G-ZERO Mill

Interactive Tutorial

G·ZERO
CAM Software

RAPID OUTPUT

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

Before You Begin

This tutorial was designed using G-ZERO Mill v4.6 installed with the default settings. Newer versions of G-ZERO will, of course, have a look that is slightly different, especially on some settings pages.

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

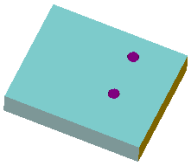
http://www.g-zero.com/DownL/InstallMillStudent.exe

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

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

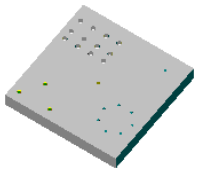
Projects in this tutorial follow a logical progression, so each project assumes you master the concepts presented in previous projects.

Contents



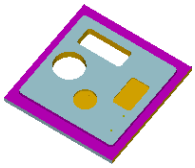
1. Initial Tour 5

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



2. Drill Mania 13

Start a new source file. Define the material. Drill a single hole. Drill a full bolt-circle. Drill random holes. Drill holes arranged in a grid. Use the REPEAT command. Save program and exit G-ZERO Mill.

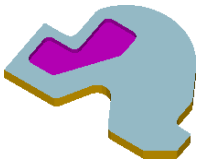


3. Rectangle & Rounds 19

Straight cut using MILL command. Mill rectangles (window and pocket). Mill circles with the ROUND command. Rough cut with the STOCK command. Mill OD step (COMP, UNCOMP, LINE, RADIUS commands)

4. Rectangle & Rounds (revisions) 27

Open an existing source file. Create a copy of a source file using Save As. Print a source file. Editing commands.

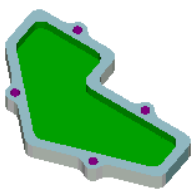


5. Turtle 33

Mill elaborate profile with unknown values. Mill pocket using circular ramping (MILL, ZMOVE, ROUND) and COMP with blend radii.

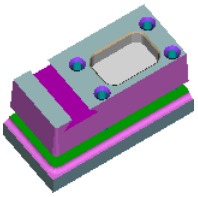
6. Puzzle Plate 41

Multiple part setup using the MULTIPLY command. Face and conventional cutting. Program two unknown radii in a row. Enter an angle in degree/minute/second format. Reverse cutter path.



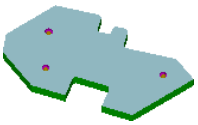
7. C-Gasket 47

Program a source file with data taken from a DXF file. Load CAD Reader and prepare DXF file (window, layer, zoom, origin). Define MAT'L using values from DXF file. Drill holes using CAD Reader Single Pick. Mill pocket and cut profile using CAD Reader Block Pick.



8. ToolType 53

Use of the %#TL, %#TT, and %#TA variables to show accurate modeling of specialized tools in the Solid Modeling environment.

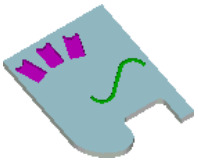


9. Sideways Face 57

Use ROTATE to program geometry with dimension given in a different axis (rotated and shifted coordinates).

10. Odd Pizza 61

Use several ROTATE commands to program geometry with dimensions given in different axes (rotated coordinates). Use of blend-on and blend-off radii.



11. Scroll 65

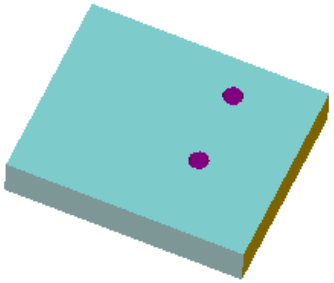
Define a geometry for subsequent multiple use (no cutting performed). Cut on a centerline. Smoothing (smooth points).

12. Comprehensive 69

Mirror simple shapes. Review important concepts learned in previous projects.

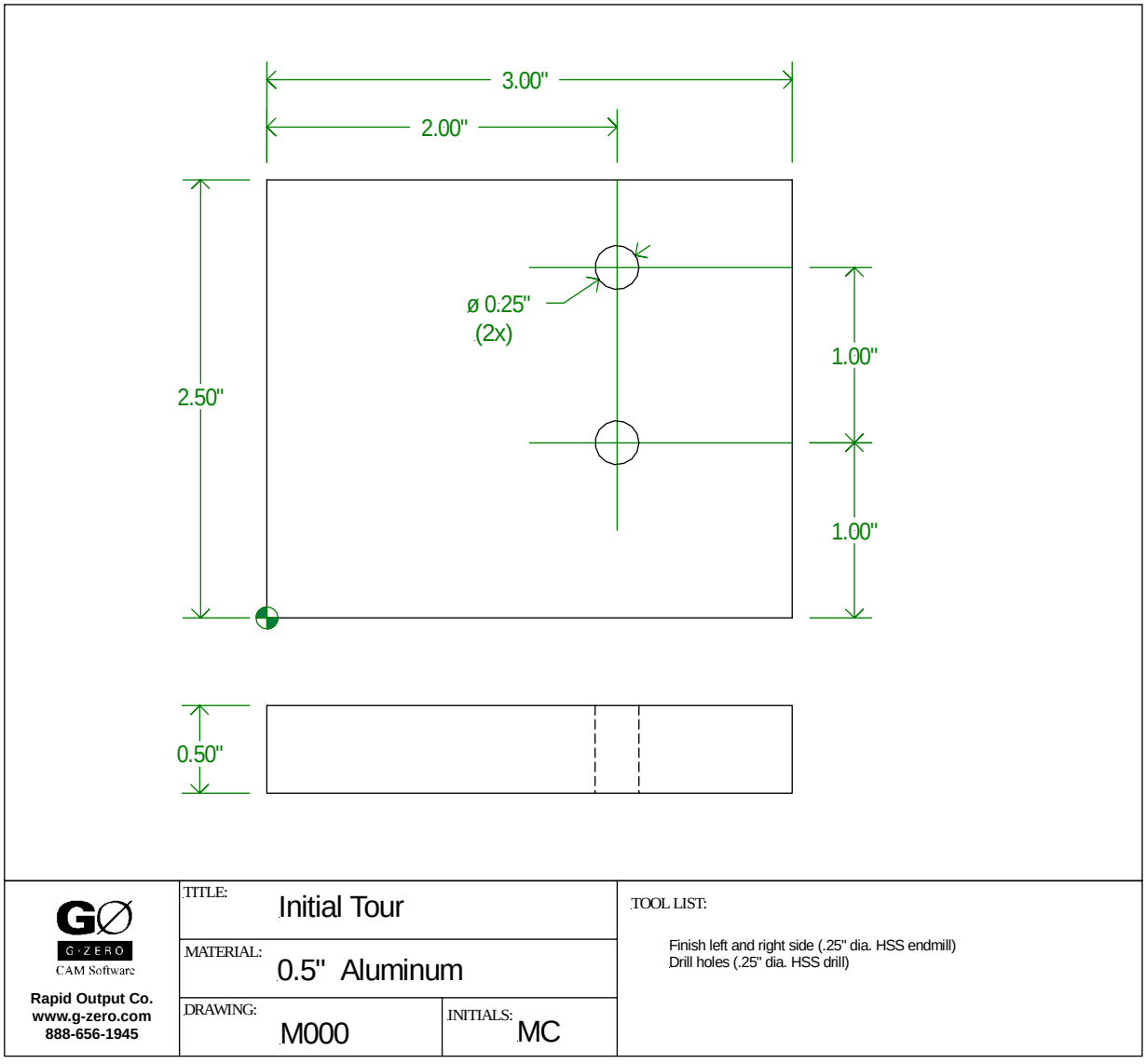
Project 1

Initial Tour



What you will learn:

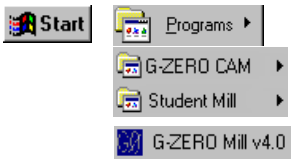
- Open G-ZERO Mill
- Start a new source file
- Use basic commands
- Use Help
- Simple editing
- View part
- Save a source file
- Convert a source file into G-code
- Exit G-ZERO Mill



G-ZERO CAM Software Rapid Output Co. www.g-zero.com 888-656-1945	TITLE: Initial Tour MATERIAL: 0.5" Aluminum DRAWING: M000 / INITIALS: MC	TOOL LIST: Finish left and right side (.25" dia. HSS endmill) Drill holes (.25" dia. HSS drill)

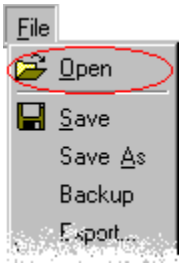
1 Start G-ZERO Mill

- From the Start menu, choose (All) Programs.
- From the G-ZERO CAM \ Mill folder, select G-ZERO Mill v5 (or v4).



2 Start a new source file

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



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

Note: G-ZERO Student version saves only 100 lines of your program to disc.

WARNING: You should never load a source from the full working version into the Student version...you will likely lose important data.

3 Use basic commands

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

Then, you need to describe the Tool you are going to use, and follow it by a cutting operation (e.g.: drill, mill) and the locations of the cut (e.g.: point, line, radius)

You can invoke a command in two different ways:

(a) Use your mouse to select the command from the left graphic (or full) menu. Example: 

(b) Use the numeric key pad to key in the number that corresponds to the specific command you want to enter, followed by the Enter key. Example: 1 7 Enter

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

Note: Each section below starts with a line of source code followed by its corresponding description or comment. For a detailed explanation of each command, see the G-ZERO Mill Reference Manual.

.

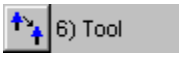
1 MAT'L xmin0 xmax3 ymin0 ymax2.5 thk.5 type0=ALUMALOY

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

According to the blueprint from the previous page, the dimensions of our part are x = 3 and y = 2.5 . We can set our origin (0;0) in the lower left corner of our part, so the dimensions will be: xmin = 0 , xmax = 3 , ymin = 0 , and ymax = 2.5 .

The thickness of the part as well as the type of material are also listed on the blueprint. Note that when you are about to answer the type of material, G-ZERO pops-up a yellow menu with the choices you currently have. You can either select it with your mouse, or key in the number that corresponds to your material.

2 TOOL 1 dia.25 flutes2 type0=HSS MILL rad0 *** CUT LEFT AND RIGHT SIDES



Define the tool for the first operation by selecting the **6) Tool** command. Just like the MAT'L command, the TOOL command also provides you with a list of tool types from which you can make your selection.

At the end of this command, you can enter a comment that will appear in your G-code file as a comment line.

---

3 MILL zrapid.1 zcut-.51 passes1 zret.1 zf45 xyf10

A MILL command tells the spindle to rapid down at the next location to a set z-value (zrapid) above the work. The spindle then feeds down, at an appropriate feedrate, to the cutting plane (zcut).

---

4 POINT x-.125 y-.125

Program a POINT to position the tool to start milling. The mill center is directly on the point; so, you need to calculate the tool radius offset.

---

5 POINT x-.125 y2.625 f5

Program this POINT for a straight cut through the material.

---

6 MILL zrapid.1 zcut-1 passes1 zret.1 zf45 xyf10

Now, we are ready to cut the right side of the material. This second MILL command makes the spindle move to the retract plane and then come down again at the next location.

---

7 POINT x3.125 y2.625

Program a POINT to position the tool to start milling at the upper right side of the part. The mill center is directly on the point; so, you need to calculate the tool radius offset.

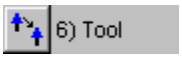
---

8 POINT x3.125 y-.125 f5

Program this POINT for a straight cut through the material.

---

9 TOOL 2 dia.25 flutes2 type20=HSS DRILL rad0 *** DRILL HOLES



Now, let's change the Tool to drill holes. Select the TOOL command and answer the questions using the information shown above .

---

10 DRILL g81=C'DRILL zrap.1 zcut-1 pecks3 tip0 zret.1 f11

A DRILL command defines the z parameter for each drill cycle. G-ZERO automatically returns to z-retract position between each location. Select the DRILL command and answer the questions using the information shown above.

---

11 POINT x2 y1

Program this POINT to the first hole to be drilled.

12 POINT x2 y2

Program this POINT to this last hole.

By now, your source code should look like this:

```
1 MAT'L   xmin0 xmax3 ymin0 ymax2.5 thk.5 type0=ALUMALOY
2  TOOL 1 dia.25 flutes2 type0=HSS MILL rad0 *** CUT LEFT AND RIGHT SIDES
3 MILL    zrapid.1 zcut-.51 passes1 zret.1 zf45 xyf10
4 POINT   x-.125 y-.125
5 POINT   x-.125 y2.625 f5
6 MILL    zrapid.1 zcut-1 passes1 zret.1 zf45 xyf10
7 POINT   x3.125 y2.625
8 POINT   x3.125 y-.125 f5
9  TOOL 2 dia.25 flutes2 type20=HSS DRILL rad0 *** DRILL HOLES
10 DRILL   g81= C'DRILL zrap.1 zcut-1 pecks3 tip0 zret.1 f11
11 POINT   x2 y1
12 POINT   x2 y2
```

4 Use Help

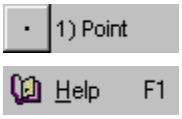
G-ZERO comes with a built-in Help system with descriptions of the commands along with some programming tips. Let's try.

- Press the F1 function key to open the Mill Commands Help.

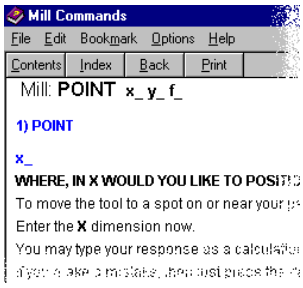
  (Click the green underlined commands to view their respective descriptions)

- Click the close icon ✕ on the upper right corner of the screen to exit the Help window.

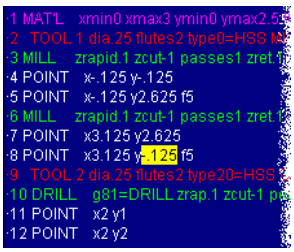
If you are in the middle of creating a command line, the F1 key will display the description of that specific command. To test it, let's create a new line with the command POINT.



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



5 Simple editing



Change a value: Let's say we want to change the y value of the point in line 8 from -.125 to .2.

- Use your mouse to click the value you want to change.

  Notice that the value you selected is highlighted and the working screen background turned to blue. This indicates that you are in the "editing" mode.
- Type in .2 to replace the highlighted value.
- Press the Enter key to exit the "editing" mode.

Replace a value with another value on the screen: Now, we are going to change the y value of the point in line 8 back to -.125 by replacing it with the y value of the point in line 4.

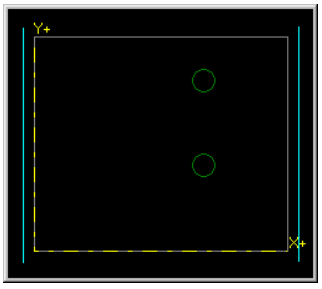
- Use your mouse to click the value you want to replace. The screen turns into "editing" mode.
- Use your mouse to click the y value (-.125) of the point in line 4. Notice that the y value of line 8 has been replaced.
- Press the Enter key to exit the "editing" mode.

Delete last line: To delete the last line of your source code, just press the Esc key.

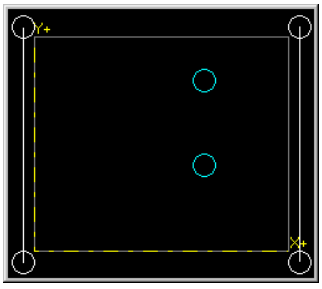
6 View / redraw the part

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

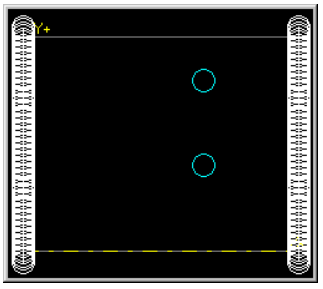
- Press F2 to redraw the entire source program.
- Press F3 to redraw the entire source program with the tool path.
- Press F4 to redraw the entire source program showing a slinky tool path.
- Press F5 to display a wireframe isometric view of the part.
- Press F6 to display a solid view of the part.



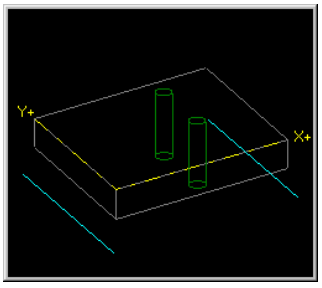
F2 - Part



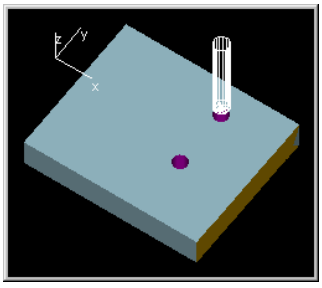
F3 - Tool



F4 - Slinky



F5 - Isometric



F6 - Solid

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

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

Fig. 1

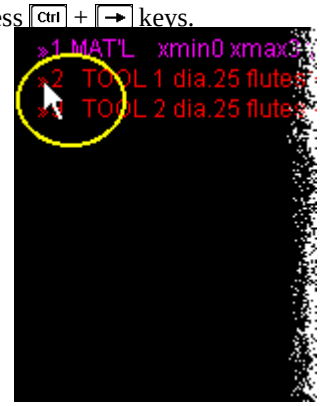


Fig. 2

Fig. 3

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

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

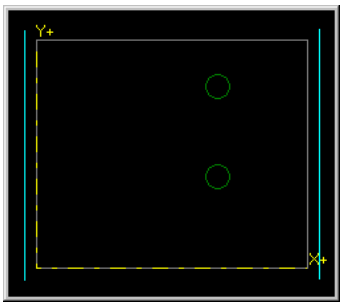


Fig. 4

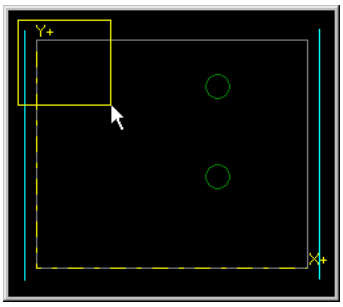


Fig. 5

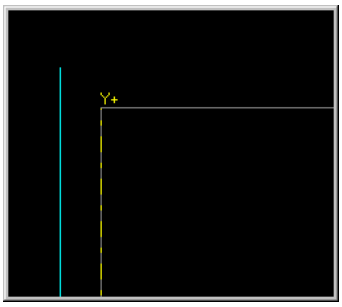


Fig. 6

7 Save a source file

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

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

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

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

8 Convert a source file into G-code

Note: The STUDENT version does not support fully-functional Post Processors. The G-code format will be mostly correct, but the numbers will make 'melted' parts.

F12: Press F12 to open a dialogue box titled Post Processor to Open... (Note that post processors are customized and sold separately)

Post: Select a tool post (for example: xFADAL.P) and watch how G-ZERO creates the appropriate G-code for the machine selected. You may have to search in a folder called Sample Posts. G-ZERO also saves the G-code file (extension .t) and displays it on your screen using Notepad, Wordpad, Codeshark, or any other text editor you might prefer. To exit the G-code editor, click the close icon  on the upper right corner of its window.

```
[  TOOL 1 dia.25 flutes2 type0=HSS MILL
      rad0 *** CUT LEFT AND RIGHT
      SIDES
[  Estimated run time for this tool = 1.35
      minutes.
[
[  TOOL 2 dia.25 flutes2 type20=HSS DRILL
      rad0 *** DRILL HOLES
[  Estimated run time for this tool = .41
      minutes.
[
[   Estimated time for this program = 1.76
      minutes.
%
N.001O1

N1G90T1M6(CUT LEFT AND RIGHT
      SIDES)
N2(CUT LEFT AND RIGHT SIDES)
N3G0E1X-.125Y-.125
N4G0S5347M3
N5H1Z.1M8
N6G1Z-.51F45.
N7Y2.625F5.
N8G0Z.1
N9X3.125
N10G1Z-1.F45.
N11Y-.125F5.
N12G0Z.1
N13M9
N14G0H0Z0M5
N15M1

N16G90T2M6(DRILL HOLES)
N17(DRILL HOLES)
N18G0E1X2.Y1.
N19G0S5347M3
N20H2Z.1M8
N21G81G99Z-1.R0.1F11.
N22Y2.
N23G0G80
N24M9
N25G0H0Z0M5
N26M1

N27/G53X0Y4.M0
N28G28E0
N29M2
%
```

Initial-Tour.t

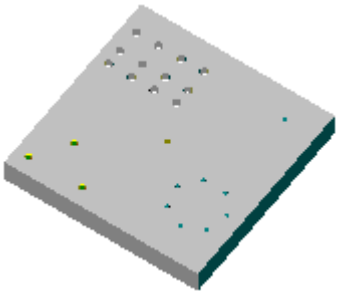
Note: Lines before the first % symbol are for your eyes only and are not sent to the CNC.

9 Exit G-ZERO

To exit G-ZERO Mill, click the close icon  on the upper right corner of its window. ■

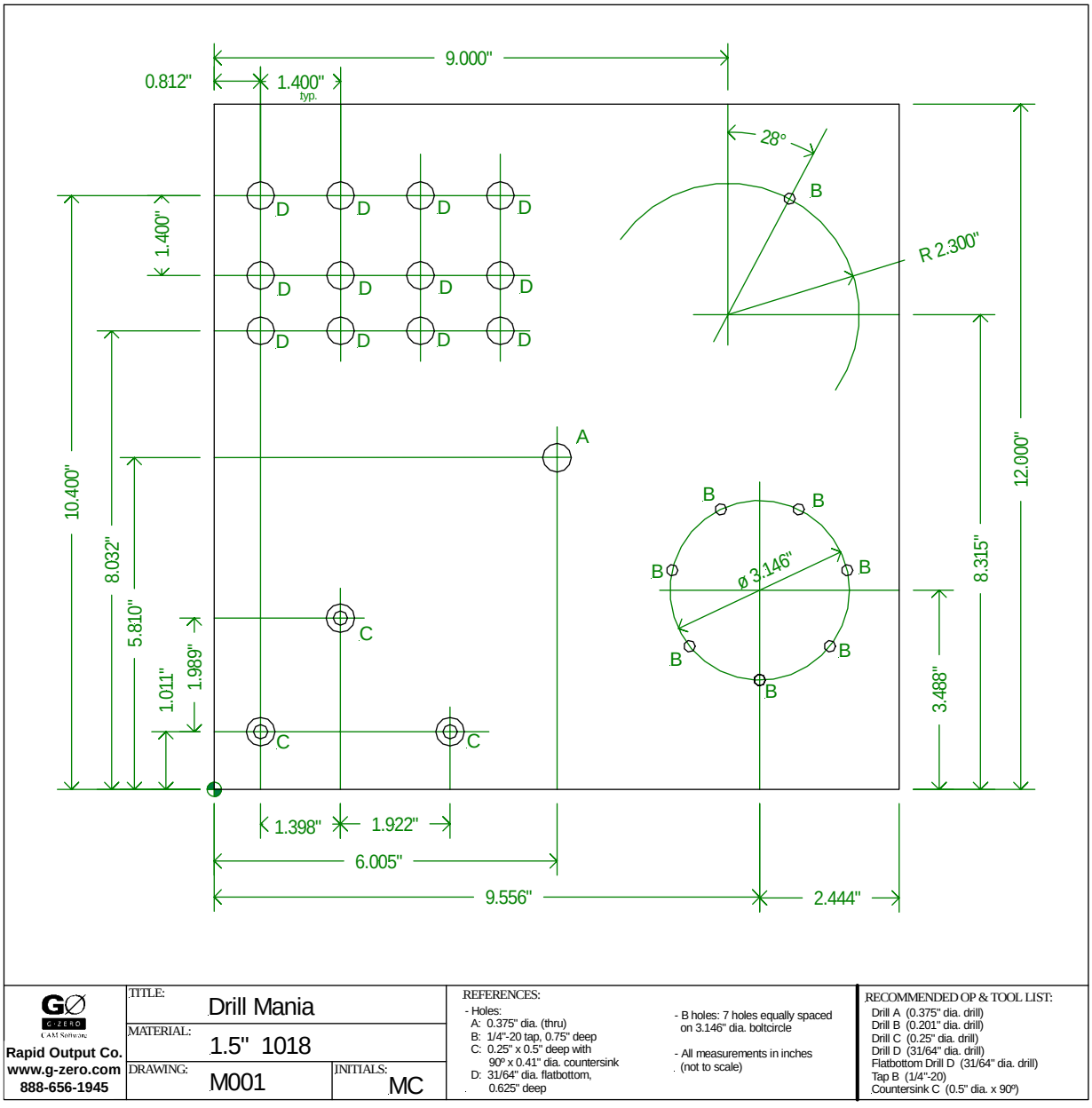
Project 2

Drill Mania



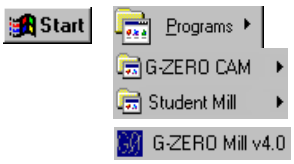
What you will learn:

- Start a new source file
- Define the material
- Drill a single hole (hole A)
- Drill a full bolt-circle (B holes)
- Drill random holes (C holes)
- Drill holes arranged in a grid (D holes)
- Use the REPEAT command
- Save and exit G-ZERO Mill



1 Start a new source file

- From the Start menu, choose Programs.
- From the G-ZERO CAM\Mill folder, select G-ZERO Mill v5 or v4.
- From the File pull-down menu, select Open to open the "Source File to Open" window.
- Type Drill-Mania in the "File Name" box, and click the Open button.
- Because the file doesn't exist, G-ZERO asks if you want to create it. Click the Yes button.

