GCG Manual

G-Code made Simple

by

Matthew Evans

kaptnknemo@yahoo.com

Manual Version 2.3
Software Version 2.4

Table of Contents

D^3 Says

After playing with your software, I am now thinking I may just do most of my programming with it instead of trying to use CAD/CAM software (at least for the simpler projects). Cause' its so easy! I downloaded some of the resulting G-code to my Mach1 software (www.artofcnc.ca) and it all read in just fine. It was kinda cool to see my simple inputs turn into a graphical tool path on the computer screen!

GCG Manual Version 2

March 2003

Introduction

GCG stands for **G**-**C**ode **G**enerator. It is a small program written in Qbasic that creates properly formatted lines of G-Code. G-Code is an industry standard method of controlling CNC machining equipment such as mills and lathes. GCG was created by Matthew Evans to eliminate the drudgery of tyfing code for low mousepower machines such as the MAXNC-10 becaus he can't typ worthe a darn. Small machines make small cuts and GCG makes creating the multiple megatudes of G-Code lines required for small cuts bearable.

GCG is feature-based program. A 'feature' in this case is a circle, line, arc or a combination of lines and arcs that form the toolpath to be machined. Some other features are circular and rectangular pockets. What GCG does is ask you a few questions about the feature, formats the G-Code lines for that feature and writes the lines to a text file that can be read by the CNC machine's code interpreter. Because GCG formats the file with the feature information before the code, you can easily edit the output file to remove features or add features from other files by cutting and pasting. See Appendix A for a sample GCG output text file. The file is then read by the CNC control software, which controls the steppers or servos on the machine itself.

GCG was written for the MAX10 with the additional driver unit for the fourth axis and spindle control. It will most likely work for any of the MAX series and most other small CNC conversions. Try it and see! If you don't have spindle control, then make sure your spindle is on before running a GCG generated program.

It has been verified to work with EMC by D^3, who lives in one of those states up near Canada, the home of Bob and Doug Mackenzie, as well as Rush.

Gimmeware

Gimmeware is the newest concept in software distribution. Someone, like me, writes some code (the *ware* part) that someone else, like you, finds useful. That someone else, like you, gives someone, like me, something, like money, beer or chocolate (the *gimme* part). How much money, beer or chocolate is up to that someone else, i.e. you. However, please note that the beer or chocolate must be of high quality, for beer that means Sam Adams or better, for chocolate, Godiva is a good start. Authors of Gimmeware will generally accept any form of money without any quality stipulations. Note that you can give me money *for* beer or chocolate if you live in Outer Mongolia or anywhere in Texas. You can give me money via Paypal, my account is the same as my e-mail kaptnknemo@yahoo.com. If you choose to give me beer or chocolate instead of money, and you do not live in Outer Mongolia or anywhere in Texas, please contact me at the above e-mail address for further instructions. If you live in Germany, the only acceptable form of payment for Gimmeware is appfleschnapps bonbons, those little booze filled candies that I remember from my youth. Any further information about my youth and booze filled candies will not be made available. The author of this manual is a chip making, tin knocking, fuse popping garage mechanic who would like to make some money to support his hobbies, but is generally to lazy to do any real work.

Disclaimer

Hey, *I* wrote this code to help *me* machine stuff. I am sharing it with you because I am a damn nice guy. It has no guaranties, warranties, or promises, expressed or implied. I also will not accept any responsibility if you cut your silly fingers off, after all you are the one using, misusing and abusing the machine anyway. No whining, moaning or other complaining will be tolerated. However, if you have complements or better yet, any of those funny flat green pieces of paper with pictures of dead presidents on them, feel free to contact me.

Installation

Well, you really don't have to install it, just unzip the file you downloaded. If you are reading this, you have allready done that. There were three files in that zip; GCG.exe which is the main program, manual.pdf, which is this manual and gcgparam.par which contains the user settable machining parameters. Put gcg.exe and gcgparam.par in the same directory. GCG will put its output file in this directory too.

