc (1=full circ, .5=half, etc.), or for more than 1 loop, give the TPI (threads per inch) [must be greater than 1].

Note 3: To helix, the next command must be a ROUND. To reverse ROUND's default direction, enter your answer as a minus #.

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

How fast, in inches per minute (ipm), would you like the tool to feed toward this point? A typical answer would be 10 ipm.

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

Note 2: If your last command was a MILL command, then your tool will automatically rapid to this point.

Note 3: If you would like the tool to STOP rapiding and continue feeding at whatever feedrate is currently active, then enter 99.99. This will leave the feedrate unchanged.

Programming Tips

Tip 1: Use ZMOVE for ramping or z-axis move like jumping clamps or moving the tool up or down.

Tip 2: When ramping, ZMOVE must follow a POINT command and ramps from the POINT to the command following ZMOVE.

Tip 3: Do not use ZMOVE with multiple passes (MILL # of passes=1). To ramp on each pass including the initial pass, program MILL, POINT, ZMOVE, REPEAT (contour). A minus (-) # of passes will ramp into each pass, excluding the initial pass.

Tip 4: When ramping, the MILL depth of cut=0 (surface of the part). The tool will feed from the zrapid plane to the surface of the part. The ZMOVE feeds to total depth of cut.

Tip 5: To use ZMOVE with RECT or ROUND, use the following programming format:

```
MILL
POINT
ZMOVE
RECT or ROUND
```




34) CURFIT

Format: CURFIT x_ y_ x_ y_ x_ y_

WHAT IS THE X DIMENSION OF 1 OF THE 3 POINTS?

Curfit will calculate a radius that goes through 3 points. Enter the X value of one of the points. Try to avoid giving 3 points that are all along a straight line.

WHAT IS THE Y DIMENSION?

Curfit will calculate a radius that goes through 3 points. Enter the Y value of one of the points. Try to avoid giving 3 points that are all along a straight line.

WHAT IS THE X DIMENSION OF 1 OF THE 3 POINTS?

Curfit will calculate a radius that goes through 3 points. Enter the X value of one of the points. Try to avoid giving 3 points that are all along a straight line.

WHAT IS THE Y DIMENSION?

Curfit will calculate a radius that goes through 3 points. Enter the Y value of one of the points. Try to avoid giving 3 points that are all along a straight line.

WHAT IS THE X DIMENSION OF 1 OF THE 3 POINTS?

Curfit will calculate a radius that goes through 3 points. Enter the X value of one of the points. Try to avoid giving 3 points that are all along a straight line.

WHAT IS THE Y DIMENSION?

Curfit will calculate a radius that goes through 3 points. Enter the Y value of one of the points. Try to avoid giving 3 points that are all along a straight line.



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?

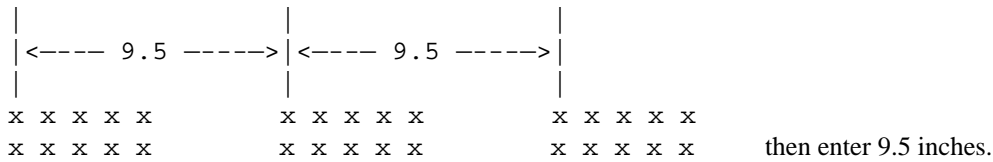
Most outside work will be at nearly 90 degrees. Most inside work at 270. Unlike straight line lettering, this angle controls the CENTER of the letter cell, not the edge.

WHAT WOULD YOU LIKE IT TO SAY?

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

HOW FAR BETWEEN GROUPS? (if more than 1)

If you only have one group of parts, then enter 0. For multiple groups, enter the distance, from group zero to group zero. For example, if your parts are to be laid out like this:



Note 1: If your groups are stepped out in Y instead of in X, then this will be the Y distance. (often minus)

Note 2: If you enter a 0 here, and you have more than 1 group, then G-ZERO assumes that you will set the distances out on your machine using G54 thru G59 work offsets.

WHICH STATION WILL USE THIS MULTIPLY? (0=all)

When you use MULTIPLY and STATION in the same program, you might want 3 parts at 1 station and 4 parts at another. If so, then enter a separate MULTIPLY statement for each STATION declared.