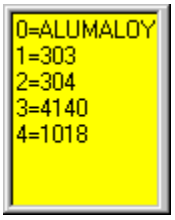


2 Define the material

1 MAT'L xmin0 xmax12 ymin0 ymax12 thk1.5 type4=1018

MAT'L is usually the first command of any source program. For G-ZERO to correctly display your part, you must describe the length and width of your material.

The origin (0;0) of our part is in the lower left corner of the material. So, the "minimum" values are going to be 0 and the "maximum" values 12. The thickness of the material is listed in the blueprint.

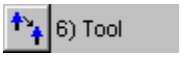


To answer the question for material type, G-ZERO displays a yellow window with all the current choices. You can either select a material with your mouse, or just key in the corresponding number.

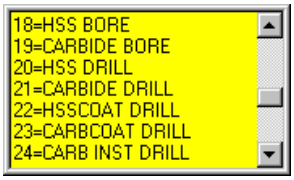
Note: The material names from the yellow window must match exactly the name of the corresponding .S files (speeds and feeds).

3 Drill a single hole (Hole A)

2 TOOL 1 dia.375 flutes2 type20=HSS DRILL rad0 ***HOLE A



A TOOL command is usually programmed after MAT'L and before a MILL or DRILL command. 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.

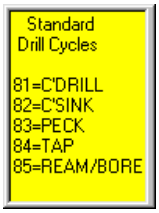


You can get the tool information from the blueprint. Note that for the type of tool, G-ZERO also displays a yellow window with all the current choices. You can either select a tool type with your mouse, or just key in the corresponding number.

The last question you are asked is to enter a comment to describe what you are going to do with the tool. Whatever you type in here will appear in your G-code file as a comment; example: HOLE A.

3 DRILL g83=PECK zrap.1 zcut-1.5 pecks6 tip1 zret.1 f1.4

Use the DRILL command to define the z parameters for drilling hole A. G83 tells G-ZERO that this operation is a multi-peck cycle.



G-ZERO automatically calculates and adds the drill tip length to zcut depth (the drill pushes completely through the material) when tip=1.

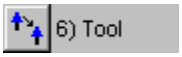
Feedrate is based upon the material and tool selected.

4 POINT x6.005 y5.81

Program this point to locate the center of hole A. The x and y values are taken from the blueprint.

4 Drill a bolt-circle (B holes)

5 TOOL 2 dia.201 flutes2 type20=HSS DRILL rad0 ***B HOLES



This TOOL command cancels the current drill cycle and retracts the spindle to tool-change position to ready for a new tool. You can get the tool information from the blueprint.

---

6 DRILL g83=PECK zrap.1 zcut-.75 pecks.3 tip1 zret.1 f1.3

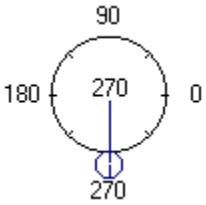
Use the DRILL command to define the z parameters for drilling the B holes. The tool automatically comes up to the z retract position and down to the z rapid position (usually the same) at each location while in drill mode.

You can enter the amount of each pecks (e.g.: .3) instead of the number of pecks.

---

7 BOLTCRC dia3.146 x9.556 y3.488 num7 st270 qu270

Now, we are going to program the full bolt-circle B (7 holes). If you can't figure out the start angle of the bolt-circle, look at the compass that appears on the screen. Mentally place the compass on top of the bolt-circle with the compass center on top of the bolt-circle center. Notice that the bottom hole lines up with the 270° axis. Enter 270º for the first and last hole of the bolt-circle. 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 first hole (first hole will not be drilled again).



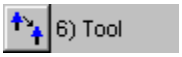
---

8 BOLTCRC dia4.6 x9 y8.315 num1 st62 qu62

To program a SINGLE hole bolt-circle, define the first and last hole angle with the same value. In our case, we are going to enter 62° (which is the complement of 28º).

5 Drill random holes (C holes)

9 TOOL 3 dia.25 flutes2 type20=HSS DRILL rad0 ***C HOLES



Now, we are going to change tools and define the new parameters for the C holes. This TOOL command cancels the current drill cycle and retracts the spindle to tool-change position to ready for a new tool. You can get the tool information from the blueprint.

---

10 DRILL g81=C'DRILL zrap.1 zcut-.5 pecks1 tip0 zret.1 f1.5

Use the DRILL command to define the new z parameters for the C holes. The tool automatically comes up to the z retract position and down to the z rapid position (usually the same) at each location while in drill mode.

G81 tells G-ZERO that this operation is a single-peck cycle.

---

11 POINT x.812 y1.011

Program this point to locate the center of the lower left C hole.

12 POINT x4.132 y1.011

Program this point to locate the center of the lower right C hole.

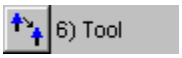
13 POINT x2.21 y3

Program this point to locate the center of the upper C hole.

6 Drill holes arranged in a grid (D holes)

14 TOOL 4 dia.484375 flutes2 type20=HSS DRILL rad0 ***D HOLES

Now, we are going to change tools and define the new parameters for the D holes. This TOOL command cancels the current drill cycle and retracts the spindle to tool-change position to ready for a new tool. You can get the tool information from the blueprint.



15 DRILL g81=C'DRILL zrap.1 zcut-.625 pecks1 tip0 zret.1 f1.1

Use the DRILL command to define the new z parameters for the C holes. The tool automatically comes up to the z retract position and down to the z rapid position (usually the same) at each location while in drill mode.

G81 tells G-ZERO that this operation is a single-peck cycle.

16 GRID num8 xnum4 x.812 xstp1.4 y10.4 ystp-1.4

GRID is an automatic cycle that drills holes dimensioned in a typical column and/or row pattern, just like our D holes. We are going to start drilling the top two rows of holes.

The total number of holes to be drilled is 8, and the number of holes in x (number of columns) is 4. The center of the first hole (upper left) is located at x=.812 and y=10.4.

The incremental distance between the center of each hole in a row is 1.4 (xstp) while the incremental distance between the center of each hole in a column is -1.4 (ystp). This y stepover is a negative value because its direction is toward the negative y axis.

17 GRID num4 xnum4 x.812 xstp1.4 y8.032 ystp0

The GRID command can also drill a single line of evenly-spaced holes. In this case, the y stepover is zero because there is no stepover in y.

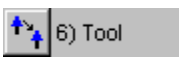
7 Use of REPEAT command (Second operations)

18 ALL SECOND OPERATIONS START HERE

Use comments for your own information or instructions to the operator. To enter a comment, just begin typing without choosing any command.

19 TOOL 5 dia.484375 flutes2 type0=HSS MILL rad0 ***FLAT BOTTOM D HOLES

The first time we drill the D holes, we used a regular angled tip drill. Now, we are going to change the drill to a flat bottomed drill (no angled tip) and drill all the D holes again.



20 DRILL g82=C'SINK zrap.1 zcut-.625 pecks5 tip0 zret.1 f1.1

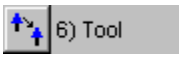
Use the DRILL command to define the new z parameters to finish the grids.

21 REPEAT from16 thru17

We can always re-use lines of codes in order to avoid retyping identical commands. In this case, we can repeat the two grid-command lines.

22 TOOL 6 dia.25 flutes2 type31=TAPMATIC NC/R rad0 ***TAP B HOLES

Now, we need to program a new tool to change the drill for a tap operation for the B holes.



23 DRILL g84=TAP zrap.2 zcut-.75 pecks20 tip0 zret.2

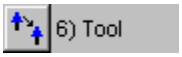
Use the DRILL command to define the new z parameters to finish the bolt-circles holes.

24 REPEAT from7 thru8

Tap both boltcircles by repeating lines 7 and 8. Using the REPEAT command makes the source programs short and easy to edit whenever necessary.

25 TOOL 7 dia.5 flutes2 type30=C'SINK rad0 ***C HOLES 1/2 IN. - 90 DEG.

Change tools and define the new parameters to countersink the C holes. Select a tool larger than .410".



26 DRILL g82=C'SINK zrap.1 zcut0 pecks90 tip.41 zret.1 f.5

Use the DRILL command to define the new z parameters to countersink the C holes.

24 REPEAT from11 thru13

Chamfer the C holes by repeating lines 11 through 13.

8 Save program and exit G-ZERO Mill

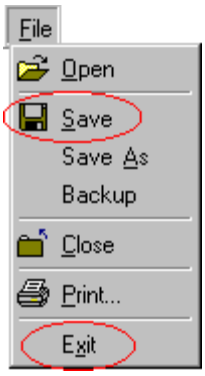
By default, G-ZERO Mill saves your work every three minutes. However, it is always a good practice to save your work whenever you think you have spent a great deal of time on your program.

To save your work, click the File pull-down menu and select Save.

Note: G-ZERO Student version saves only 100 lines of your program to disc.

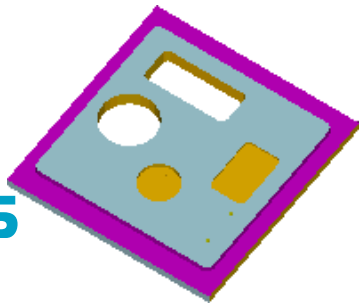
WARNING: You should never load a source from the full working version into the Student version...you will likely lose important data.

Now, that you have finished and saved your project, you can exit the program by selecting Exit from the File pull-down menu. ■



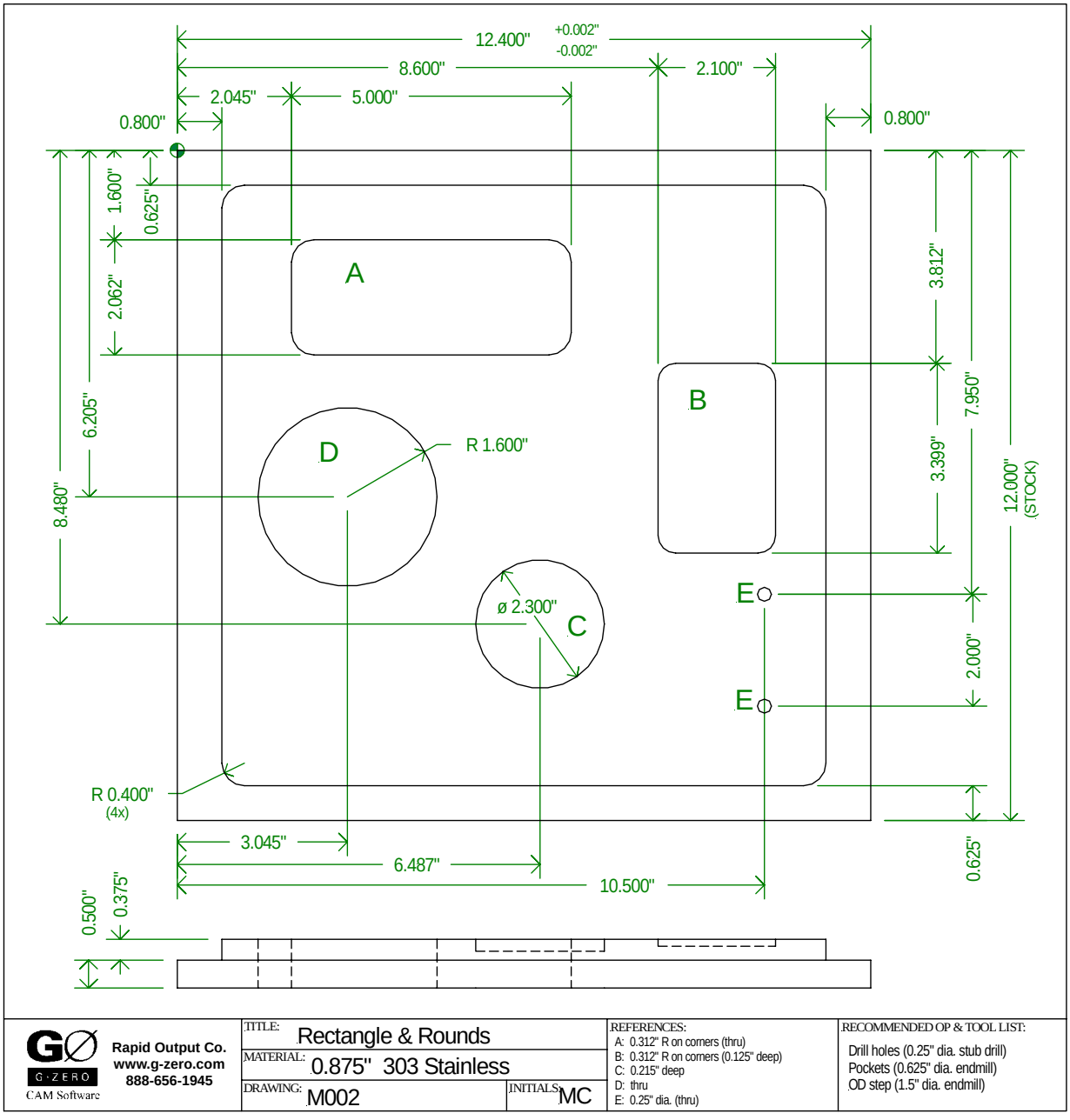
Project 3

Rectangle & Rounds



What you will learn:

- Straight cut using MILL command
- Mill rectangles (window and pocket)
- Mill circles with the ROUND command
- Rough cut with the STOCK command
- Mill OD step (COMP, UNCOMP, LINE, RADIUS commands)



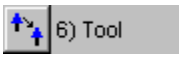
1 Define material and drill E holes

1 MAT'L xmin-.5 xmax12.8 ymin-12.5 ymax.5 thk.875 type1=303

Begin every source program with the MAT'L command. This command tells G-ZERO important information about the size and proportions of the part, the thickness and the type of material.

---

2 TOOL 1 dia.25 flutes2 type21=CARBIDE DRILL rad0 ***COBALT STUB DRILL



Define the tool for the first operation -- drilling the E holes.

---

3 DRILL g83=PECK zrap.1 zcut-.875 pecks5 tip1 zret.1 f2.8

Use the DRILL command to define parameters to drill the holes.

---

4 POINT x10.5 y-9.95

Program this point to locate the center of the lower E hole.

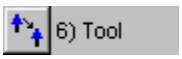
---

5 POINT x10.5 y-7.95

Program this point to locate the center of the upper E hole.

2 Straight cut using MILL command (Left and right edges of material)

6 TOOL 2 dia.625 flutes4 type1=CARBIDE MILL rad0 ***CUTTING MILL



Change the drilling tool to a milling tool to get ready for the next operation.

---

7 MILL zrapid.1 zcut-.9 passes1 zret.1 zf2.4 xyf5.3

A MILL command tells the spindle to rapid down at next location to a set z-value (zrapid) above the work. The spindle then feeds down, at an appropriate feedrate, to the cutting plane (zcut).

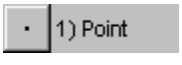
The spindle stays down at the cutting plane until one of three commands is programmed:

- TOOL: spindle retracts to the toolchange position.
- MILL: spindle moves to the retract plane (zret) and moves to the next location, and then come down.
- ZMOVE: spindle moves up or down as commanded.

The feedrates (z-feed and xy-feed) are based upon the material type, tool diameter, number of flutes, and tool type. The suggested feedrates displayed at the bottom of the window come from the modifiable Feed and Speed charts.

---

8 POINT x-.3125 y-12.5



This POINT command begins milling the left edge of the stock material to its final size. Since the mill center is directly on the point, we must calculate the tool radius offset (half of .625).

1) Point

9 POINT x-.3125 y.5 f5

Program the last point of a straight cut through the material.

7) Mill

10 MILL zrapid.1 zcut-.9 passes1 zret.1 zf2.4 xyf5.3

We need this MILL command to move the spindle to the retract plane before moving to the next location.

If this MILL command were not programmed here, the tool would cut through the material instead of rapiding above.

1) Point

11 POINT x12.7125 y.5

Program this POINT to position the tool to mill the right edge of the material. Remember to add the tool radius offset.

1) Point

12 POINT x12.7125 y-12.5 f5

Program the last point of a straight cut through the material.

3 Mill rectangles (window and pocket)

7) Mill

13 MILL zrapid.1 zcut-.9 passes3 zret.1 zf2.4 xyf11

This MILL command brings the tool up (at rapid) and ready it to come over and down at next location.

The value 3 for the number of passes divides the z-depth equally between three passes.

9) Rect

14 RECT xmin2.045 xmax7.045 ymin-3.662 ymax-1.6 thru1

RECT is an automatic cycle that mills 4-sided pockets or windows. Rectangle A is a window (cut through, thru=1), so the tool is going to cut along the sides of the rectangle without cleaning the floor. In other words, the center of the rectangle is left in one piece.

The corner radii of the rectangle are always equal to the radius of the current tool.

7) Mill

15 MILL zrapid.1 zcut-.125 passes1 zret.1 zf2.4 xyf17.1

Use the MILL command to define parameters to mill pocket.

9) Rect

16 RECT xmin8.6 xmax10.7 ymin-7.211 ymax-3.812 thru0

Since rectangle B is a pocket (thru=0), the tool is going to cut starting from the center and spiral outward so that the center of rectangle is also cleaned.

Note: The normal climb-cut spirals counterclock-wise from center outward. If you rather have a conventional-cut (spirals clockwise from center outward), then swap the xmin and xmax values. You can press the F2 key to view the tool path.

The corner radii of the rectangle are always equal to the radius of the current tool.

4 Mill circles with the ROUND command

17 MILL zrapid.1 zcut-.215 passes1 zret.1 zf2.4 xyf13

This MILL command brings the tool up (at rapid) and readies it to come over and down at next location.

---

18 ROUND dia-2.3 x6.487 y-8.48 thru0

The ROUND command mills a counterbore, circular pocket, window or standing boss. A negative (-) diameter places the tool in the inside of the circle. A positive (+) diameter places the tool in the outside of the circle (standing boss).

To mill pocket C, we need a negative diameter with thru=0

5 Rough cut with the STOCK command

19 STOCK xystk.02 zstk0

STOCK leaves extra material on the cutting surfaces by setting the distance the tool should stay away from the finished dimension of part walls and/or floor for later cleanup.

STOCK must be programmed before describing the contour's cutting path. STOCK is ON until toggled OFF with another STOCK command.

Note that the z depth value (in MILL command) as well as the contour values are given in finished dimensions. The amount of extra material is controlled by the STOCK command.

In our case, we are using a xy stock of 0.02", and z stock of 0" (since we are cutting through)

---

20 MILL zrapid.1 zcut-.9 passes3 zret.1 zf2.4 xyf11

This MILL command brings the tool up (at rapid) and readies it to come over and down at next location.

---

21 ROUND dia-3.2 x3.045 y-6.205 thru1

Program this ROUND command to mill the circular shape D. Use a negative diameter to place the cutting tool in the inside of the round. Use thru=1 to cut from the center and spiral outward so that the center of the round is also cleaned.

---

22 STOCK xystk0 zstk0

Toggle STOCK OFF by setting the xy and z stocks to zero.

---

23 MILL zrapid.1 zcut-.9 passes1 zret.1 zf2.4 xyf5.3

Program this MILL command to reset its parameters for a finish cut.

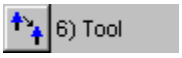
---

24 REPEAT from21 thru21 ***

Take a finish pass on round D by repeating the ROUND command on line 21.

25 TOOL 3 dia1.5 flutes4 type1=CARBIDE MILL rad0 ***CUT OD STEP

Change to a larger tool to mill the step around the contour.



---

26 STOCK xystk.01 zstk.005

Program this STOCK command to tell G-ZERO the amount of stock to leave to the final wall and floor dimensions of the outside profile.

---

27 MILL zrapid.1 zcut-.375 passes1 zret.1 zf3.1 xyf21.8

This MILL command brings the tool up and readies it to come over and down at next location.

---

28 POINT x-1.25 y-11

Program an “approach” POINT before the contour whenever possible; in other words, move the tool to a safe location (just off the part). This allows the machine’s cutter COMP to engage properly.

---

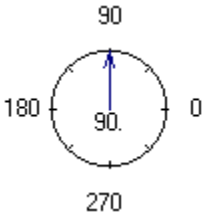
29 COMP angle90 cl/con1 lookahead0

COMP (compensate or compute) is a powerful command that releases you from calculating geometry and offsets for cutter radius.

We are going to start defining our contour from the lower left corner and going up and around the material. Therefore, the angle that our tool will be moving as we first begin compensating for the radius of the cutter will be 90º.

Our tool is going to be on the left side of the cutting path, so the cutter direction is climb (cl/con=1).

Lookahead checks for gouges by the tool. In other words, if we are using an oversized endmill to rough a contour, we will like to have G-ZERO check if the cutter fits into all the little nooks and crannies. However, since it takes quite a long time to process on long contours, we want to limit its use. In our case, we don’t need it, so lookahead=0.



---

30 POINT x.8 y-11.375

Program this POINT to bring the tool onto the contour. Note that we don’t need to add offsets for tool radius because all cutter compensation calculations are automatically done with the COMP command.

---

31 LINE angle90

Give the angle (in decimal degrees) that your tool will be moving as it travels along the line.

32 RADIUS .4 type2 x.8 y-.625

The next element we have in the contour is a radius.

We use a positive value for the radius when the tool is going to cut on the outside of the circle, and a negative radius when the tool is cutting along the inside of the circle. In this case, we need a positive 0.4 radius.

There are three types of radii:

- Center: both the x and y center dimensions of the radius are known.
- Corner: the Radius is at the intersection Point (corner) of two lines and both x and y values for the corner are known.
- Unknown: one or none of the x and y center dimensions are known.

In this case, it is a corner (type=2) radius with the corner point located at x=.8 and y=-.625. G-ZERO will calculate the center of the radius and display it later in parenthesis:

32 RADIUS .4 type2 x.8 y-.625 (xc1.2 yc-1.025)

---

33 LINE angle0

The tool will next travel along a horizontal line toward the upper right corner of the material; the angle will be 0º.

---

34 RADIUS .4 type2 x11.6 y-.625

The next radius (upper right corner) is also a corner radius because it is at the intersection of two lines and we know the values for the corner point.

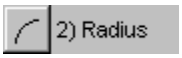
---

35 LINE angle270

The tool will next travel along a vertical line down toward the lower right corner of the material; the angle will be 270º.

---

36 RADIUS .4 type2 x11.6 y-11.375



The next radius (lower right corner) is also a corner radius.

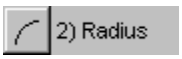
---

37 LINE angle180

The tool will next travel along a horizontal line toward the lower left corner of the material; the angle will be 180º.

---

38 RADIUS .4 type2 x.8 y-11.375



The next radius (lower left corner) is also a corner radius.

---

39 UNCOMP angle90

UNCOMP tells G-ZERO to stop compensating (calculating) for cutter radius; in other words, it turns COMP OFF.

Give the angle that your tool will be moving at the very end of the contour.

40 POINT x-1.25 y-11 f30

Program this “retract” POINT just off the part so the cutter pulls off the part without leaving a dwell mark. The tool will move from the uncomp angle on the radius to the retract point without stopping.

41 STOCK xystk0 zstk0

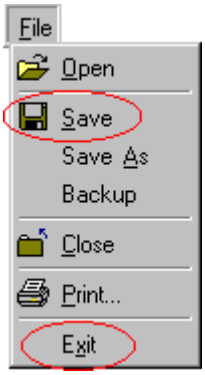
Turn off STOCK in preparation for a finish pass (stock = zero)

42 REPEAT from28 thru40 ***

To cut to finished dimensions, repeat the cutting path reusing source lines 28 to 40.

7 Save and exit

Now that you are done with this project, save your file and exit the program. ■

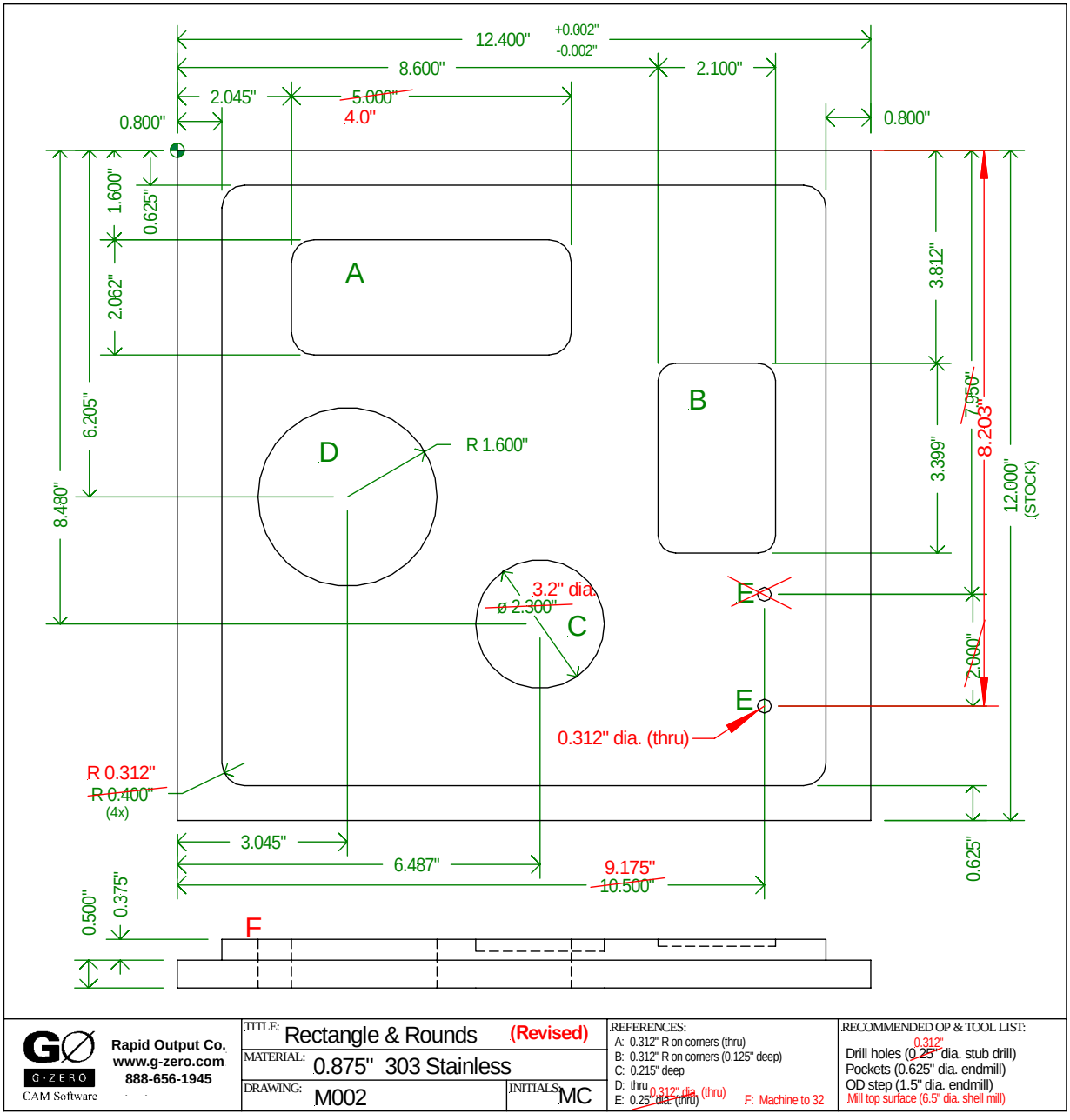


Project 4

Rectangle & Rounds (revisions)

What you will learn:

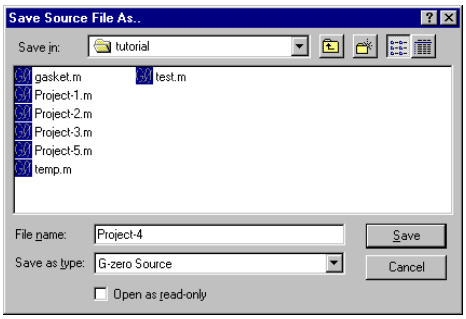
- Open an existing source file
- Create a copy of a source file using Save As
- Print a source file
- Editing commands

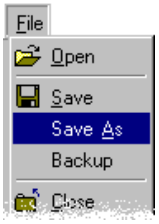


1 Open an existing file. Create a copy of a source file using Save As

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

1. Open your source program Project-3.m (you may have used another file name, such as Rectangles-and-Rounds). If you are starting G-ZERO, select the file Project-3.m in the "Source File to Open" window.
2. From the File pull-down menu, select Save As.
3. Type the new file name Project-4 (use other name if you wish) in the File name section and click the Save button.





Your Project-3.m is still intact in your computer. A new file named Project-4.m was created and saved in the same directory as Project-3.m. The new file Project-4.m is now the active program; any change you make is going to affect the new file.

2 Print a source file

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

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

G-ZERO will ask you a couple of questions to determine the range of source codes you want to print.

In this case, we want to print the whole source program, from line 1 to line 42. Your current source program will be printed on the default printer set up on your computer.

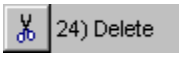
Note that this command line (PRINT from1 thru42) is NOT added into your source program.

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

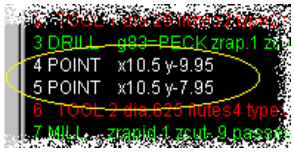
3 Replace two holes with one hole (Hole E)

Delete lines that correspond to the two E holes

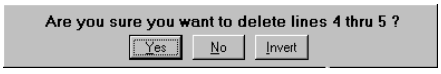
42 DELETE from4 thru 5



After checking the souce codes to identify the lines that correspond to the E holes (lines 4 and 5), we are going to delete these two lines.



G-ZERO will ask to confirm this deletion. Click yes.



Note 1: this command line is NOT added into your source program.

Note 2: Line numbers are not changed; line numbers 4 and 5 are skept.

Add a line for the new E hole

42 INSERT after3

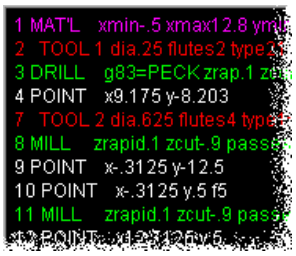
Use the INSERT command to tell G-ZERO you want to add **one** line of command right after line 3. Note that this command line is NOT added to your source program.

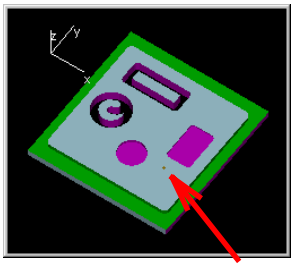
Since you are in the "editing" mode, the working screen background turned to blue. G-ZERO is now waiting for you to enter the command line you want to add as line 4.

4 POINT x9.175 y-8.203

Check the revised blueprint to get the values for the center of the new hole E.

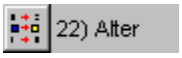
Note: New point command is in line number 4. Lines number 5 and 6 are skept (2 lines were previously deleted). The rest of the line codes are resequenced.

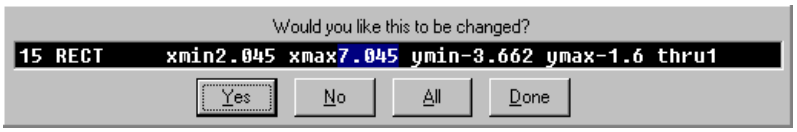




4 Change value of a rectangle using the ALTER command (Rectangle A)

43 ALTER line15 from7.045 to6.045





Checking the source code, we can find out that the new line number for Rectangle A is line 15.

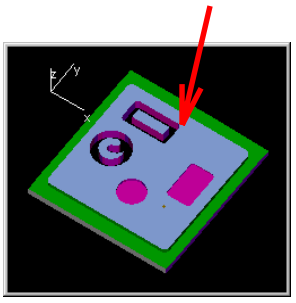
We need to change the value 7.045 to 6.045. G-ZERO displays a window to confirm the value you want to change. You have 4 options for this confirmation:

Yes: G-ZERO will change the incorrect value and look down the Source program for another occurrence of the same incorrect value.

No: G-ZERO will not change the incorrect value but will look down the Source program for another occurrence of the same incorrect value.

Done: G-ZERO will not change the incorrect value and will not look for more occurrences.

All: G-ZERO will change the incorrect value and EVERY number in the source that also matches the incorrect value — without double-checking. WARNING: Using "All" can be very dangerous.



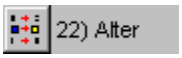
Click the Yes button to confirm this change.

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

5 Change corner radii on step using the ALTER command

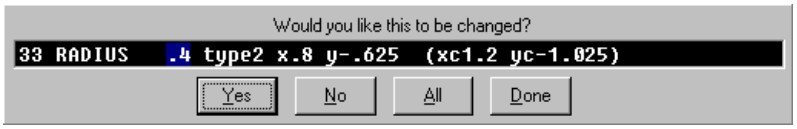
Now, we are going to change all 4 corner radii on the step (lines 33, 35, 37 and 39) using one ALTER command.

43 ALTER line33 from.4 to.312



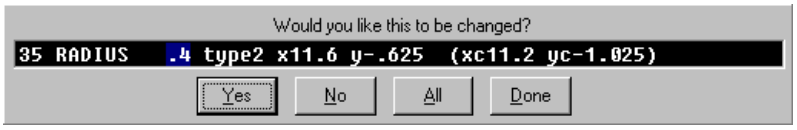
Use the ALTER command to change the value .4 to .312 on line 33.

G-ZERO displays a window to confirm the value you want to change.



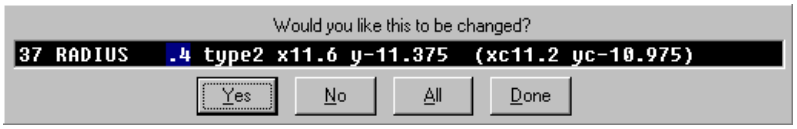
Click the Yes button to confirm this change.

G-ZERO displays a second screen to confirm another value .4 it found on line 35.



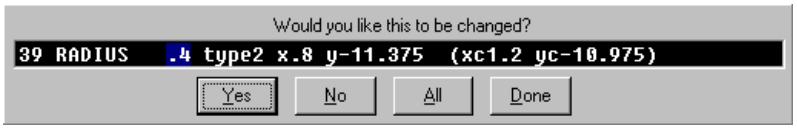
Click the Yes button to confirm this change.

G-ZERO displays a third screen to confirm another value .4 it found on line 37.



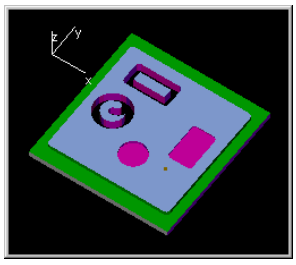
Click the Yes button to confirm this change.

G-ZERO displays a fourth screen to confirm another value .4 it found on line 39.



Click the Yes button to confirm this change.

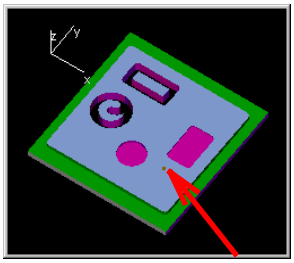
Note that this ALTER command line is NOT added to your source program.



6 Change drill diameter using mouse and keypad (Hole E)

Another way to change a value is by using your mouse. To change diameter value .25 to .312 from line 2, follow these steps:

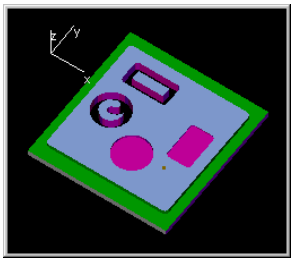
1) Locate the value you need to change and select it with your mouse. Now, the value you selected is highlighted and the working screen background turned to blue (you are in the "editing" mode).
2) Key in the new value .312. This will replace the highlighted value with the number you typed.
3) Press the Enter key to exit the "editing" mode returning your working screen to black background.



7 Change round diameter using mouse only (Round C same as Round D)

Now, we are going to replace the Round C diameter (-2.3) on line 19 with Round D diameter (-3.2) on line 22 using mouse only:

1) Locate the value you need to change and select it with your mouse (-2.3 from line 19). Now, the value you selected is highlighted and the working screen background turned to blue (you are in the "editing" mode).
2) Use your mouse to select the value you want to change to (-3.2 from line 22). Notice that the diameter on line 19 is replaced with the new value.
3) Press the Enter key to exit the "editing" mode returning your working screen to black background.

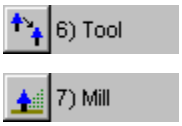


8 Machine top surface of the part (Move lines of codes)

To machine the top surface of the part, program the whole cutting sequence (Tool, Mill, Points) at the end of the source program. Then, MOVE this sequence of commands after line 1 so that it becomes the first cutting operation of the program.

Program cutting sequence

```
44   TOOL  4   dia6.5 flutes8 type5=CARBIDE INSERT MILL rad0 *** SHELL MILL
45   MILL       zrapid.1 zcut0 passes1 zret.1 zf3.1 xyf21.8
46   POINT      x-3.5 y-3
47   POINT      x12 y-3 f3.9
48   POINT      x12 y-9 f3.9
49   POINT      x-3.5 y-9 f3.9
```



When you try to enter dia "6.5" and flutes "8", G-ZERO may not allow you to do it because you are attempting to enter a number outside the system's default limits. Since you are sure the numbers are valid, press the letter O key to OVERRIDE, and then the Enter key.

Move lines of code

49 MOVE from44 thru49 after1

The line codes we need to move are from 44 to 49, and we want to move them to the beginning of the program, right after the MAT'L command.

Note that after you moved these lines, G-ZERO automatically renumbered all lines and updated the line numbers inside the REPEAT command.

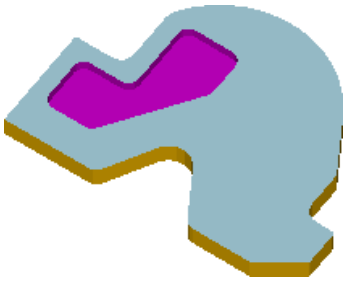
The MOVE command will not be added to your source program.

9 Save and exit

Now that you are done with this project, save your file and exit the program. ■

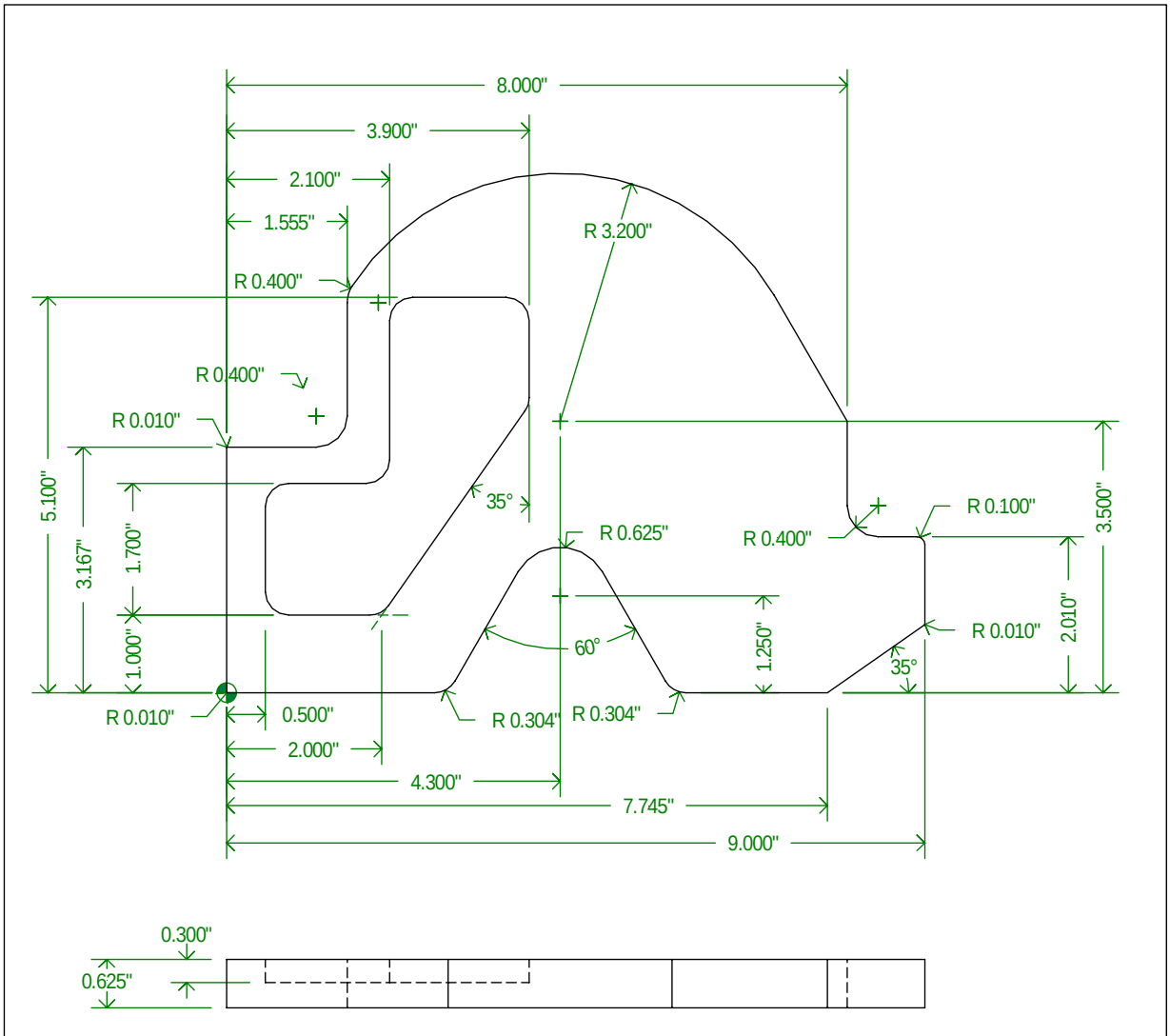
Project 5

Turtle



What you will learn:

- Mill elaborate profile with unknown values
- Mill pocket using circular ramping (MILL, ZMOVE, ROUND) and COMP with blend radii.



Rapid Output Co. www.g-zero.com 888-656-1945	TITLE: Turtle	REFERENCES:	RECOMMENDED OP & TOOL LIST:
	MATERIAL: 0.625" 1018 Cold Roll	Pocket: 0.3" deep Fillets and rounds 0.3"R	Rough OD (.75" dia. hogmill) Finish OD (.5" dia carbide endmill) Rough & Finish pocket (.25" dia HSS endmill)
	DRAWING: M003 / INITIALS: MC		

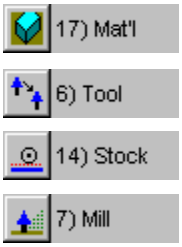
Mill elaborate profile with unknown values

1 MAT'L xmin-.5 xmax9.5 ymin-.5 ymax7.2 thk.625 type4=1018

2 TOOL 1 dia.75 flutes4 type6=HOGMILL rad0 ***ROUGH PROFILE

3 STOCK xystk.025 zstk0

4 MILL zrapid.02 zcut-.65 passes1 zret.4 zf4.3 xyf10.5



Use the blueprint to get information for the MAT'L and TOOL commands.

Since we are milling a rough profile first, we are programming the command STOCK with 0.025" of material left on part walls.

---

5 POINT x-.6 y-.6

Remember to program an approach POINT to move the tool to a safe location and allow the machine's cutter COMP to engage properly. Since we are going to start cutting from the lower left corner of the material, a safe point will be (-.6;-.6).

---

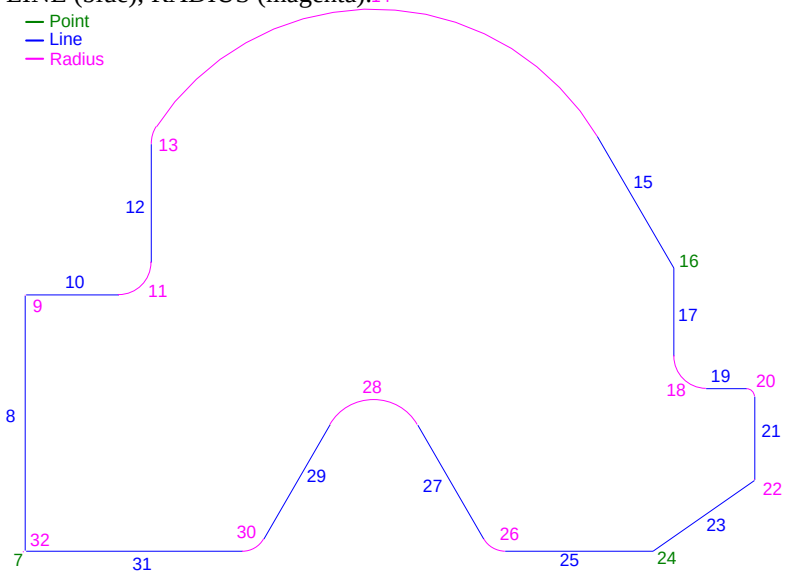
6 COMP angle90 cl/con1 lookahead0

COMP automatically calculates offsets for cutter radius. We are going to start defining our contour from the lower left corner and go up and around the material. Therefore, the angle that our tool will be moving as we first begin compensating for the radius of the cutter will be 90º.