Note 1: To make the same part on several sides of a cube on your horizontal machine, enter a 360 now. Be sure that the number of groups in this MULTIPLY matches the total number of STATION statements.

Note 2: To make the same part on just 2 sides of a cube, GLUE two stations together by entering ##. For example, to use the same subs on station 1 and 3 enter 1.3 here. Be sure you entered 2 GROUPS at the previous question.



37) FORTHAX

Format: FOURTHAX ang_ feed_ g_ zsafe

ROTATE THE B (or A) AXIS TO WHAT DIMENSION?

Enter the angle you would like to have the 4th axis rotated to. Clamp and unclamp codes will automatically be activated.

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

How fast, in inches per minute (ipm), would you like the tool to feed toward this point? A typical answer would be 10 ipm.

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

ACTIVATE WHAT WORK OFFSET?

Enter the work offset number (from 52 thru 59) that you will be using with the 4th axis in this new position.

MOVE TO WHAT SAFE Z DIMENSION BEFORE SPINNING?

To avoid banging your cutter against the corner of your fixture, you will probably want to pull away before the axis turns. Enter the Z dimension to retract to.

Programming Tip

Tip 1: Only the programmed part is displayed on screen. The subroutines and loops for the rest will be part of the output generated by your post processor(s).



38) +VIEW

Format: +VIEW in_ out_ step_ ramp_ blend_

WHAT PATTERN NUMBER CONTAINS THE VIEWS?

+VIEW generally requires 2 input shapes — the first is a pattern containing the top view (stack-axis) of your part, the second is a pattern of the end view (cross-section) of the part. The end view will almost always start at X0Y0 and end somewhere down and to the right of X0Y0.

Note 1: To use a section of your source program instead of a pattern, just give a decimal pattern number in the form: (start line).(end line) (eg.: to input from line 12 to line 15 — 12.015). The maximum line number is 9999. **BE SURE TO SEPARATE EACH SHAPE WITH A COMMENT.**

Note 2: Just 1 input shape for a surface-of-revolution.

Note 3: 3 input shapes (separated by 2 comments) will trigger +VIEW's powerful merging routine. The top view must be a contour made up of 2 major elements (POINTS or RADs) separated by an unknown RAD. 1 or 2 LINES may be in between. Instead of giving 1 end view, give 2.

PUT RESULT INTO WHAT PATTERN NUMBER?

Enter the number of the pattern that you would like to have the surface data stored into for later use. Be sure not to step on a pattern that you are already using in this program. See +TILT section for more help on patterns.

Note 1: For a surface of revolution that zig-zags back and forth, enter this as a minus pattern number.

STEPOVER HOW FAR PER PASS?

As your tool creates the desired surface, it must step over a little between each pass. A large stepover is good for rough cutting, and a small stepover is good for finishing. A typical answer might be .05 or .007.

Note 1: Sometimes you may desire a specific number of passes rather than a stepover. If so, then enter the number of passes.

Note 2: You may prefer to have G-ZERO calculate the appropriate stepover based on the scallop height that is allowable on your part. To trigger scallop calculation, add 99 to your answer. For example, to produce a scallop of .0005 on your part, enter 99.0005.

Note 3: When roughing, you may prefer to have your tool step in even depths per cut. Enter this number as a minus to activate even step cutting.

SHOULD THE TOOL RAMP BETWEEN EACH PASS? (1=yes,0=no)

If your top view contour ends several inches from where it starts, then you will probably want the cutter to lift all the way out of the part, rapid back to the start of the next pass, and then feed back down. On the other hand, many top views start and stop in the same place, in which case it would be silly to hop up and down. Enter a 0 here to hop. Enter a 1 to ramp to the next z level.

Note 1: To create a partial surface-of-revolution, enter the sweep angle here. For example, enter -180 for the bottom half of a surf-of-rev. This will force ramping unless you go back and make your output pattern negative.