Our tool will be on the left side of the cutting path, so the cutter direction is climb (cl/con=1).

The following graph shows the different elements we are going to program within the COMP-UNCOMP commands. The numbers shown in the graph indicates the source program line number used in this project.

Note that each command is shown in a different color: POINT (green), LINE (blue), RADIUS (magenta).14



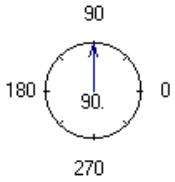
7 POINT x0 y0

This is the first point of the contour.

---

8 LINE angle90

This line has angle 90º because the tool will be moving in that direction as it travels along the line.



---

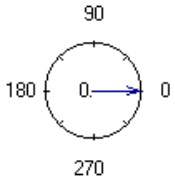
9 RADIUS .01 type2 x0 y3.167

The tool is cutting along the outside of the radius, so it has a positive radius. It is a corner (type 2) radius with the intersection point at (0;3.167).

---

10 LINE angle0

The tool will be moving horizontally towards the right in the 0º direction.



---

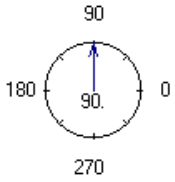
11 RADIUS -.4 type2 x1.555 y3.167

The tool is cutting along the inside of the radius, so it has a negative radius. It is a corner (type 2) radius with the intersection point at (1.555;3.167).

---

12 LINE angle90

This line has angle 90º because the tool will be moving in that direction as it travels along the line.



---

13 RADIUS .4 type0

The tool is cutting along the outside of the radius, so it has a positive radius. It is an unknown radius because we do not know the (x;y) values of the center (nor corner) of this arc. Enter type=0 for unknown type and let G-ZERO calculate the center of the arc (it will be listed on your source code in parenthesis after the next location is given).

---

14 RADIUS 3.2 type1 x4.3 y3.5

The tool is cutting along the outside of the radius, so it has a positive radius. We know the center of this radius (type 1) is located at (4.3;3.5).

---

15 LINE angle(

The angle of this line is unknown. In this case, type the “open parenthesis” and G-ZERO automatically calculates the unknown angle after the next known location is given.

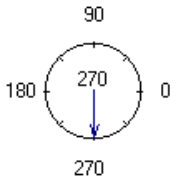
---

16 POINT x8 y3.5

Describe the intersection of two lines (sharp corner) as a point.

17 LINE angle270

This line has angle 270º because the tool will be moving in that direction as it travels along the line.

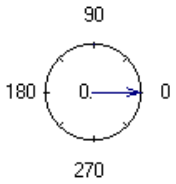


18 RADIUS -.4 type2 x8 y2.01

The tool is cutting along the inside of the radius, so it has a negative radius. It is a corner (type 2) radius with the intersection point at (8;2.01).

19 LINE angle0

The tool will be moving horizontally towards the right in the 0º direction.

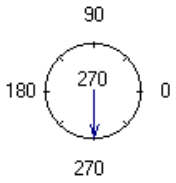


20 RADIUS .1 type2 x9 y2.01

The tool is cutting along the outside of the radius, so it has a positive radius. It is a corner (type 2) radius with the intersection point at (9;2.01).

21 LINE angle270

This line has angle 270º because the tool will be moving in that direction as it travels along the line.

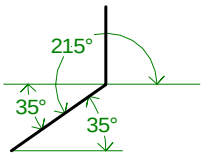


22 RADIUS .01 type0

The tool is cutting along the outside of the radius, so it has a positive radius. It is an unknown radius because we do not know the (x;y) values of the center (nor corner) of this arc. Enter type=0 for unknown type and let G-ZERO calculate the center of the arc (it will be listed on your source code in parenthesis after the next location is given).

23 LINE angle215

The tool will be moving in a 215º direction.

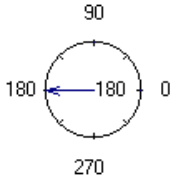


24 POINT x7.745 y0

Describe the intersection of two lines (sharp corner) as a Point.

25 LINE angle180

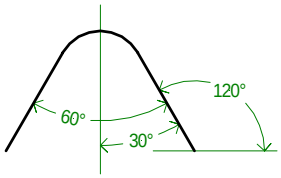
This line has angle 180º because the tool will be moving horizontally toward the left as it travels along the line.



26 RADIUS .304 type0

The tool is cutting along the outside of the radius, so it has a positive radius. This is an unknown radius, so enter type=0.

27 LINE angle120

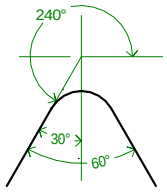


The tool will be moving in a 120º direction.

---

28 RADIUS -.625 type1 x4.3 y1.25

The tool is cutting along the inside of the radius, so it has a negative radius. This is a center radius (type=1) with center of radius located at (4.3;1.25).



---

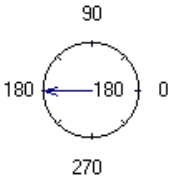
29 LINE angle240

The tool will be moving in a 240º direction.

---

30 RADIUS .304 type0

The tool is cutting along the outside of the radius, so it has a positive radius. This is an unknown radius, so enter type=0.



---

31 LINE angle180

The tool will be moving horizontally toward the lower left corner of the material in a 180º direction.

---

32 RADIUS .01 type2 x0 y0

The tool is cutting along the outside of the radius, so it has a positive radius. This is a corner radius (type 2) with the intersection point at (0;0).

---

33 UNCOMP angle90

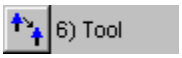
UNCOMP tells G-ZERO to stop compensating (calculating) for cutter radius; in other words, it turns COMP OFF.

Give the angle that your tool will be moving at the very end of the contour.

---

34 POINT x-.6 y-.6 f0

Program this “retract” POINT to pull the cutter off the part without leaving a dwell mark.



---

35 TOOL 2 dia.5 flutes2 type1=CARBIDE MILL rad0 *** FINISH PROFILE

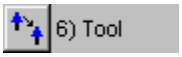
36 MILL zrapid.02 zcut-.65 passes1 zret.1 zf7.3 xyf10.7

37 REPEAT from5 thru34

Change tools to finish the already roughed contour. (The TOOL command resets STOCK to 0).

To take a finish cut around the defined profile, repeat the cutting path by using source lines starting at “approach” point through “retract” point.

38 TOOL 3 dia.5 flutes2 type0=HSS MILL rad0 ***ROUGH AND FINISH POCKET



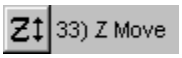
Change tools to rough and finish pocket.

---

39 MILL zrapid.02 zcut0 passes1 zret.1 zf2 xyf7.7

40 ZMOVE z-.3 ramp1 f5

41 ROUND dia-1.7 x3 y4 thru45



The MILL command brings the tool over (at rapid) and readies to come down at the next location. Because the tool will be ramping into the pocket (using ZMOVE in line 40), we program this zcut = 0 (surface of the part). The tool will feed from the zrapid plane to the surface of the part.

Lines 40 and 41 describe a circular ramping from the surface of the part (z=0) to the floor of the pocket (z=-.3). ZMOVE controls the depth of the cut (0.3") and the feedrate (5 inches per minute). ROUND makes it a circular ramping. In other words, the tool will start ramping from the surface of the part (zcut=0) at 45º of the edge of the circle (thru = 45), make a circular cut while feeding into the pocket, and finish the ramping at the floor of the pocket (z=-.3) back to the 45º of the edge of the circle.

Fig. 1 below shows how the tool feeds into the pocket in circular motion.

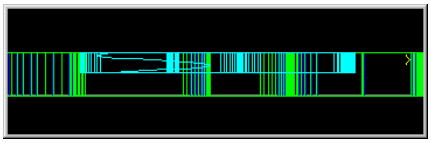


Fig.1: Side view

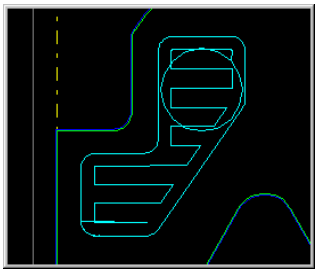
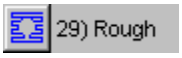


Fig.2: Top view

42 ROUGH stk.02 stp80 angle270 cleanup1



The ROUGH command leaves cleanup STOCK (wall), and includes an optional automatic cleanup pass so the STOCK and REPEAT commands (finish cut) are not necessary (unless you need to change tools for the finish cut).

We are going to define the stepover per pass as a percentage of the tool (80%). This means that the 0.5" tool will step 0.4" per pass. (If the step value is 2 or smaller, G-ZERO will assume it is an absolute amount.)

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. The first and second ROUGH must be identical for G-ZERO to calculate a roughing cycle.

G-ZERO automatically roughs the contour given between the first and second ROUGH commands. Actual roughing does not occur until the second ROUGH command is programmed.

The roughing angle is the general direction of material removal, not the back and forth motion of the Tool. So, our angle of 270º indicates that the tool begins roughing at the top and finishes at the bottom of the pocket.

43 COMP angle270 cl/con1 lookahead0

4) Comp

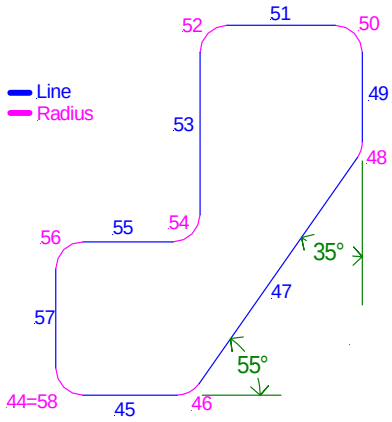
Program a COMP to tell G-ZERO to calculate offsets for cutter radius and define the beginning of the cutter path.

The angle 270º is the tangent angle to the blend-on RADIUS of the first RADIUS command. As the tool starts cutting the inside of the RADIUS, the tool is moving down (270º).

44 RADIUS -.3 type1 x.8 y1.3
45 LINE angle0
46 RADIUS -.3 type1 x2 y1.3
47 LINE angle55
48 RADIUS -.3 type0
49 LINE angle90
50 RADIUS -.3 type2 x3.9 y5.1
51 LINE angle180
52 RADIUS -.3 type2 x2.1 y5.1
53 LINE angle270
54 RADIUS .3 type2 x2.1 y2.7
55 LINE angle180
56 RADIUS -.3 type2 x.5 y2.7
57 LINE angle270
58 RADIUS -.3 type1 x.8 y1.3

2) Radius

3) Line



Program lines 44 to 58 to define the contour of the pocket. Refer to the graph at the right to identify the elements (lines and arcs) you are programming. Note that the numbers correspond to the source code line numbers. Lines are in blue and arcs (radii) in magenta.

Because the tool is going to cut along the inside of all arcs (except for line 54) the value of the radii are negative (radius for line 54 is positive).

Notice that line 44 and 58 program the same radius. This radius is called a blend radius because it follows the COMP command (blend-on radius) or precedes the UNCOMP command (blend-off radius). All blend radii need to be:

1) Always negative (tool is on the inside)
2) Always dimensioned by its center (type=1)
3) Always tangent to the first surface to be cut.
4) At least twice the diameter of the tool to avoid a gouge.

59 UNCOMP angle15

5) Uncomp

This UNCOMP command tells G-ZERO to stop compensating for cutter radius. The UNCOMP angle (15º) is the tangent angle from the radius (line 58). As the tool cuts around the inside of the blend-off radius it stops as it reaches 15º.

60 ROUGH stk.02 stp80 angle270 cleanup1

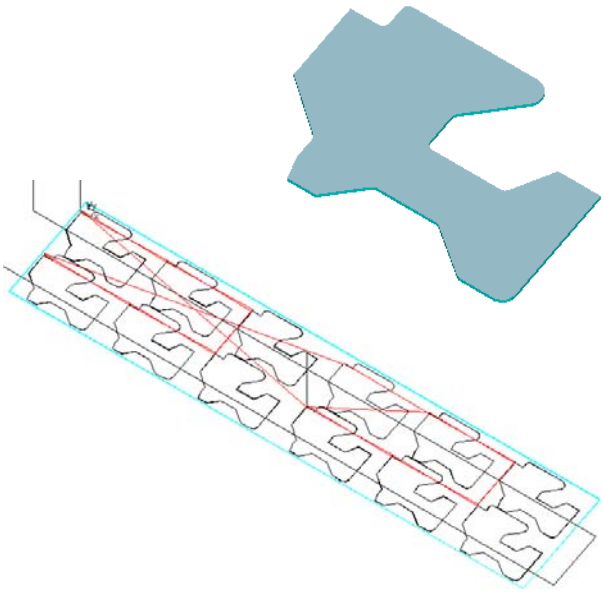
29) Rough

Program this second ROUGH to "close" the pocket and activate the pocket roughing. G-ZERO will automatically zig-zag rough the contour programmed between the first and last ROUGH commands. Angle 270 sets the tool stepping direction. (See Fig. 2 on previous page)

(Lines 42 and 60 should be identical.) ■

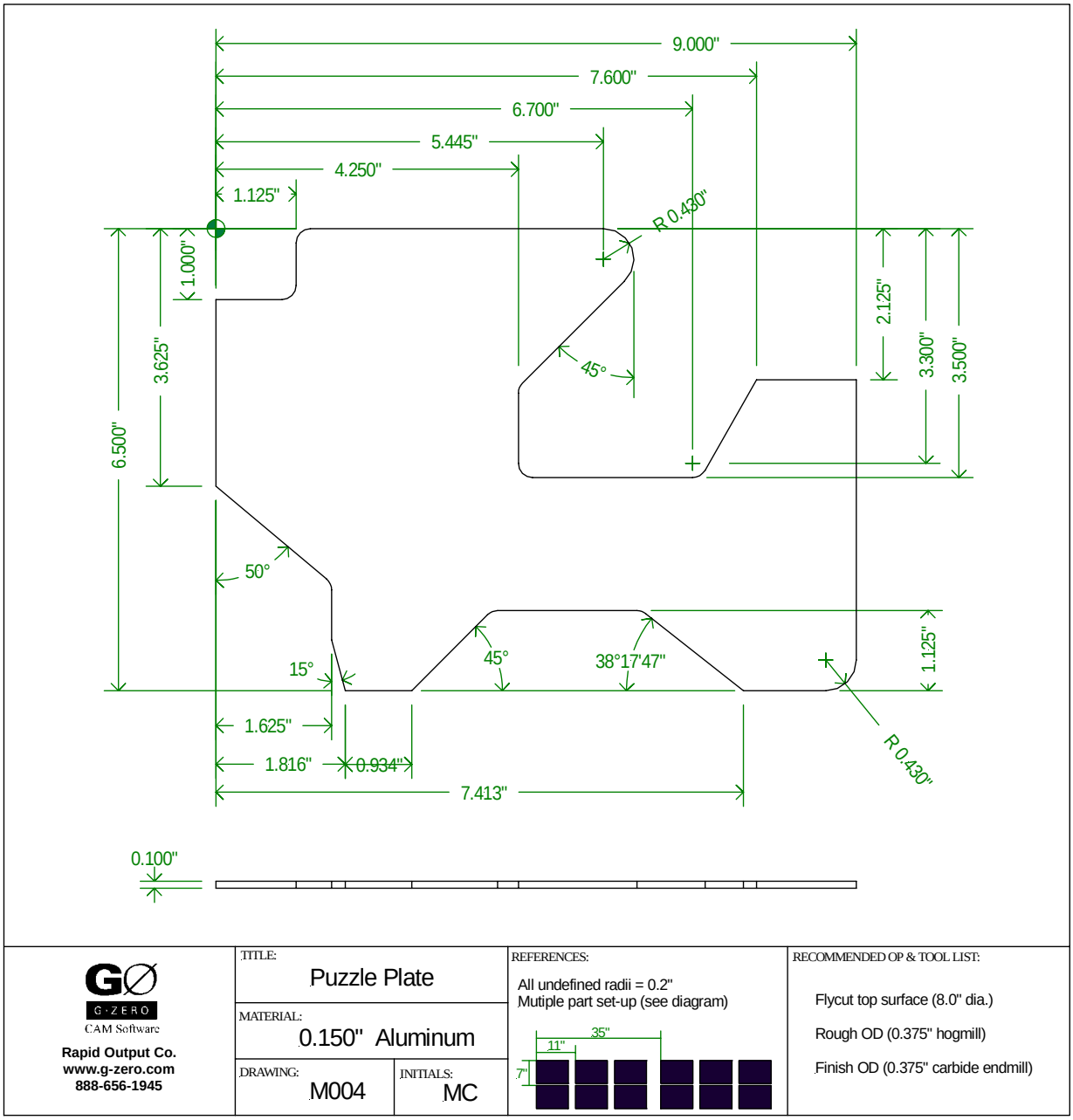
Project 6

Puzzle Plate



What you will learn:

- Multiple part setup using the MULTIPLY command
- Flycut using comment ZTOP
- Conventional cutting
- Program two unknown radii in a row
- Enter radius in degree/minute/second format
- Reverse cutter path



G·ZERO CAM Software **Rapid Output Co.** **www.g-zero.com** **888-656-1945**	TITLE: Puzzle Plate MATERIAL: 0.150" Aluminum DRAWING: M004 INITIALS: MC	REFERENCES: All undefined radii = 0.2" Mutiple part set-up (see diagram) 	RECOMMENDED OP & TOOL LIST: Flycut top surface (8.0" dia.) Rough OD (0.375" hogmill) Finish OD (0.375" carbide endmill)

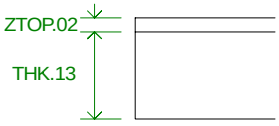
This project includes a multiple part setup. Use Tool 1 as a flycutter over all twelve parts. Program Tools 2 and 3 for one part only and let MULTIPLY generate the subroutines and loops in your G-code for the remaining parts.

1 Define material

1 ZTOP.02

2 MAT'L xmin-.5 xmax66.5 ymin-14 ymax.5 thk.13 type0=ALUMALOY

Since we are going to program a flycut over the whole material, we need to enter a ZTOP comment line right before the MAT'L command. In our case, we are going to show 0.02" of material to be cut. The thickness given in the MAT'L command (thk=0.13) is the actual thickness of the part after flycut.



As you can see in the blueprint, the finished part is 9"x 6.5". However, we are going to cut multiple parts, so the material command needs to reflect the dimension of the raw material.

2 Flycut over whole material

3 TOOL 1 dia8 flutes6 type5=CARBIDE INSERT MILL rad0 ***FLYCUT ALL

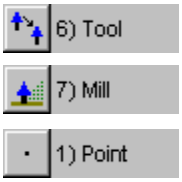
4 MILL zrapid.05 zcut0 passes1 zret.1 zf2.7 xyf11

5 POINT x-4.5 y-3.75

6 POINT x70.5 y-3.75 f10

7 POINT x70.5 y-11.7 f10

8 POINT x-4.5 y-11.7 f10



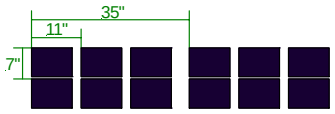
This tool cuts across the entire setup (6 parts across, 2 parts down). The flycutter will NOT be included as a multiple part because this tool is programmed before the MULTIPLY command.

The MILL command directs the flycutter to bring the material to a z-depth of 0 (zero) before the cutting.

3 Program MULTIPLY command

9 MULTIPLY xn3 yn2 xs11 ys-7 gn2 gs35 sta0 ***

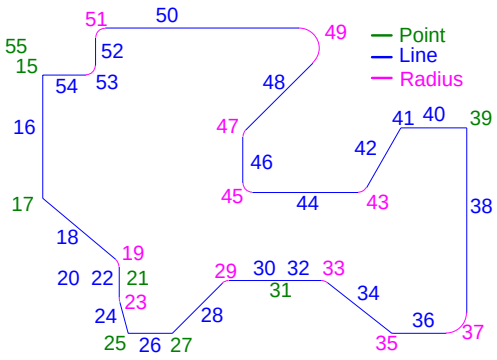
This MULTIPLY command programs 12 identical parts: 2 groups (gn2), each one with 3 parts in a row (xn3) and 2 parts in a column (yn2).



Only one part is displayed on screen; the subroutines and loops for the remaining parts are generated by your post processor(s).

MULTIPLY should be programmed immediately before the first tool included in the multiple parts. MULTIPLY stays in effect throughout the entire program and is automatically cancelled when the program terminates.

4 Rough Profile (with conventional cut, unknown radii, angle in degrees)



10 TOOL 2 dia.375 flutes4 type6=HOGMILL rad0 *** ROUGH PROFILE
11 STOCK xystk.02 zstk0
12 MILL zrapid.02 zcut-.115 passes1 zret.1 zf13 xyf38.6

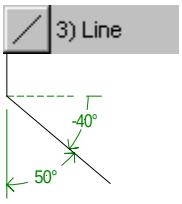
13 POINT x-.2 y-1

Program an approach point just off the part to drop the cutter in a safe location. The tool is centered directly on the coordinates (-.2;-1).

14 COMP angle270 cl/con2 lookahead0

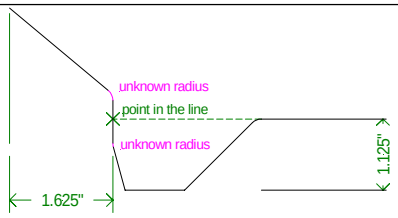
COMP automatically calculates offsets for cutter radius. The angle (270º) is the direction the tool will be traveling between line 15 and 16.

Note that we introduced here **conventional** type of cut. This means that the tool will remain on the right side of the material during the cut (cl/con=2). To activate your CNC's G42 cycle for conventional cutting, type **42** instead of **2**.



15 POINT x0 y-1
16 LINE angle270
17 POINT x0 y-3.625
18 LINE angle-40

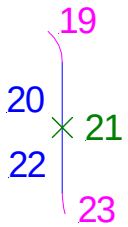
19 RADIUS -.2 type0
20 LINE angle270
21 POINT x1.625 y-5.375
22 LINE angle270
23 RADIUS .001 type0



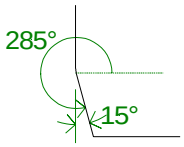
Radii on lines 19 and 23 are unknown because the blueprint does not provide the (x;y) locations of these radii. We cannot program two consecutive unkwnon radii (type=0) even if they are separated by a line.

However, we can easily find the (x;y) values of a point within the line and insert it as a "fake" point. This point could be x=1.625 and y=-(6.5-1.125)=-5.375.

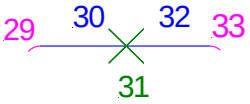
LINE commands from lines 20 and 22 should be identicals because they are in fact the same line.

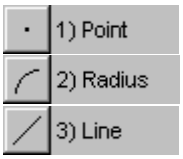


24 LINE angle285
25 POINT x1.816 y-6.5
26 LINE angle0
27 POINT x2.75 y-6.5
28 LINE angle45



29 RADIUS -.2 type0
30 LINE angle0
31 POINT x4.5 y-5.375
32 LINE angle0
33 RADIUS -.2 type0



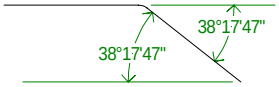


Radii on lines 29 and 33 are unknown because the blueprint does not provide the (x;y) locations of these radii. We cannot program two consecutive unkwnon radii (type=0) even if they are separated by a line.

However, we can easily find the (x;y) values of a point within the line and insert it as a “fake” point. The x value of this point could be between 4.250 and 5.445 (see top dimensions on blueprint). A point on this line is x=4.5 and y=-(6.5-1.125)=-5.375.

LINE commands from lines 30 and 32 should be identicals because they are in fact the same line.

34 LINE angle-38.29639



G-ZERO needs angles in decimal format. To program an angle dimensioned in degrees/minutes/seconds, type in the angle using the format **dd.mmss** followed by the quote ["] key to convert to decimal, and press the [Enter] key.

In this case,	to program:	**-38°17’47”**
	type this:	**-38.1747”[Enter]**
and G-ZERO will convert to:		**-38.29639**

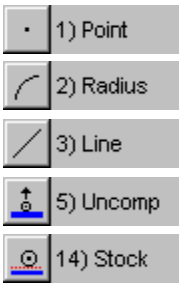
35 RADIUS .001 type2 x7.413 y-6.5
36 LINE angle0
37 RADIUS .43 type2 x9 y-6.5
38 LINE angle90
39 POINT x9 y-2.125
40 LINE angle180
41 POINT x7.6 y-2.125

42 LINE angle(

The angle of this line is unknown. By typing the open parenthesis [( ] key, G-ZERO automatically calculates the unknown angle after the next known location is program.

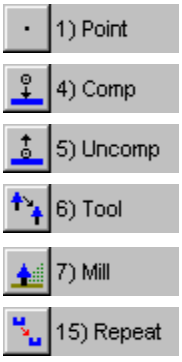
In this case, the angle 240.3154 will appear after you enter the next RADIUS command.

```
43  RADIUS    -.2 type1 x6.7 y-3.3
44  LINE      angle180
45  RADIUS    -.2 type2 x4.25 y-3.5
46  LINE      angle90
47  RADIUS    -.2 type0
48  LINE      angle45
49  RADIUS    .43 type1 x5.445 y-.43
50  LINE      angle180
51  RADIUS    .2 type2 x1.125 y0
52  LINE      angle270
53  RADIUS    -.2 type2 x1.125 y-1
54  LINE      angle180
55  POINT     x0 y-1
56  UNCOMP    angle180
57  POINT     x-.2 y-.8 f0
58  STOCK     xystk0 zstk0
```



5 Reverse cutter path

```
59   TOOL  3   dia.375 flutes2 type1=CARBIDE MILL rad0 ***REVERSE CUTTER
               PATH
60  MILL       zrapid.02 zcut-.115 passes1 zret.1 zf9 xyf26.6
61  POINT      x-.2 y-.8
62  COMP       angle0 cl/con1 lookahead0
63  REPEAT     from55 thru15 ***
64  UNCOMP     angle90
65  POINT      x-.2 y-1 f0
```



Since we are programming a reverse cutter path (opposite direction from the previous step), note these changes:

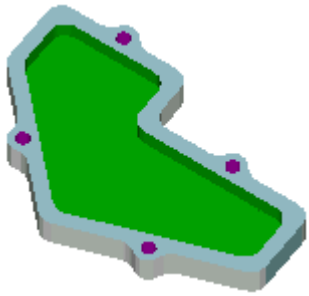
The approach point in line 61 is the point used as retract point in line 57.

COMP command uses climb cutting (cl/con=1) instead of conventional cutting (cl/con=2) used in the previous step. The COMP start angle also needs to be diffeent (0º) because the tool will be moving in the opposite direction.

Reversing the repeated order of the source lines describing the contour forces G-ZERO to create a cutter path in reverse order. (We are repeating the lines from the point before UNCOMP to the point after COMP) ■

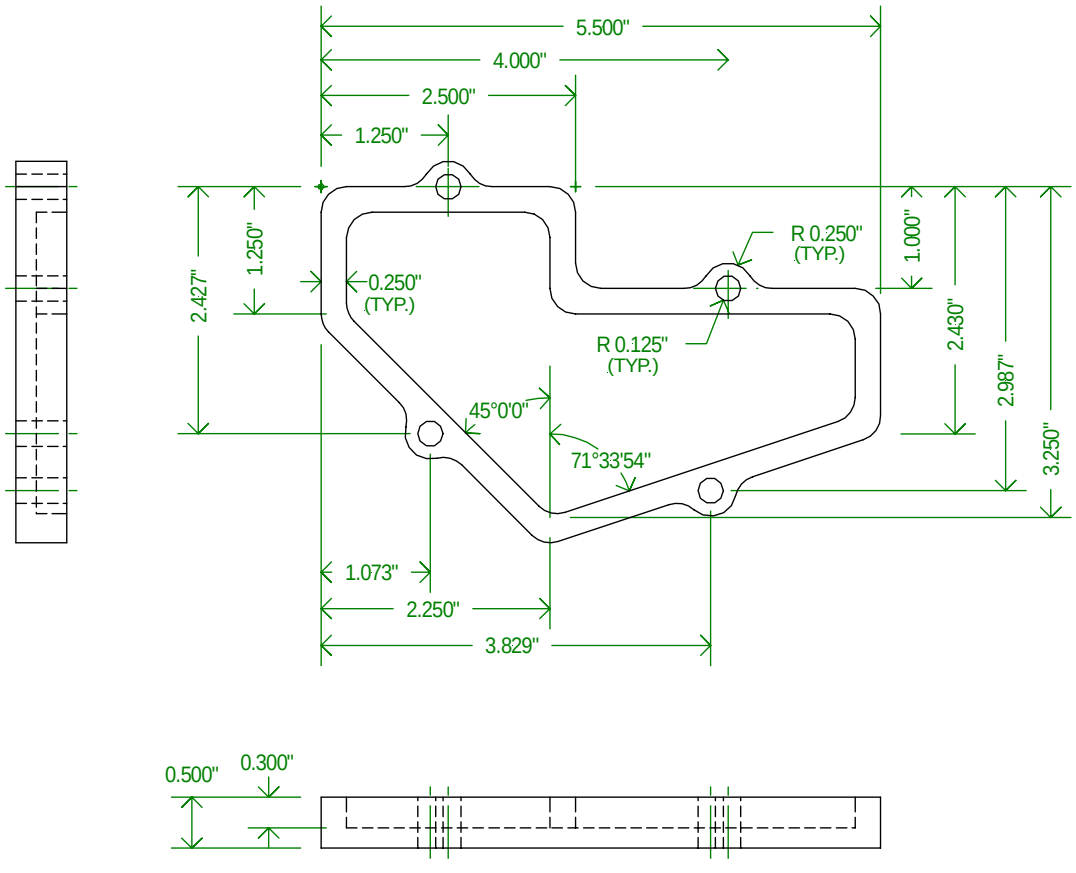
Project 7

C-Gasket



What you will learn:

- Load CAD Reader and open DXF file
- Prepare DXF file to import to source program (window, layer, zoom, origin)
- Define Material using values from DXF file
- Drill holes using CAD Reader Single Pick
- Mill pocket using CAD Reader Block Pick
- Cut profile using CAD Reader Block Pick



Rapid Output Co. www.g-zero.com 888-656-1945	TITLE: C-Gasket		REFERENCES: All fillets & rounds 0.25"R
	MATERIAL: 0.5" Aluminum		
	DRAWING: M010	TITLE: MC	

If you have a DXF file (or DWG, or VCD, or GCD), you can get the data for your geometry directly from the DXF file rather than digging into your blueprint.

The first time you press F9, G-ZERO Mill loads CAD Reader (G-ZERO CAD Import Interface) and allows you to select the DXF file you want to load. Every subsequent time you press F9, G-ZERO Mill will close or reopen the CAD Reader.

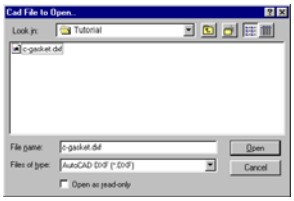
Note 1: In order to complete this project, you need the file C-Gasket.dxf (located in the C:\MILL\Tutorial directory).

Note 2: In this project, regular source codes are shown in blue, and values/codes added from CAD Reader are shown in *green*.

1 Load CAD Reader and open DXF file

1 *CAD C:\MILL\TUTORIAL\c-gasket.dxf*

After you start G-ZERO and open a new file (example: Project-7), press F9 (Tools | Cad F9) to open the C-Gasket.dxf file and load G-ZERO CAD Reader.



After a few seconds, your DXF file will be loaded on the CAD Reader (G-ZERO CAD Import Interface) window.

Notice that a line is added into your source file to establish a link to the DXF file.

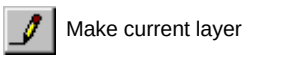
2 Prepare DXF file

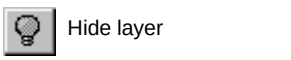
WINDOW. Like any window, you can adjust the size and location of your CAD Reader window by dragging its edges. If you want (if the size of your monitor permits), you can place the CAD Reader window next to the G-ZERO CAM programming window. In this case, you can just click on the window you want to activate without the need of pressing F9 to open or close the CAD Reader window.

LAYER. Your DXF file is opened with all its layers.

1. Click the *Toggle Display of Layers* button from the toolbar to display the *Layer Mgr.* window.

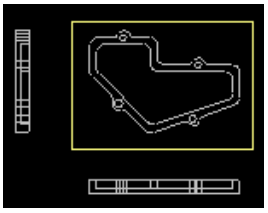
2. Make sure that the Short List check box (located at the bottom of the Layer Mgr. window) is selected. At this point, 3 layers should be displayed: BORDER, DIMENSION and DRAWING.
3. Since we only need the DRAWING layer, we can make it the "current" layer and hide the rest.
   Select the DRAWING layer and click the pencil icon to make it current.
   Select the DIMENSION layer and click the gray light bulb to hide it.
   Select the BORDER layer and click the gray light bulb to hide it.

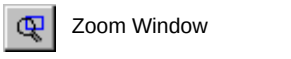




4. You can now close the Layer Mgr. window by clicking on the *Toggle Display of Layers* button again.

ZOOM. Since we are going to work on the top view, click the Zoom Window button and create a yellow box around the area you want to display (example: click the upper left corner of the top view, hold and drag the mouse to the lower right corner of the top view to create the yellow box around it, and release your mouse)

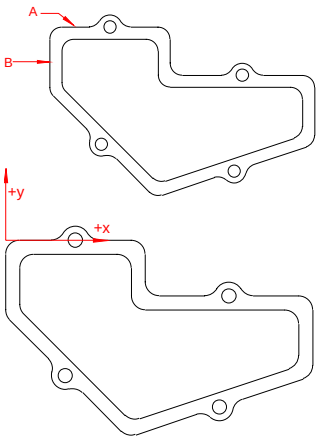




ORIGIN. It is very important to make sure that the coordinates of the DXF file match the coordinates of the source program. We are going to use in both cases the intersection of line A and line B as the origin of coordinates.

1. Press F2 (make sure that the CAD Reader window is currently active).
2. Click line A to pick the horizontal line that contains the origin.
3. Click line B to pick the vertical line that contains the origin.

You will see the new coordinates displayed with its origin in the intersection of lines A and B.

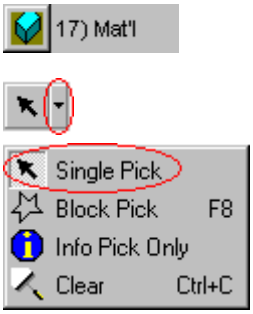


Note 1: CAD Reader will display your entire drawing; if you need to zoom in a section, use ZOOM WINDOW as described in the ZOOM section on the previous page.

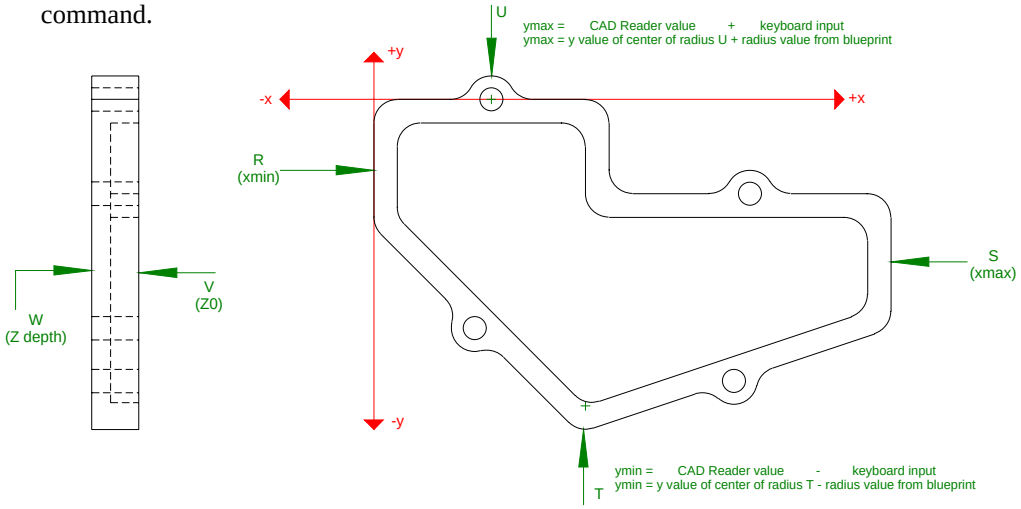
Note 2: The new coordinates are shown in a new layer (1023). If you want to hide it, select layer 1023 and click the gray light bulb to hide it (see LAYER section on the previous page).

3 Define material using values from DXF file

2 MAT'L xmin*0* xmax*5.5* ymin-*3.5* ymax.*25* thk.*5* type0=ALUMALOY



1. Start the MAT'L command.
2. Click the display options button next to the Pick button to list all the Pick choices.
3. Select Single Pick option.
4. For xmin: click line R and press the Enter key.
5. For xmax: click line S and press the Enter key.
6. For ymin: click radius T (y value of the center of radius) add the negative value of the radius (- .25) *(See Reference in blueprint)* and press the Enter key.
7. For ymax: click radius U (y value of the center of radius) add the positive value of the radius (+ .25) *(See Reference in blueprint)* and press the Enter key.
8. For thk: Select Single Pick
   click line V for the first line for Z0
   click line W for the Z depth
   and press the Enter key.
9. Select the material type from the pop-up window to finish the MAT'L command.

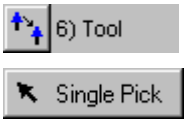


4 Drill holes using CAD Reader Single Pick

3 TOOL 1 dia.25 flutes2 type20=HSS DRILL rad0 *** DRILL HOLES

The diameter of this tool will be the diameter of the hole we are going to drill. Therefore, we can get this value from the DXF file.

1. Start the TOOL command, and enter 1 for the tool number.
2. Select Single Pick.
3. Click any circle to get its diameter and press the Enter key.
4. Program the rest of the TOOL command as you normally do.



4 DRILL g83=PECK zrap.1 zcut-.5 pecks2 tip1 zret.1 f27.8

In this DRILL command, we can get the zcut value from the DXF file.

1. Start programming the DRILL command as you normally would.
2. For zcut: select Single Pick,
   click line V for the first line for Z0 (See graphic on Mat'l section),
   click line W for the Z depth,
   and press the Enter key.
3. Program the rest of the DRILL command as you normally do.

5 POINT x1.25 y0

To define the center of the holes to drill, we are going to use the Single Pick option and select the appropriate circles in the CAD Reader window.

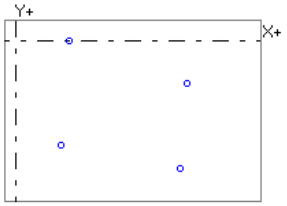
1. Select the Single Pick option.
2. Click the first circle (upper left) and see how line 5 is added into your source program. Notice that the element you select is displayed in green so you can confirm that the correct object was picked.

6 POINT x4 y-1

7 POINT x3.8291 y-2.9872

8 POINT x1.0732 y-2.4268

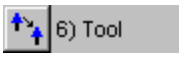
Since you are still using the Single Pick mode, now click/pick the other three circles to add new lines in your source program to define the center of the holes to be drilled.



5 Mill pocket using CAD Reader Block Pick

9 TOOL 2 dia.375 flutes2 type1=CARBIDE MIL rad0 *** MILL POCKET

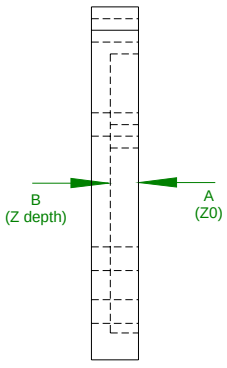
Program the TOOL command as you normally would.



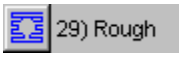
10 MILL zrapid.1 zcut-.3 passes1 zret.1 zf33.9 xyf14.5

In this MILL command, we can get the zcut value from the DXF file.

1. Start programming the MILL command as you normally would.
2. For zcut: Select Single Pick,
   click line A for the first line for Z0,
   click line B for the Z depth,
   and press the Enter key.
3. Program the rest of the MILL command as you normally would.



11 ROUGH stk.02 stp80 angle270 cleanup1



Program the ROUGH commands as you normally do.

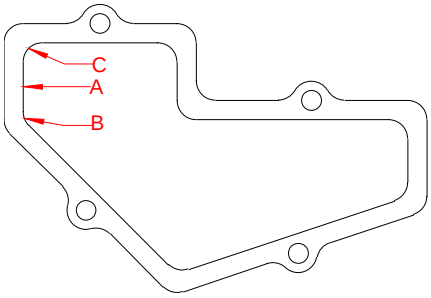
12 COMP angle270 cl/con41 lookahead0
13 POINT x.25 y-.5
14 POINT x.25 y-1.1464
15 RADIUS -.25 type1 x.5 y-1.1464
16 POINT x-.3232 y-1.3232
17 POINT x2.1407 y-3.1407
18 RADIUS -.25 type1 x2.3175 y-2.964
19 RADIUS -.25 type1 x5 y-2.0698
20 RADIUS -.25 type1 x.5 y-1.5
21 RADIUS .25 type1 x2.5 y-1
22 RADIUS -.25 type1 x2 y-.5
23 RADIUS -.25 type1 x.5 y-.5
24 UNCOMP angle270

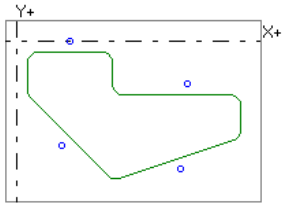
To define the boundary of the pocket to rough, we are going to use the Block Pick option, where we only select the first, second, and last elements of the boundary. (Note that all lines and radii need to be physically connected in the DXF file)

1. Click the display options button next to the Pick button to list all the Pick choices.
2. Select the Block Pick option.

3. Click line A as the first element of the block.
4. Click radius B as the second element of the block. This second element defines the direction of your cut, whether it will be climb cut or conventional cut.
5. Click radius C as the last element of the block.

Now watch how all elements of the block are sequencially selected in the CAD Reader window and displayed in the viewport, and lines 12 to 24 are added to your source program including the COMP and UNCOMP commands.



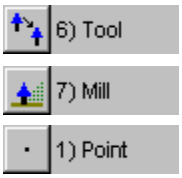


25 ROUGH stk.02 stp80 angle270 cleanup1

To finish roughing the pocket, program the ending ROUGH command identical to the initial ROUGH in line 11.

6 Cut profile using CAD Reader Block Pick

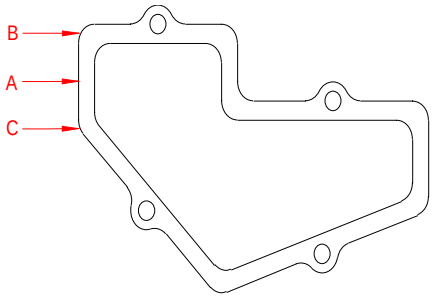
```
26   TOOL  3    dia.5 flutes3 type1=CARBIDE MILL rad0 *** MILL PROFILE
27   MILL       zrapid.1 zcut-.52 passes1 zret.1 zf20 xyf10
28   POINT      x-.375 y-1.25
```



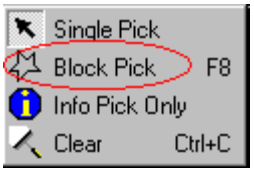
Program the TOOL and MILL commands as you normally do.

Also program an approach point.

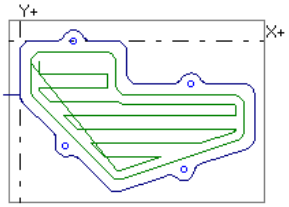
```
29   COMP       angle90 cl/con41 lookahead0
30   POINT      x0 y-1.25
31   POINT      x0 y-.25
32   RADIUS     .25 type1 x.25 y-.25
33   RADIUS     -.25 type1 x.817 y.25
34   RADIUS     .25 type1 x1.25 y0
35   RADIUS     -.25 type1 x1.683 y-.25
36   RADIUS     .25 type1 x2.25 y-.25
37   RADIUS     -.25 type1 x2.75 y-.75
38   RADIUS     -.25 type1 x3.567 y-.75
39   RADIUS     .25 type1 x4 y-1
40   RADIUS     -.25 type1 x4.433 y-.75
41   RADIUS     .25 type1 x5.25 y-1.25
42   RADIUS     .25 type1 x5.25 y-2.25
43   RADIUS     -.25 type1 x4.3189 y-3.0874
44   RADIUS     .25 type1 x3.8291 y-2.9872
45   RADIUS     -.25 type1 x3.4973 y-3.3613
46   RADIUS     .25 type1 x2.25 y-3.25
47   RADIUS     -.25 type1 x1.2026 y-2.9097
48   RADIUS     .25 type1 x1.0732 y-2.4268
49   RADIUS     -.25 type1 x.5903 y-2.2974
50   RADIUS     .25 type1 x.25 y-1.25
51   UNCOMP     angle90
```



To define the boundary of the profile, we are going to use the Block Pick option, where we only select the first, second, and last elements of the boundary. (Note that all lines and radii need to be physically connected in the DXF file)



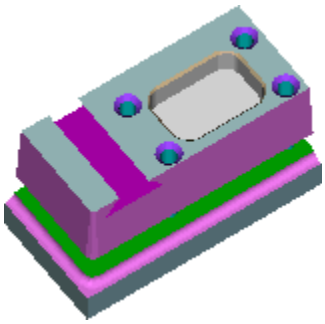
1. Click the display options button next to the Pick button to list all the Pick choices.
2. Select the Block Pick option.
3. Click line A as the first element of the block.
4. Click radius B as the second element of the block. This second element defines the direction of your cut, whether it will be climb cut or conventional cut.
5. Click radius C as the last element of the block.