IF MORE THAN 2 VIEWS, WOULD YOU LIKE BLENDING? (1=yes,0=no)

+VIEW usually “collides” the end views together. If you would like to “blend” instead, then enter a 1 here. This will cause a whole bunch of points to be generated instead of the circles that you usually get. If you enter a 1 here, then you must have at least 3 views (1 top, 2 end). This will also work with 4 views (1 top, 2 end, 1 top). The second top view allows you to describe how the cutter moves on the last pass.

Note 1: To set the blend stepover, enter the step instead of a 1.

Note 2: If your 2 top views (in a 4-view) are not both climb (or conventional), then enter your answer as a minus number.



39) +TILT

Format: +TILT in_ out_ pitch_ roll_ rot_ x_ y_ z_ stp_

WHAT LINE NUMBERS (or PATTERN) WOULD YOU LIKE TO TILT?

A pattern is like a memory location in a calculator. You may store data in a pattern and then retrieve it later. Patterns are numbered from 1 to 999999 and may be used in any order. Enter the number of the pattern that you would like to have rotated in all three axes.

Note 1: To use a section of your source program instead of a pattern, just give a decimal pattern number in the form: (start line).(end line) (eg.: to input from line 12 to line 15 — 12.015). The maximum line number is 999.

Note 2: To untilt data already tilted, remember that the order of rotation is crucial. You must untilt using up to three tilts. 1 xy, 1 xz, and 1 yz.

PUT RESULT INTO WHAT PATTERN NUMBER?

Enter the number of the pattern that you would like to have the rotated data stored into for later use.

Note 1: If any data is currently stored in this output pattern, then it is automatically cleared. Since patterns are shared by all your programs, this means that you might clobber another program's important pattern. If such interference occurs, then just redraw the other program WITH the tool. Redraw will automatically rebuild your patterns.

TILT HOW MANY DEGREES TOWARD YOU? (minus for away)

Enter the angle that you would like data rotated around the X- axis.

Note 1: If you are using TILT for translation only, then enter 0.

TILT HOW MANY DEGREES TO THE LEFT? (minus to the right)

Enter the angle that you would like data rotated around the Y- axis.

ROTATE HOW MANY DEGREES IN THE STANDARD XY PLANE?

Enter the angle that you would like data rotated around the Z- axis. This is just like choice 16) ROTATE.

HOW FAR IN X WOULD YOU LIKE THE PATTERN TRANSLATED?

This will generally be the X dimension of the point about which some angled dimensions are given.

Note 1: If you are just translating and not rotating, then give the amount that you would like to displace in X.

HOW FAR IN Y?

This will generally be the Y dimension of the point about which some angled dimensions are given.

Note 1: If you are just translating and not rotating, then give the amount that you would like to displace in Y.

HOW FAR IN THE Z AXIS?

This will generally be the Z dimension of the point about which some angled dimensions are given.

Note 1: If you are just translating and not rotating, then give the amount that you would like to displace in Z.

BREAK ARCS INTO WHAT SIZE STEPS?

Most controllers can only handle arcs that are in the X plane, Y plane or Z plane. Arcs that are at some funny angle must be broken into points so the controller can digest them. Enter the largest allowable step between points along an arc.

Note 1: To activate your CNC's special sideways arc feature (usually G18 and G19), just enter a zero here. But remember, then you cannot perform any other operations on this output pattern. For example, you cannot TRIM data off a G18 pattern.

Note 2: To break arcs every so many degrees, just enter the angular spacing as a minus. (eg: to break every 2°, enter -2)

Programming Tips

PROCEDURE

Step 1 Define the XY contour

Step 2 MILL command

Step 3 Select 39) +TILT. This will let you rotate the contour in up to 3 axes as well as translate. Example: Mat1ymax1 thk3 type4=1018

EXAMPLE

```
1 VIEW      xmin-5 xmax5 ymin-1 ymax1
2           This is a simple example of how +TILT is used.
3   TOOL 1  dia.5 flutes2 type=HSS rad.25 *** BALL ENDMILL
4           No mill command yet!
5   COMP    angle0 c1/con1
6   POINT   x-4 y0
7   LINE    angle0
8   RADIUS  .25 type0 (xc-1.224745 yc-.25)
9   RADIUS  -1 type1 x0 y0
10  RADIUS  .25 type0 (xc1.224745 yc-.25)
11  LINE    angle0
12  POINT   x4 y0
13  UNCOMP  angle0
14         Now that the flat shape has been defined,
15         we can begin actual machining.
16  MILL    zrapid.1 zcut0 passes1 zret.1
17  +TILT   in5.013 out1 pitch90 roll0 rot0 x0 y0 z0 stp0
```

HAZARDS!!!