Now watch how all elements of the block are sequencially selected in the CAD Reader window and displayed in the viewport, and lines 29 to 51 are added to your source program including the COMP and UNCOMP commands. ■

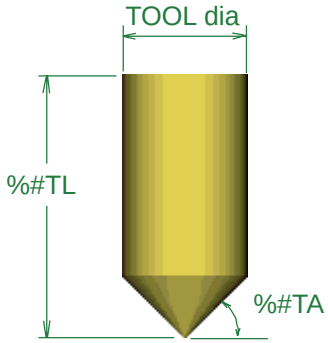
Project 8

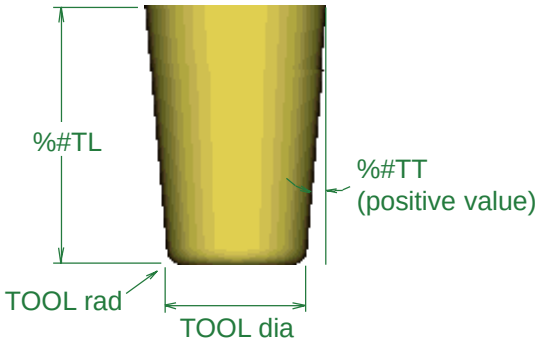
ToolType

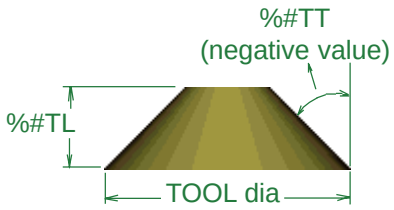


What you will learn:

- Use of the %#TL, %#TT, and %#TA variables to show accurate modeling of specialized tools in the Solid Modeling environment







Project 1 (Initial Tour) showed you how to use F6 to display a solid view of your current source program. However, if you are using specialized tools (such as tapered tools), the tools' special shapes are not taken into consideration in the viewport unless you use the correct tool type variables.

These variables (or reserved words) are %#TL, %#TT, and %#TA and should be used in any place inside the comment section of the TOOL commands. In other words, they can be placed before, in the middle, or after your normal tool comments.

%#TL defines the length (in inches) of the tool to be displayed in the viewport.

%#TT defines the side cutting edge angle (angle between the side cutting edge and a plane that is parallel to the side of the shank). In the case of a dovetail cutter, this angle should be negative.

%#TA defines the end cutting edge angle (angle made by the end cutting edge with respect to a plane perpendicular to the axis of the tool shank.

1 Program Mat'l

1 MAT'L xmin0 xmax2 ymin0 ymax1 thk1 type0=ALUMALOY

Program a MAT'L command using the values provided here.

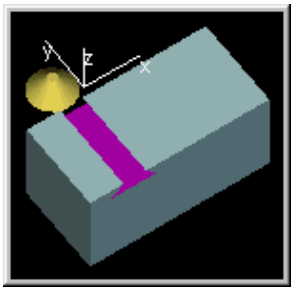
2 Tool 1: 45° Dovetail -- %#TL, %#TT

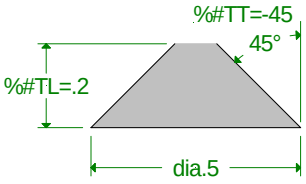
2 TOOL 1 dia.5 flutes4 type1=CARBIDE MILL rad0 *** %#TL=.2 %#TT=-45 DOVETAIL

3 MILL zrapid.1 zcut-.125 passes1 zret.1 zf30 xyf30

4 POINT x.5 y-.3

5 POINT x.5 y1.3 f10





Note that the diameter of the tool (0.5) is entered as you normally do in the TOOL command when it asks: "What is the diameter of this tool?"

Use %#TL in the comment section of the TOOL command to set the flute length of the tool.

Use %#TT in the comment section of the TOOL command to set the tool taper, which is the angle between the side cutting edge and a plane that is parallel to the side of the shank. Note that, in this case, the tool taper angle is negative because it is a dovetail cutter.

3 Tool 2: 3/16 thick Woodruff -- %#TL

6 TOOL 2 dia1 flutes12 type1=CARBIDE MILL rad0 *** %#TL=.1875 WOODRUFF

7 MILL zrapid.1 zcut-.6875 passes1 zret.1 zf30 xyf30

8 POINT x2.5 y-.6

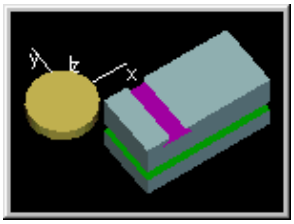
9 COMP angle180 cl/con41 lookahead0

10 POINT x2.5 y.25

11 RADIUS .125 type2 x.25 y.25

12 POINT x.25 y1.1

13 UNCOMP angle0



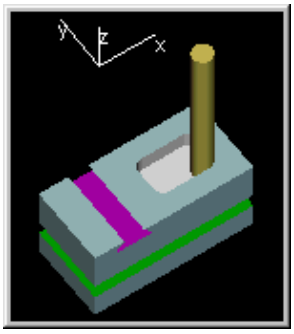
Note that the diameter of the tool is entered as you normally do in the TOOL command, when it asks: "What is the diameter of this tool?"

To setup the correct display of a 3/16 thick woodruff, convert the fraction into decimal (.1875), and enter %#TL=.1875 in the comment section of the TOOL command.

Note that the tool taper defaults to zero, so it is not required to use %#TT.

4 Tool 3: Use default tool length

```
14   TOOL  3    dia.25 flutes2 type1=CARBIDE MILL rad0 *** NOT USING ANY TOOL VARIABLES
15   MILL       zrapid.1 zcut-.05 passes1 zret.1 zf30 xyf30
16   POINT      x1.5 y.5
17   Z MOVE     z-.2 ramp1 f10
18   RECT       xmin1 xmax1.75 ymin.25 ymax.75 thru0
```



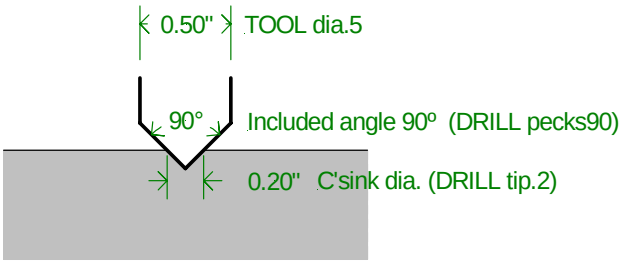
When the tool you are using does not need to show specific tool length or tool taper, you don't need to use any tool variables.

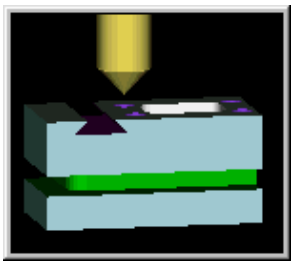
The default tool length shown in the viewport is 1.

5 Tool 4: Countersink

```
19   TOOL  4    dia.5 flutes1 type30=C'SINK rad0 *** .2 DIA x 90 DEG C'SINK
20   DRILL      g82=C'SINK zrap.1 zcut0 pecks90 tip.2 zret.1 f9.2
21   GRID       num4 xnum2 x.875 xstp1 y.25 ystp.5
```

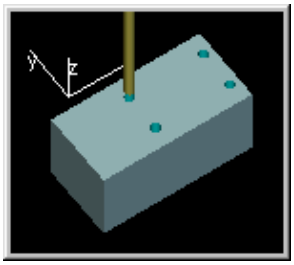
When a countersink tool is used, the included angle and the depth of the countersink is determined by the pecks and tip values of the DRILL command.





6 Tool 5: Thru holes

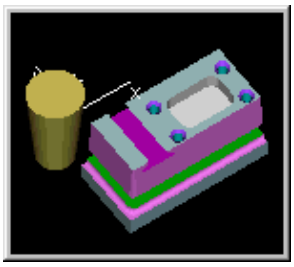
```
22   TOOL  5    dia.125 flutes2 type20=HSS DRILL rad0 *** THRU HOLES
23   DRILL      g81=DRILL zrap.1 zcut-1 pecks1 tip1 zret.1 f24.4
24   REPEAT     from21 thru21 ***
```

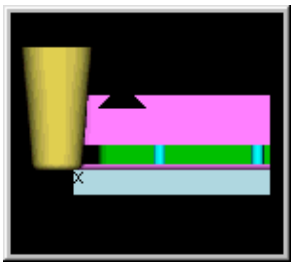


Notice that the DRILL command overwrites the variable %#TA with its own angle. If your drill has a non-standard tip angle, enter its angle in the TIP section of the DRILL command.

7 Tool 6: Tapered Endmill

```
25   TOOL  6    dia.5 flutes4 type1=CARBIDE MILL rad.0625 *** %#TL=1.25  %#TT=5
26   MILL       zrapid.1 zcut-.75 passes1 zret.1 zf20 xyf10
27   POINT      x2.05 y-.3
28   COMP       angle180 cl/con41 lookahead0
29   POINT      x2.05 y.075
30   RADIUS     .05 type2 x.075 y.075
31   POINT      x.075 y1.05
32   UNCOMP     angle0
```

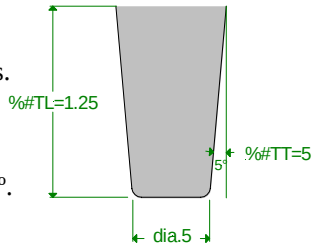




We can improve the way a tapered endmill looks in a viewport using tool type variables.

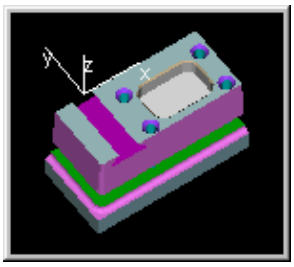
To setup a tool length of 1.25”, use %#TL=1.25.

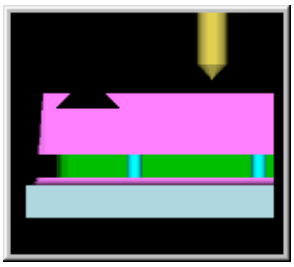
Use %#TT=5 to set the tool taper angle to 5º.



8 Tool 7: Deburr pocket

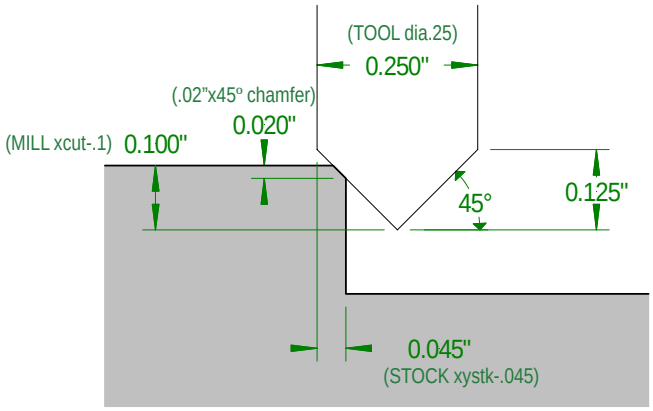
```
33   TOOL  7    dia.25 flutes1 type30=C’SINK rad0 *** DEBURR POCKET WITH 1/4” CSINK  %#TA=45
34   STOCK      xystk-.045 zstk0
35   MILL       zrapid.1 zcut-.1 passes1 zret.1 zf20 xyf10
36   COMP       angle0 cl/con41 lookahead0
37   POINT      x1.5 y.25
38   RADIUS     -.125 type2 x1.75 y.25
39   RADIUS     -.125 type2 x1.75 y.75
40   RADIUS     -.125 type2 x1 y.75
41   RADIUS     -.125 type2 x1 y.25
42   POINT      x1.6 y.25
43   UNCOMP     angle0
```



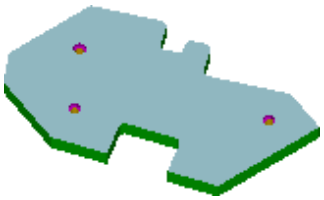


To achieve a .02”x45º chamfer, we can add a negative STOCK to the tool (-.045) to make contact with the part.

Use %#TA=45 to display a 45º chamfer tool.



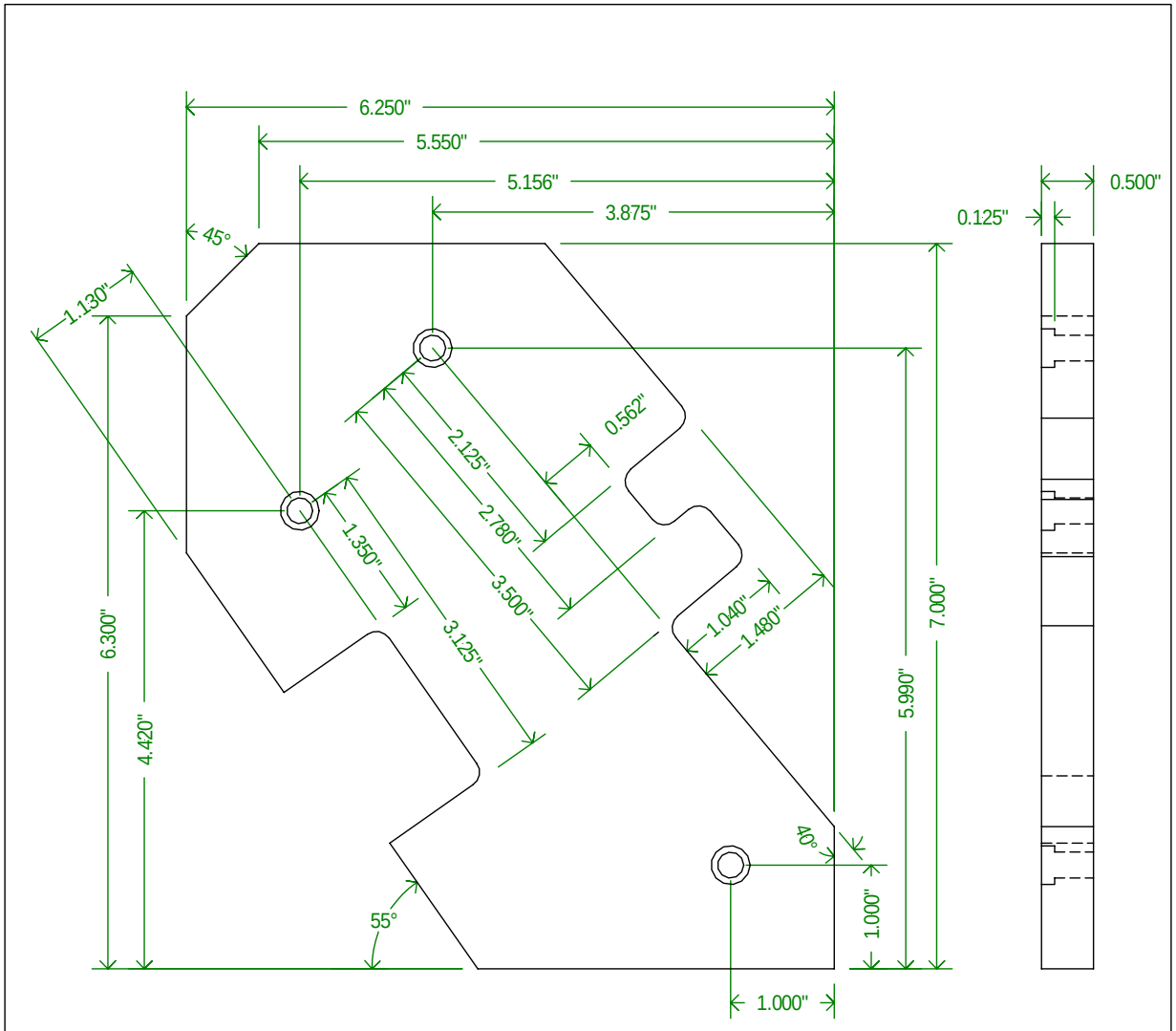
Project 9



Sideways Face

What you will learn:

- Use ROTATE to program geometry with dimension given in a different axis (rotated and shifted coordinates).



Rapid Output Co. www.g-zero.com 888-656-1945 G-ZERO CAM Software	TITLE: Sideways Face MATERIAL: 0.5" Nickel Alloy 4140 DRAWING: M005 INITIALS: MC	REFERENCES: 1. Break all corners 0.005" R 2. All radii = 0.15" R 3. All holes .25" dia. thru with counterbore 0.375" dia. x 0.125" deep 4. All measurements in inches (n.t.s.)	RECOMMENDED OP & TOOL LIST: 1. Drill holes (.25" dia. stub drill) 2. Counterbores (.25" dia. carbide endmill) 3. Rough and Finish profile (.25" dia. 4-flute carbide endmill)

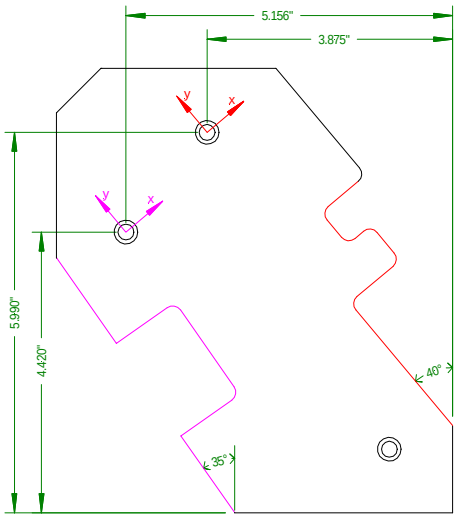
1 Drill Holes - Counterbore Holes


```
1   MAT'L      xmin-7 xmax.5 ymin-.5 ymax7.5 thk.5 type3=4140
2     TOOL  1  dia.25 flutes2 type20=HSS DRILL rad0 *** STUB-DRILL HOLES
3   DRILL      g83=PECK zrap.02 zcut-.5 pecks3 tip1 zret.02 f4
4   POINT      x-1 y1
5   POINT      x-5.156 y4.42
6   POINT      x-3.875 y5.99
7     TOOL  2  dia.25 flutes2 type1=CARBIDE MILL rad0 *** COUNTERBORE
                HOLES
8   MILL       zrapid.02 zcut-.125 passes1 zret.1 zf9.2 xyf4.6
9   ROUND      dia-.375 x-1 y1 thru0
10  MILL       zrapid.02 zcut-.125 passes1 zret.1 zf9.2 xyf4.6
11  ROUND      dia-.375 x-5.156 y4.42 thru0
12  MILL       zrapid.02 zcut-.125 passes1 zret.1 zf9.2 xyf4.6
13  ROUND      dia-.375 x-3.875 y5.99 thru0
```

Use information from blueprint to program the MAT'L command.

Program the three holes with a DRILL and three POINT commands. To counterbore, use the MILL with ROUND commands instead of Drill and Repeat-Point commands.

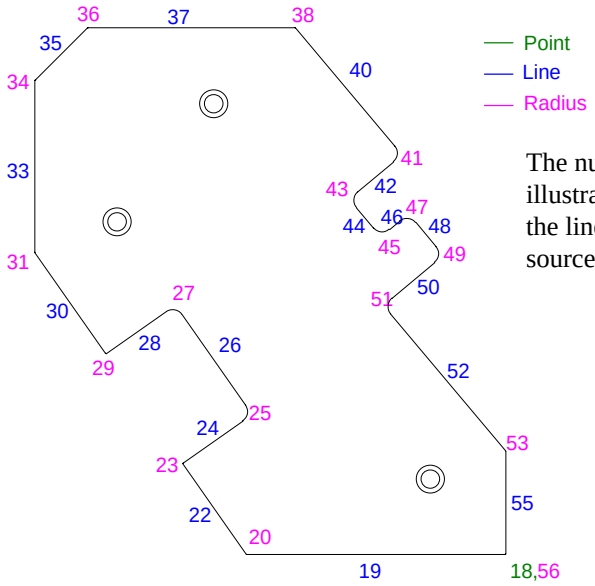
2 Mill Profile with different coordinates references



If you take a closer look at the blueprint on the first page of this project, you will find out that the dimensions are given according to three different coordinates:

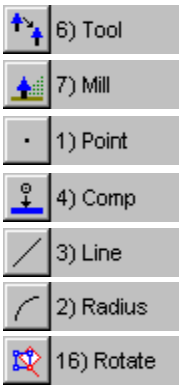
1. Lines and arcs in BLACK shown above: center of coordinates in lower right corner of part.
2. Lines and arcs in RED shown above: center of coordinates rotated 40º, shifted -3.875" in x, and shifted 5.99" in y in relation to the first coordinates (lower right corner of the part).
3. Lines and arcs in MAGENTA shown above: center of coordinates rotated 35º, shifted -5.156" in x, and shifted 4.42" in y in relation to the first coordinates (lower right corner of the part).

We are going to use the first coordinates (lower right corner of part) as the base to program the profile milling. As we reach the lines and arcs with dimensions based on a different coordinates, we program the ROTATE command to make up for its difference.



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

```
14   TOOL  3    dia.25 flutes2 type1=CARBIDE MILL rad0 *** PROFILE
15   MILL       zrapid.02 zcut-.51 passes3 zret.1 zf9.2 xyf4.6
16   POINT      x.5 y-.5
17   COMP       angle180 cl/con1 lookahead0
18   POINT      x0 y0
19   LINE       angle180
20   RADIUS     .005 type0
```



Program an approach point at (.5;-.5) before the COMP command. We are going to mill the profile using climb cut starting from the lower right corner of the part going in a counter-clockwise direction. Our center of coordinates will be the lower right corner of the part

```
21   ROTATE     angle35 xpiv-5.156 ypiv4.42
22   LINE       angle90
23   RADIUS     .005 type2 x-1.13 y-3.125
24   LINE       angle0
25   RADIUS     -.15 type2 x0 y-3.125
26   LINE       angle90
27   RADIUS     -.15 type2 x0 y-1.35
28   LINE       angle180
29   RADIUS     .005 type2 x-1.13 y-1.35
30   LINE       angle90
31   RADIUS     .005 type0
32   ROTATE     angle0 xpiv0 ypiv0
```

Dimensions of elements (lines and arcs) shown in magenta on the previous page are based on a coordinates shifted -5.156 in the x axis, shifted 4.42 in the y axis, and rotated 35° from the center of coordinates (lower right corner of part).

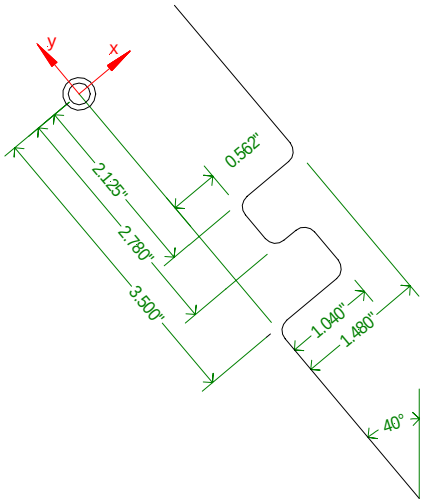
Using this information, program the ROTATE command in line 21. The next ten lines and radii are going to be referred to this new (rotated and shifted) coordinates. In this way, we can enter the dimensions the way it appears on the blue print, and let G-ZERO do the calculations to make up for the shifted and rotated values.

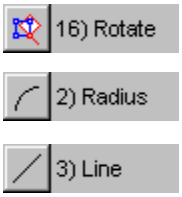
Make sure to program a ROTATE angle0 xpiv0 ypiv0 to tell G-ZERO to turn off ROTATE and go back to the original coordinates.

33 LINE angle 90
34 RADIUS .005 type2 x-6.25 y6.3
35 LINE angle45
36 RADIUS .005 type2 x-5.55 y7
37 LINE angle0
38 RADIUS .005 type0

The dimensions of the next 6 elements (lines and arcs) are referred to the original coordinates (lower right corner of part). So, continue programming them as you would normally do.