Data already stored in an output pattern is automatically cleared when another pattern with the same pattern number is entered. Since patterns are shared by all Source programs, it is possible to clobber another Source's important pattern. If a pattern in another program has been "clobbered", REDRAW the clobbered program with the tool to automatically rebuild the overwritten pattern.

The order of rotation to untilt data already tilted is CRUCIAL. To reverse the +TILT process, you must use a separate +TILT command for each angle of rotation: 1XY, 1XZ, 1YZ.

NOTE

To tilt already tilted data into a pattern, do not use the (line#).(line#) technique. Just enter the pattern number. The arcs in the first pattern must have been broken (no step-0)!



40) +TRIM

Format: +TRIM in_ out_ bound_ in/out_

WHAT PATTERN NUMBER WOULD YOU LIKE TRIMMED?

This routine takes a pattern and trims all the unwanted data from it. To do this, it discards any points that are outside (or inside) a specific flat xy boundary. If one point in the pattern is inside the boundary and the next is outside, then +TRIM will generate the intersect point automatically. Enter the number of the pattern that you would like to have G-ZERO chop on. **DO NOT** give the boundary pattern yet.

Note 1: See the +TILT section of this manual for more details about patterns.

PUT RESULT INTO WHAT PATTERN NUMBER?

Enter the number of the pattern that you would like to have the trimmed data stored into for later use.

Note 1: See the +TILT section of this manual for more details about patterns.

WHAT PATTERN NUMBER CONTAINS THE BOUNDARY?

Enter the number of the pattern that contains the boundary shape.

Note 1: See the +TILT section of this manual for more detail about patterns.

WOULD YOU LIKE TO STAY: 1) inside OR 2) outside THE BOUNDARY?

Enter a 1 if you wish to keep data that is inside of your boundary. Enter a 2 to keep data that is outside the boundary.

Note 1: Your boundary may be made up of points AND radiuses. To work correctly, +TRIM must split each radius into points. You can set the stepover by adding it to your answer here. For example, enter 1.01 to stay inside and break arcs into .01 segments. The maximum step is .999



41) +PROJ

Format: +PROJ in_ out_ surf_ uponly_

WHAT PATTERN NUMBER WOULD YOU LIKE PROJECTED?

This routine takes a pattern and changes all the Z values so they “ride above” another pattern. If a point in the pattern “falls off the edge”, then the Z value is left unchanged.

DO NOT give the “project upon” pattern yet.

Note 1: If you are confused about patterns, then see +Tilt’s help.

PUT RESULT INTO WHAT PATTERN NUMBER?

Enter the number of the pattern that you would like to have the projected data stored into.

Note 1: If you are confused about patterns, then see +Tilt’s help.

Note 2: If the input pattern contains arcs, then they must be broken into small linear steps. To set step distance, add it to the pattern number. Example: to set step at .05 for output pattern 3, enter 3.05.

WHAT PATTERN NUMBER CONTAINS THE SURFACE?

Enter the number of the pattern that contains the shape to be projected down upon.

Note 1: Surfaces that contain large arcs and circles may produce output that is too grainy. To set the step angle, add it to the pattern number. Example: to step every quarter degree around pattern 3, enter 3.25.

WOULD SHOULD Z BE HANDLED: 1=up only, 0=full project?

Normally enter 0. Enter a 1 if you wish to have the surface “push up through” your data, leaving higher Z values from the input pattern alone.

Note 1: To “warp” your input pattern (add the input pattern Z to the surface Z), just enter a Z offset value (other than 0 or 1).