39 ROTATE angle40 xpiv-3.875 ypiv5.99
40 LINE angle270
41 RADIUS .15 type2 x1.48 y-2.125
42 LINE angle180
43 RADIUS -.15 type2 x.562 y-2.125
44 LINE angle270
45 RADIUS -.15 type2 x.562 y-2.78
46 LINE angle0
47 RADIUS .15 type2 x1.04 y-2.78
48 LINE angle270
49 RADIUS .15 type2 x1.04 y-3.5
50 LINE angle180
51 RADIUS -.15 type2 x0 y-3.5
52 LINE angle270
53 RADIUS .005 type0
54 ROTATE angle0 xpiv0 ypiv0



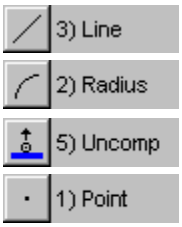


Dimensions of elements (lines and arcs) shown in red in the beginning of this section are based on a coordinates shifted -3.875 in the x axis, shifted 5.99 in the y axis, and rotated 40º from the center of coordinates (lower right corner of part).

Using this information, program the ROTATE command in line 39. The next 14 lines and radii are going to be referred to this new (rotated and shifted) coordinates.

When done, make sure to program a ROTATE angle0 xpiv0 ypiv0 to tell G-ZERO to turn off ROTATE and go back to the original coordinates.

55 LINE angle270
56 RADIUS .005 type2 x0 y0
57 UNCOMP angle180
58 POINT x-.2 y-.5 f0



Finish programming the profile with a LINE and RADIUS commands using the original coordinates.

The UNCOMP angle 180º is the tangent angle from the final RADIUS (line 56). As the tool cuts around the outside of the RADIUS, it last touches the part as it faces left (180º).

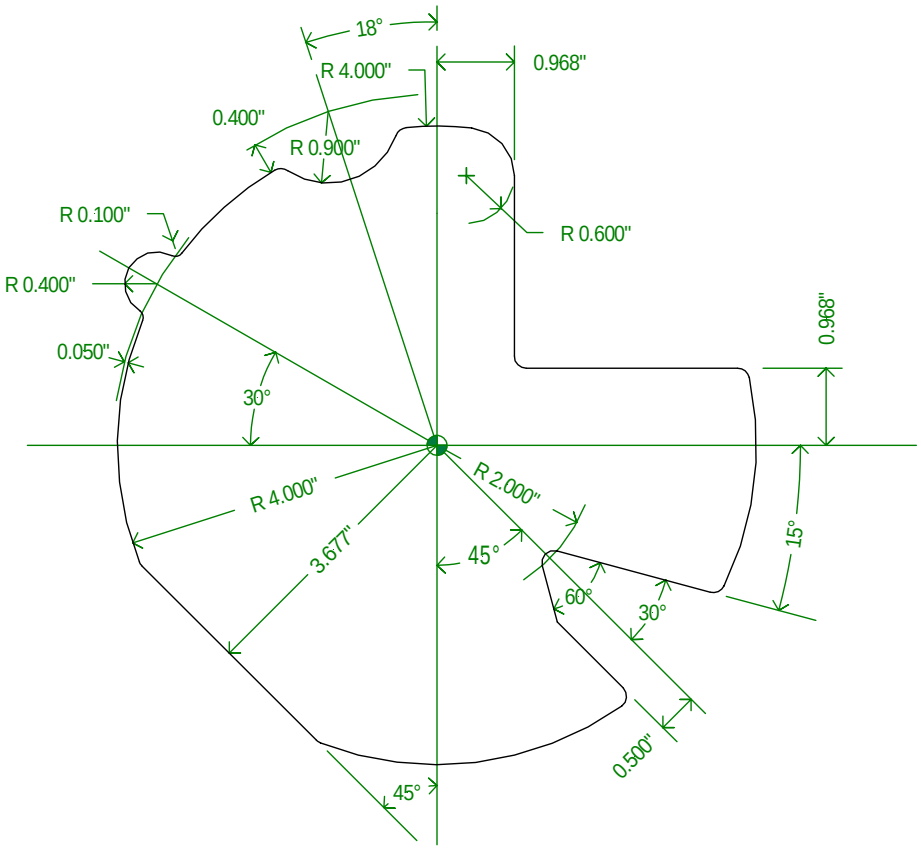
Program a retract point just off the part so the cutter pulls off the part without leaving a dwell mark. ■

Project 10

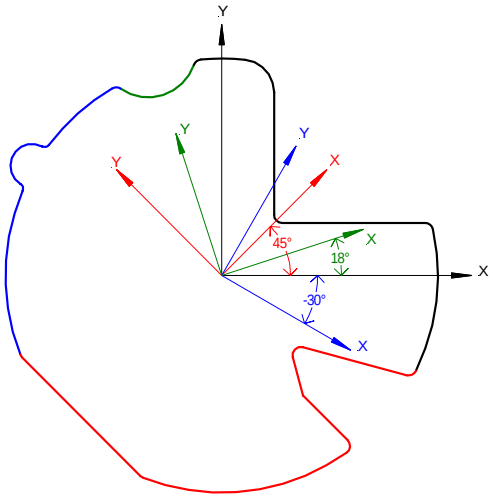
Odd Pizza

What you will learn:

- Use several ROTATE commands to program geometry with dimensions given in different axes (rotated coordinates).
- Use of blend-on and blend-off radii.

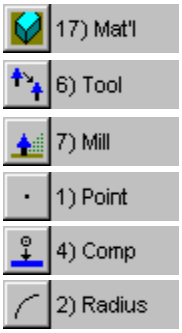


Rapid Output Co. www.g-zero.com 888-656-1945	TITLE: Odd Pizza	REFERENCES: 1. All measurements in inches (n.t.s.) 2. All undimensioned radii = 0.150" 3. 0.1"R (2 places)	RECOMMENDED OP & TOOL LIST: Rough and Finish Profile (.187" dia. endmill)
	MATERIAL: 0.125" Alumaloy		
	DRAWING: M006 INITIALS: MC		



-- Original coordinates
-- Coordinates rotated 45º
-- Coordinates rotated -30º
-- Coordinates rotated 18º

1 Original coordinates



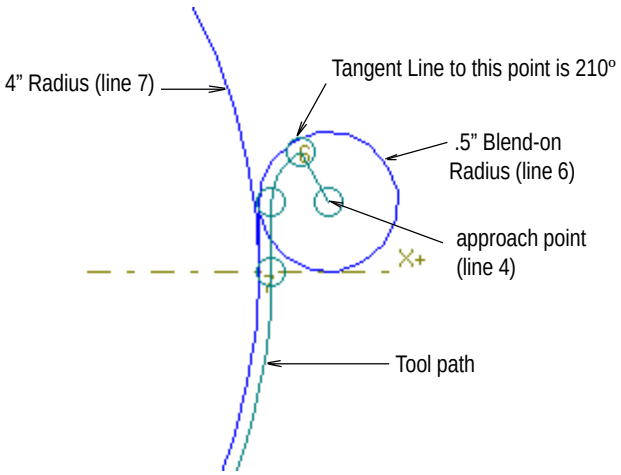

```
1    MAT'L      xmin-5 xmax5 ymin-5 ymax5 thk-.125 type0=ALUMALOY
2     TOOL  1   dia.187 flutes2 type0=HSS MILL rad0 *** PROFILE CUT
3    MILL       zrapid.02 zcut-.13 passes1 zret.1 zf32.9 xyf11.4
4    POINT      x4.5 y.5
5    COMP       angle210 cl/con1 lookahead0
6    RADIUS     -.5 type1 x4.5 y.5
7    RADIUS     4 type1 x0 y0
```

Round Material. Notice that we used a "negative" thickness in the MAT'L command. This is because we want the viewport to show a round material.

Program an "approach Point" just off the part to drop the cutter in a safe location (e.g.: x=4.5, y=.5).

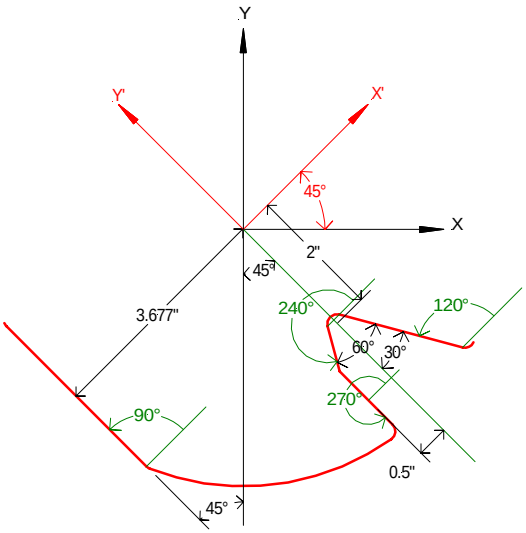
Blend-on Radius. We are programming a "blend-on" Radius (line 6) to ensure that the tool does not leave a dwell mark. A blend-on Radius is not part of the contour. Program a COMP angle of 210º so that as the tool starts cutting the Radius (line 6), it will begin cutting as it moves from 210º tangent position. It will cut around the inside of the blend-on Radius until the next programmed geometry (line 7).

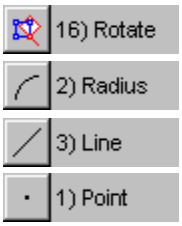
Changing the value of the COMP angle when using a blend-on Radius determines where the tool starts on the blend-on Radius. Try it and see.



2 Coordinates rotated 45º

8	ROTATE	angle45 xpiv0 ypiv0
9	RADIUS	.15 type0
10	LINE	angle120
11	RADIUS	-.15 type1 x0 y-2
12	LINE	angle240
13	RADIUS	-.15 type0
14	LINE	angle270
15	POINT	x-.5 y-3.75
16	LINE	angle270
17	RADIUS	.15 type0
18	RADIUS	4 type1 x0 y0
19	RADIUS	.15 type0
20	LINE	angle90
21	POINT	x-3.677 y0
22	LINE	angle90
23	RADIUS	.15 type0



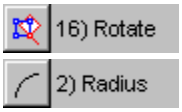


Rotate. This section of the contour (the cutout in the 4th quadrant and the flat in the 3rd) is dimensioned at a +45º angle to the XY plane.

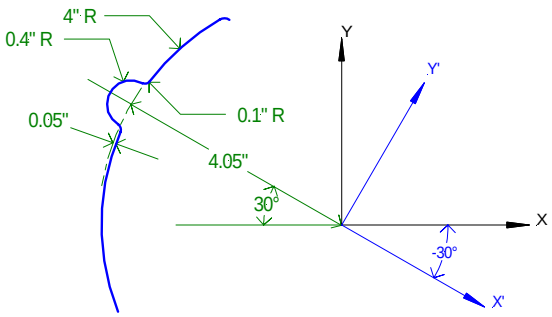
Line-Point-Line technique. Use this technique to let G-ZERO calculate the locations of the 2 unknown Radii separated by a Line (lines 14-16 and 20-22). Program any Point along the Line segment between the two unknown Radii.

3 Coordinates rotated -30º

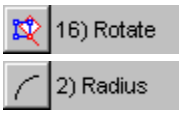
24	ROTATE	angle-30 xpiv0 ypiv0
25	RADIUS	4 type1 x0 y0
26	RADIUS	-.1 type0
27	RADIUS	.4 type1 x-4.05 y0
28	RADIUS	-.1 type0
29	RADIUS	4 type1x0 y0
30	RADIUS	.15 type0



Rotate. Line 24 cancels the active Rotate and begins a new Rotate command. To program the bump in the 2nd quadrant, Rotate all following dimensions -30º. Since the pivot point is x0 y0, the origin remains the same.

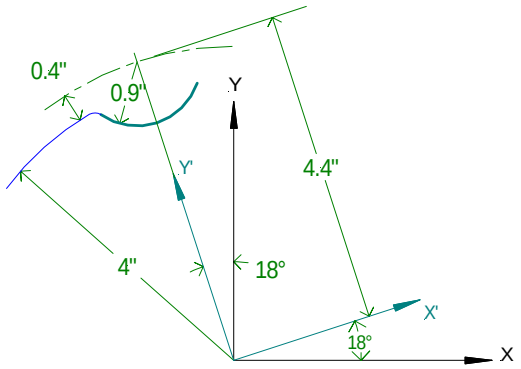


4 Coordinates rotated 18º

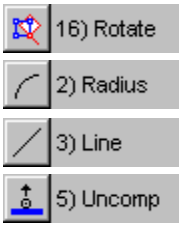


31 ROTATE angle18 xpiv0 ypiv0
32 RADIUS -.9 type1 x0 y4.4

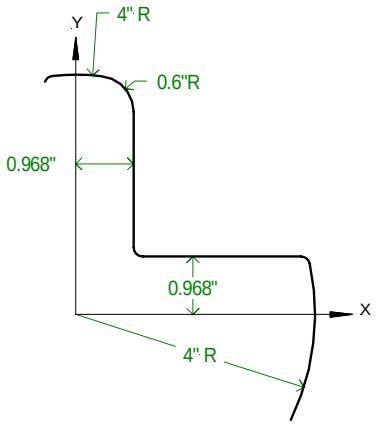
Rotate. Line 31 cancels the active rotate and starts a new Rotate command. To program the inside radius in the 2nd quadrant, Rotate the next radius 18º. Since the pivot point is x0 y0, the origin remains the same.



5 Back to Original Coordinates

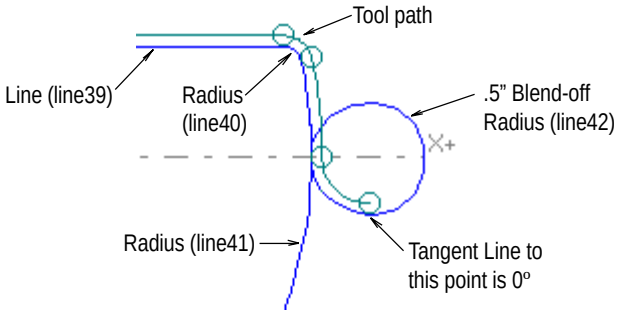


33 ROTATE angle0 xpiv0 ypiv0
34 RADIUS .15 type0
35 RADIUS 4 type1 x0 y0
36 RADIUS .6 type0
37 LINE angle270
38 RADIUS -.15 type2 x.968 y.968
39 LINE angle0
40 RADIUS .15 type0
41 RADIUS 4 type1 x0 y0
42 RADIUS -.5 type1 x4.5 y0
43 UNCOMP angle0



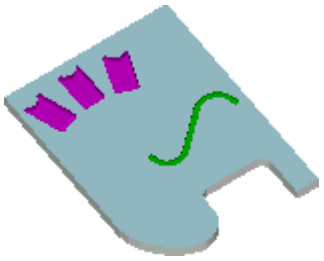
Rotate OFF. The rest of the dimensions are based on the original coordinates. Turn Rotate OFF (cancel) by entering zeros for all the questions.

Blend-off Radius. Radius command in line 42 is a blend-off radius (not on the blue print) and we program it to prevent the tool from leaving a dwell mark on the part. Un UNCOMP angle of 0º forces the cutter to travel a full quarter-arc on the blend-off radius.



Project II

Scroll

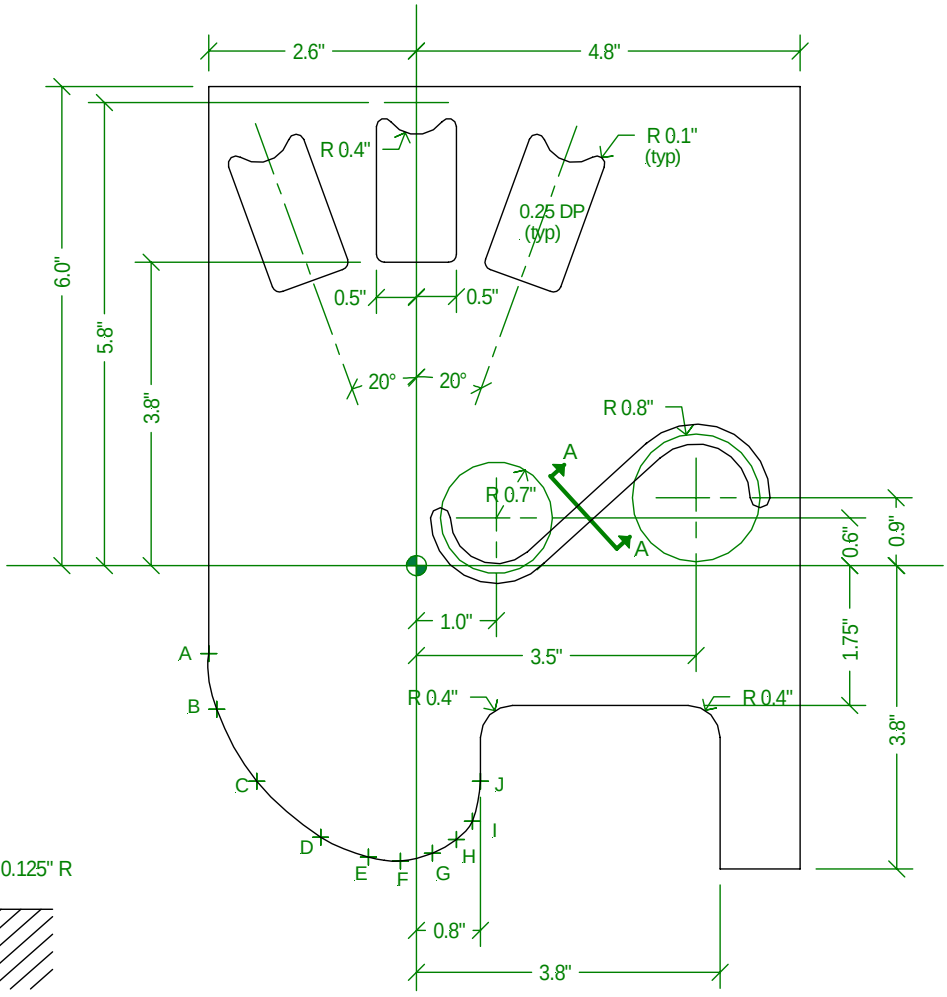


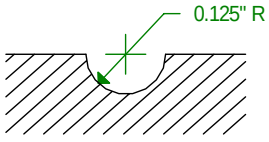
What you will learn:

- Define a geometry for subsequent multiple use (no cutting performed).
- Cut on a centerline.
- Smoothing (smooth points).

Coordinates:

Point	X	Y
A	-2.6	-1.1
B	-2.5	-1.8
C	-2.0	-2.7
D	-1.2	-3.4
E	-0.6	-3.65
F	-0.2	-3.7
G	0.2	-3.6
H	0.5	-3.43
I	0.7	-3.2
J	0.8	-2.7



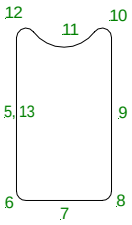


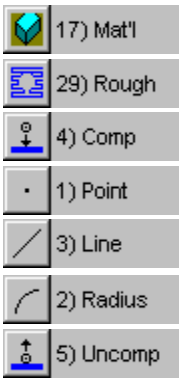
Detail A-A

Rapid Output Co. www.g-zero.com 888-656-1945	TITLE: Scroll	REFERENCES:	RECOMMENDED OP & TOOL LIST:
	MATERIAL: 0.5" Aluminum	1. All measurements in inches (n.t.s.) 2. Coordinates (see upper-left table) 3. Pockets 0.25" deep (3)	1. Pockets - cut left to right (.1875" dia. endmill) 2. Scroll (.25" ball endmill) 3. OD (.625" dia. endmill)
	DRAWING: M007 INITIALS: MC		

1 Define geometry

1	MAT'L	xmin-3 xmax5 ymin-4.5 ymax6.5 thk.5 type0=ALUMALOY
2	ROUGH	stk.005 stp60 angle180 cleanup1
3	COMP	angle270 cl/con1 lookahead0
4	POINT	x-.5 y4.8
5	LINE	angle270
6	RADIUS	-.1 type2 x-.5 y3.8
7	LINE	angle0
8	RADIUS	-.1 type2 x.5 y3.8
9	LINE	angle90
10	RADIUS	-.1 type0
11	RADIUS	.4 type1 x0 y5.8
12	RADIUS	-.1 type0
13	LINE	angle270
14	POINT	x-.5 y4.8
15	UNCOMP	angle270
16	ROUGH	stk.005 stp60 angle180 cleanup1





Program the middle pocket before the first TOOL command so that it will NOT generate G-code. (We'll later ROTATE and REPEAT these lines).

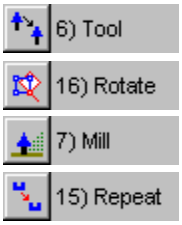
POINT on line 4 brings the tool down on the left wall of the pocket. The computer automatically calculates offsets for tool radius in COMP mode.

POINT on line 14 uses the same coordinates as line 4 to ensure that the cutter makes a complete pass around the pocket profile.

Note: numbers shown on graphic above, correspond to the source code line number that is programmed for that particular geometry.

2 Mill pockets using pre-defined geometry

17	TOOL 1	dia.1875 flutes2 type1=CARBIDE MILL rad0 *** POCKETS
18	ROTATE	angle20 xpiv0 ypiv0
19	MILL	zrapid.05 zcut-.25 passes2 zret.1 zf44.5 xyf15.5
20	REPEAT	from2 thru16 ***
21	ROTATE	angle0 xpiv0 ypiv0
22	REPEAT	from19 thru20 ***
23	ROTATE	angle-20 xpiv0 ypiv0
24	REPEAT	from19 thru20 ***



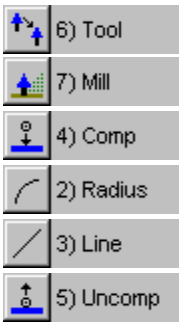
Left Pocket. To machine the left pocket, program a ROTATE +20º around the center of coordinates. Line 20 repeats the lines programmed before the first tool (lines 2 to 16) to Rough and finish the left pocket.

Center Pocket. Line 21 cancels the Rotate on line 18 and readies to repeat lines 19 to 20 to Rough and finish the center pocket. (In this case, we are repeating the MILL command as well as the set of geometry programmed before the first TOOL command).

Right Pocket. ROTATE on line 23 sets a new -20º rotated coordinates around the center of coordinates. Repeating lines 19 to 20 will repeat the MILL command and the pocket contour (lines 2 to 16) in order to Rough and finish the right pocket.

3 Cutting on a centerline (scroll)

```
25    TOOL  2    dia.25 flutes2 type1=CARBIDE MILL rad0 *** SCROLL
26  MILL         zrapid.05 zcut-.125 passes3.005 zret.1 zf37.7 xyf14.5
27  COMP         angle270 cl/con0 lookahead0
28  RADIUS       -.7 type1 x1 y.6
29  LINE         angle (
30  RADIUS       .8 type1 x3.5 y.85
31  UNCOMP       angle270
```

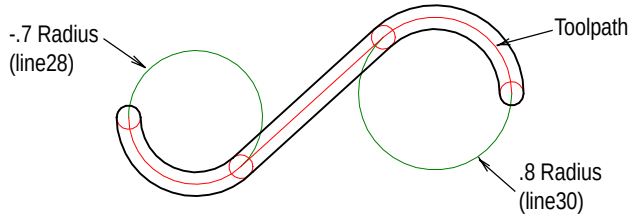


Note: A TOOL change always cancels any active ROTATE command automatically.

MILL passes. When G-ZERO prompts *How many passes to full depth?*, type *3.005*. This tells G-ZERO to make three passes leaving .005 zStock. G-ZERO automatically makes a fourth pass to remove the remaining .005 zStock.

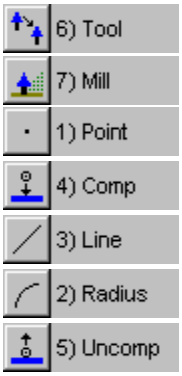
COMP cl/con. When you answer **0** (zero) for climb cut or conventional cut, G-ZERO keeps the tool center directly on the defined toolpath instead of compensating to the left or right. When cutting on the centerline (cl/con0), inside (-) and outside (+) Radii are judged as if the tool were climb-cutting.

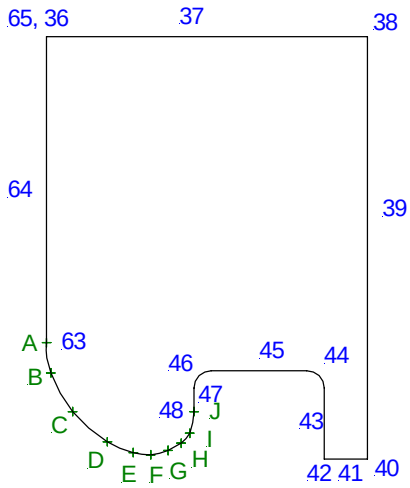
Since we don't know the angle of the line (between the two radius), just type (open parenthesis for angle and let G-ZERO calculate it for you.



4 Cut profile

```
32    TOOL  3    dia.625 flutes4 type0=HSS MILL rad0 *** PROFILE
33  MILL         zrapid.05 zcut-.51 passes1 zret.1 zf15.6 xyf21.4
34  POINT        x-2.9 y6.3
35  COMP         angle0 cl/co1 lookahead0
36  POINT        x-2.6 y6
37  LINE         angle0
38  POINT        x4.8 y6
39  LINE         angle270
40  POINT        x4.8 y-3.8
41  LINE         angle180
42  POINT        x3.8 y-3.8
43  LINE         angle90
44  RADIUS       -.4 type2 x3.8 y-1.75
45  LINE         angle180
46  RADIUS       -.4 type2 x.8 y-1.75
47  LINE         angle270
48  POINT        x.8 y-2.7
49  UNCOMP       angle270
```



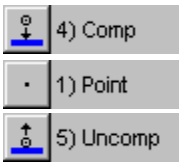


Coordinates:

	X	Y
A	-2.6	-1.1
B	-2.5	-1.8
C	-2.0	-2.7
D	-1.2	-3.4
E	-.6	-3.65
F	-.2	-3.7
G	.2	-3.6
H	.5	-3.43
I	.7	-3.2
J	.8	-2.7

Note: The numbers in illustration on the left correspond to the line numbers in the source code.

```
50  COMP     angle270 cl/con-1 lookahead0
51  POINT    x.8 y-2.7
52  POINT    x.7 y-3.2
53  POINT    x.5 y-3.43
54  POINT    x.2 y-3.6
55  POINT    x-.2 y-3.7
56  POINT    x-.6 y-3.65
57  POINT    x-1.2 y-3.4
58  POINT    x-2 y-2.7
59  POINT    x-2.5 y-1.8
60  POINT    x-2.6 y-1.1
61  UNCOMP   angle90
```



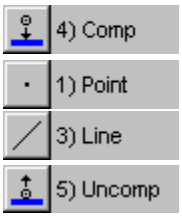
Smoothing. COMP on line 50 is for smoothing only. When smoothing, program a MINUS sign in front of climb-cut/conventional-cut.

Smooth can process POINTs only, so separate profiles that have Lines and /or Radii into separate COMP/UNCOMP groups.

Note: Point J (.8;-2.7) is the end of the normal contour before smoothing (line 48), AND it is also the first Point to smooth (line 51). We must program Point J again to include it in the smooth group of points.

After UNCOMP is programmed, G-ZERO redraws the Points as a smoothed contour.

```
62  COMP     angle90 cl/con1 lookahead0
63  POINT    x-2.6 y-1.1
64  LINE     angle90
65  POINT    x-2.6 y6
66  UNCOMP   angle90
67  POINT    x-2.9 y6.37 f0
```



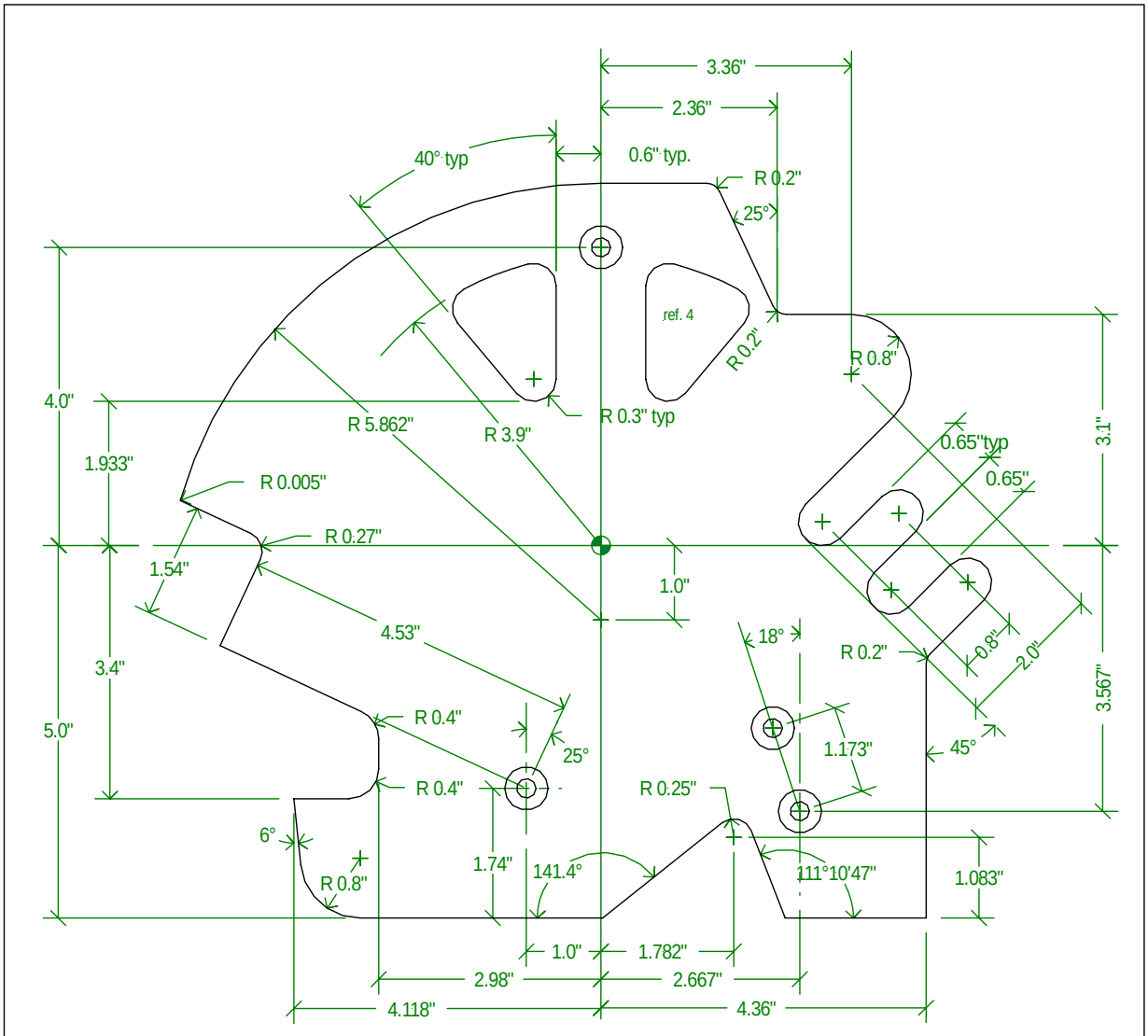
Note that the Point in line 63 is the same as the last point smoothed (line 60). Programming this Point again ensures that the tool cuts correctly between Point A and the top corner of the profile. ■

Project 12

Comprehensive

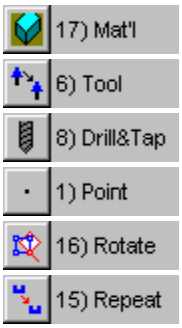
What you will learn:

- Mirror simple shapes
- Review important concepts learned in previous projects



GO G-ZERO CAM Software **Rapid Output Co.** **www.g-zero.com** **888-656-1945**	TITLE: Comprehensive	REFERENCES: 1. All measurements in inches (n.t.s.) 2. All holes 0.257" dia. (thru) with 82º x 0.58" dia. countersink 3. Break all corners (.001" fillet) 4. Pockets are 0.25" deep (typical) with 0.3" R (typical)	RECOMMENDED OP & TOOL LIST: 1. Drill (.257" dia.) 2. Countersink (.75" x 82º) 3. Rough OD (.5" dia. endmill) 4. Finish OD (.375" dia. endmill) 5. Pockets (.5" dia. endmill)
	MATERIAL: 0.5" Aluminum		
	DRAWING: M008 / INITIALS: MC		

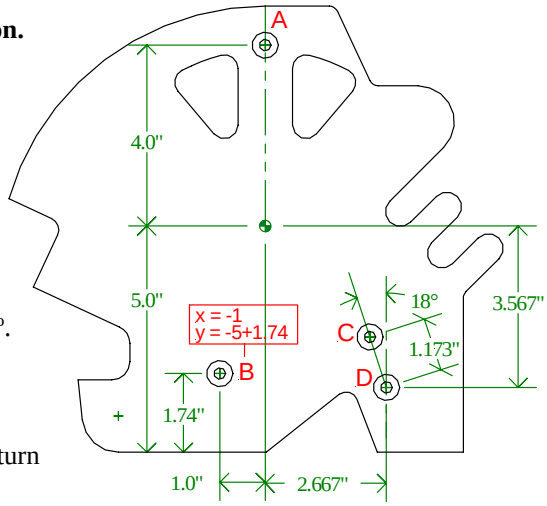
1 Stub Drill and Countersink




```
1   MAT'L      xmin-6 xmax5.5 ymin-5.5 ymax5.5 thk.5 type0=ALUMALOY
2              %MAXS5000
3    TOOL  1   dia.257 flutes2 type20=HSS DRILL rad0 *** STUB DRILL
4   DRILL      g81=C'DRILL zrap.1 zcut-.5 pecks1 tip1 zret.1 f27.1
5   POINT      x0 y4
6   POINT      x-1 y-3.26
7   ROTATE     angle18 xpiv2.667 ypiv-3.567
8   POINT      x0 y1.173
9   POINT      x0 y0
10  ROTATE     angle0 xpiv0 ypiv0
11   TOOL  2   dia.75 flutes1 type30=C'SINK rad0 *** COUNTERSINK
12  DRILL      g82=C'SINK zrap.05 zcut0 pecks82 tip.58 zret.05 f9.2
13  REPEAT     from5 thru10
```

%MAXS. This % Command sets the maximum allowable rpm (spindle speed) for all the tools in this program. Use %MAXS before the first Tool.

Input Math Calculation. Y center coordinates of hole B is not directly given in the blueprint. Enter the math operation *-5+1.74* and let G-ZERO calculate it for you.



Rotate. Centers of holes C and D are at 18º. Hole D is the anchor of the angle and the pivot point (x2.667 y-3.567). Program a ROTATE to turn the dimensions by +18º (counterclockwise is always positive) and translate dimensions by 2.667 in X and -3.567 in Y.

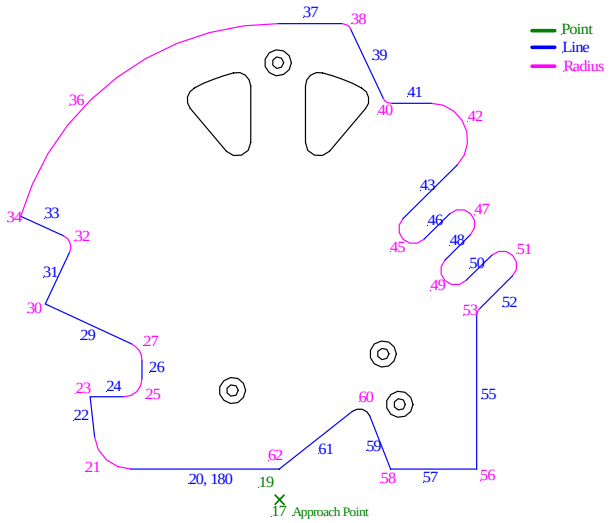
Countersink. Use a DRILL command (line12) to countersink the holes. Note that *zcut0* is the surface of part, *pecks82* is the angle of countersink (in degrees), and *tip.58* is the diameter of the finished countersink.

Repeat. Countersink the holes by reusing source lines 5 through 10 using command REPEAT.

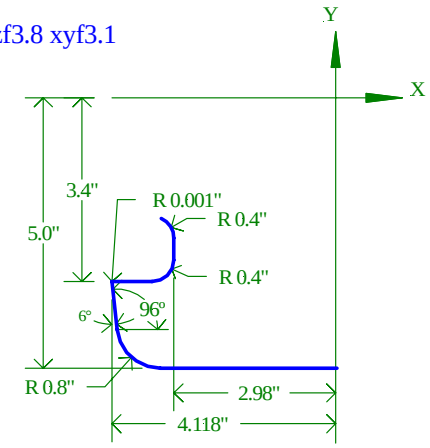
2 Rough and Finish Profile

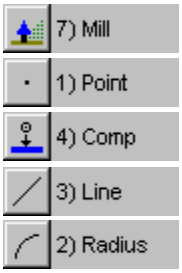

```
14   TOOL  3   dia.5 flutes4 type0=HSS MILL rad0 *** ROUGH PROFILE
15  STOCK      xystk.03 zstk0
```

Stock. Stock leaves extra material on the cutting surfaces by setting the distance the tool should stay away from the finished dimension of part walls and/or floor for later cleanup. STOCK must be programmed before describing the contour's cutting path.



16	MILL	zrapid.05 zcut-.52 passes1 zret.1 zf3.8 xyf3.1
17	POINT	x0 y-5.3
18	COMP	angle180 cl/con1 lookahead0
19	POINT	x0 y-5
20	LINE	angle180
21	RADIUS	.8 type0
22	LINE	angle96
23	RADIUS	.001 type2 x-4.118 y-3.4
24	LINE	angle0
25	RADIUS	-.4 type2 x-2.98 y-3.4
26	LINE	angle90
27	RADIUS	-.4 type0

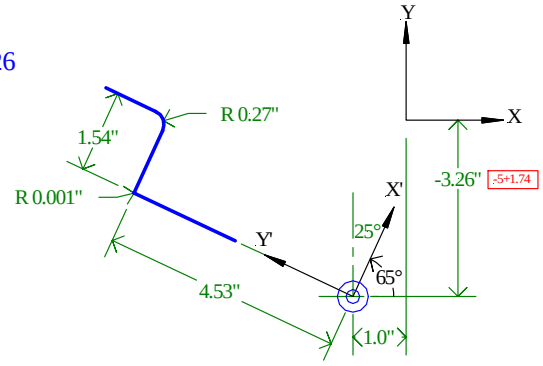


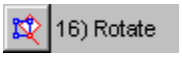


Program an “approach” Point just off the part (e.g.: x=0, y=-5.3) to drop the cutter in a safe location.

Program COMP to offset for cutter Radius. The COMP angle (180º) is determined by the direction between the Point (line 19) and Radius (line 21).

28	ROTATE	angle65 xpiv-1 ypiv-3.26
29	LINE	angle90
30	RADIUS	.001 type2 x0 y4.53
31	LINE	angle0
32	RADIUS	-.27 type2 x1.54 y4.53
33	LINE	angle90
34	RADIUS	.005 type0
35	ROTATE	angle0 xpiv0 ypiv0

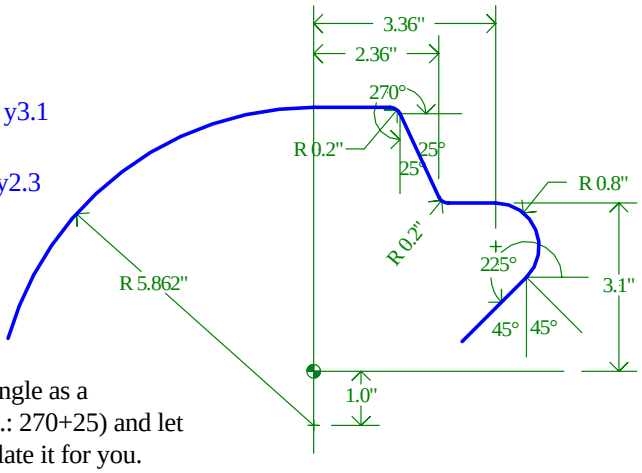




This section of the contour is dimensioned at a +65º (you can key in calculation 90-25) angle to the normal XY plane. The pivot point of the angled dimensions is x=-1 and y=-3.26 (can be entered as a calculation: -5+1.74). The pivot point becomes a temporary origin.

Remember to turn Rotate off by answering all the questions with 0 (zero).

36	RADIUS	5.862 type1 x0 y-1
37	LINE	angle0
38	RADIUS	.2 type0
39	LINE	angle295
40	RADIUS	-.2 type2 x2.36 y3.1
41	LINE	angle0
42	RADIUS	.8 type1 x3.36 y2.3
43	LINE	angle225



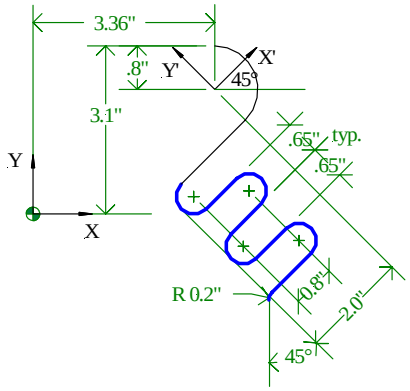
Calculations

Line 39.
You can
enter this line angle as a
calculation (e.g.: 270+25) and let
G-ZERO calculate it for you.

Line 42. Y value can be entered as 3.1-.8

Line 43. A calculation for this line angle can be: 270-45)

44	ROTATE	angle45 xpiv3.36 ypiv2.3
45	RADIUS	-.325 type1 x-1.675 y-1.125
46	LINE	angle0
47	RADIUS	.325 type1 x-.875 y-1.775
48	LINE	angle180
49	RADIUS	-.325 type1 x-1.675 y-2.425
50	LINE	angle0
51	RADIUS	.325 type1 x-.875 y-3.075
52	LINE	angle180
53	RADIUS	-.2 type0
54	ROTATE	angle0 xpiv0 ypiv0



Rotate. This section of the contour is dimensioned at a +45º angle to the normal XY plane. The pivot Point of the angled dimension is x3.36 y -2.3 (y value can be entered as 3.1-.8). Program a ROTATE command to rotate dimensions +45º and translate by 3.36 in x and 2.3 in y. The pivot point becomes a temporary origin x0 y0.

Calculations. Let G-ZERO make the calculations for you:

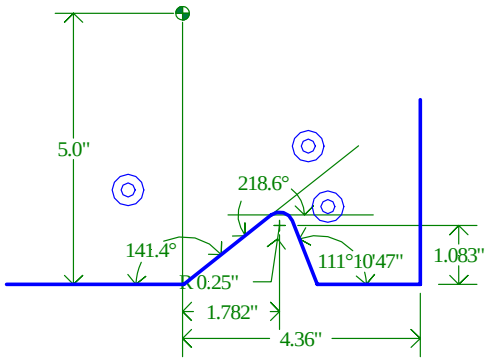
Line 45. Radius = -.65/2 x = -2+(.65/2) y = -.8-(.65/2)

Line 47. Radius = .65/2 x = -2+(.65/2)+.8 y = -.8-.65-(.65/2)

Line 49. Radius = -.65/2 x = -2+(.65/2) y = -.8-.65-.65-(.65/2)

Line 51. Radius = .65/2 x = -2+(.62/2)+.8 y = -.8-.65-.65-.65-(.65/2)

55	LINE	angle270
56	RADIUS	.001 type2 x4.36 y-5
57	LINE	angle180
58	RADIUS	.001 type0
59	LINE	angle111.1797
60	RADIUS	-.25 type1 x1.782 y-3.917
61	LINE	angle218.6
62	RADIUS	.001 type0
63	LINE	angle180
64	POINT	x0 y-5
65	UNCOMP	angle180
66	POINT	x0 y-5.3 f20



Angles in degrees/minutes/seconds. Line 59. To convert an angle defined in degrees/minutes/seconds, use the format dd.mmss and press the " quote key. In this case, to enter 111º10'47", type 111.1047". G-ZERO will convert this number to the decimal format: 111.1797.

Calculations. Line 60. Y value can be entered as calculation -5+1.083
Line 61. Line angle can be entered as calculation 180+(180-141.4)

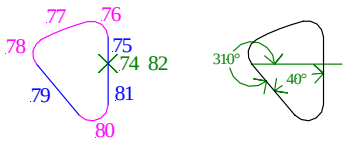
6) Tool
7) Mill
15) Repeat

```
67   TOOL  4    dia.375 flutes4 type1=CARBIDE MILL rad0 *** FINISH PROFILE
68   MILL       zrapid.02 zcut-.52 passes1 zret.1 zf15.3 xy9.2
69   REPEAT     from18 thru66
```

REPEAT. Finish the profile using the cutter path previously defined by reusing the source lines from COMP (line 18) through retract point after UNCOMP (line 66).

3 Rough and Finish Pokets

```
70   TOOL  5    dia.5 flutes2 type0=HSS MILL rad0 *** RUF & FINISH POCKETS
71   MILL       zrapid.1 zcut-.25 passes2 zret.1 zf26.7 xyf10.7
72   ROUGH      stk.005 stp.2 angle0 cleanup1
73   COMP       angle90 cl/con41 lookahead0
74   POINT      x-.6 y3
75   LINE       angle90
76   RADIUS     -.3 type0
77   RADIUS     -3.9 type1 x0 y0
78   RADIUS     -.3 type0
79   LINE       angle310
80   RADIUS     -.3 type1 x-.9 y2.233
81   LINE       angle90
82   POINT      x-.6 y3
83   UNCOMP     angle90
84   ROUGH      stk.005 stp.2 angle180 cleanup1
```



29) Rough
4) Comp
1) Point
3) Line
2) Radius
5) Uncomp

Rough. Program a ROUGH command with cleanup=1 using dimensions provided on first page of this project.

Define left pocket. POINT commands on lines 74 and 82 correspond to the start and stop points. X and Y values of RADIUS in line 80 can be entered as calculations: X = .6+.3 and Y = 1.933+.3

```
85                MIRROR RIGHT POCKET
86   MILL         zrapid.1 zcut-.25 passes2 zret.1 zf26.7 xyf10.7
87   ROUGH        stk.005 stp.2 angle0 cleanup1
88   COMP         angle270 cl/con41 lookahead0
89   REPEAT       from-82 thru74
90   UNCOMP       angle270
91   ROUGH        stk.005 stp.2 angle0 cleanup1
```

7) Mill
29) Rough
4) Comp
15) Repeat
5) Uncomp
O Override

Comment. Line 85 is a comment line. Comments describe the part or operation, or give instructions to the operator. (Comments that begin with % can be sent to the CNC if the control can read them.)

Mirror. To mirror a cutter path in the X axis, use a REPEAT command with a - *minus* sign in front of the first value (from). (Since it is not a normal procedure to use a minus sign in REPEAT, you need to type O before Enter to override the system's limitations. *(To mirror in the Y axis, use a minus sign in front of the "thru" value).* Note that you need to reverse the cut (from line 82 through 74) to reverse the direction your cutter takes along a surface. ■