Section 0 The Main Menu

Start GCG by typing GCG in a DOS window in the directory where you have placed GCG. It will ask you for the name of the G-Code file you are about to create. . Keep the filename to eight characters or less. GCG will add the .txt suffix for you, so remember to type the complete filename.txt when using your screen plotter or G-code interpreter. After you enter the file name, GCG looks for the parameter file and loads the parameters that you can set with option 9. Then the main menu pops up.

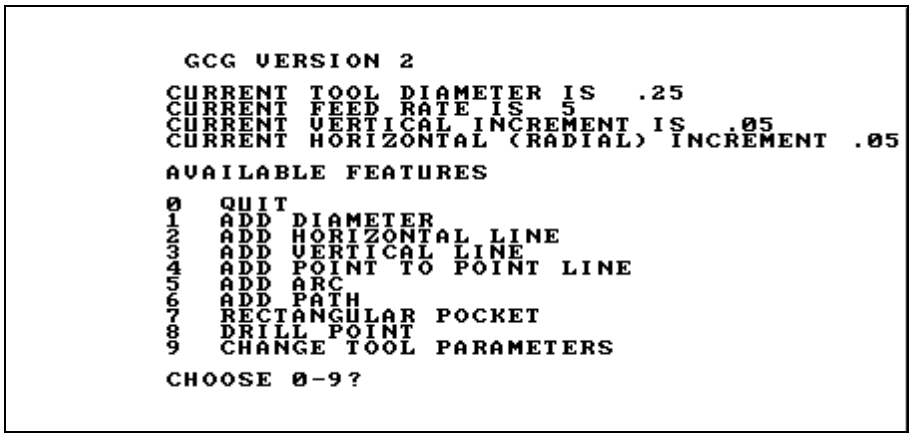


Figure 1

The main menu shows the current tool parameters and the features available to add to the open file. Choose 0 to close the file and exit GCG.

Section 1 Add Diameter

You can machine several types of features with the "Add Diameter" function. You can cut a through hole, a blind hole (otherwise known as a pocket) or cut out a plug. Carefully examine the following two drawings of a bearing block. Most of the machining on this block is accomplished by combining several GCG features. The first feature we are going to do is cut the large bearing pocket. The 1.657 pocket is .400 deep. A 1.250 hole goes through the remainder of the material. In this case, the center portion can be removed as a plug for the first cut. Using the lower left hand corner of the block (where the .000 ,.000 is, regardless of how you hold the page). The center of the pocket is at X 1.580 Y .992. Photo #1 shows the plate mounted on the MAXNC10 before the first cut. Note that GCG starts the cut at the largest Y value and cuts clockwise.

The default mill diameter is .25 and the default horizontal and depth increments are .050. We are going to use a .125 end mill for this cut and the default horizontal and depth increments will change to .020. Use menu pick 9 " change tool parameters to change these values. The new "default" values are stored in the .par file that comes with GCG. Make sure that both GCG and the .par file are in the same directory.

There are several advantages to cutting a smaller diameter first, right inside the finished one. With the depth and horizontal increment set to .020, most of the material is removed on the first cut, making a .125 x .020 groove; this leaves only a .020 x .020 increase on one side for the second cut. This small second cut will leave a better finish and a more accurate cut on the final diameter. Second is that the cut is wider than the end mill, giving some of the chips an easier way out and the end mill runs cooler because it is not rubbing against two sides at once. I use a can of compressed air with a tube to periodically blow the chips out. If the material was steel instead of aluminum, I'd use a cutting fluid to lubricate and cool the cutter as it works.

To make this cut as described above, enter 1 from the main menu. GCG will ask you for the center of the hole then the start and finish diameter. For this cut we will use .960 and 1.250 respectively. The inner diameter equals the outer diameter minus 2 times the mill diameter minus 2 times the extra space we want for

the chips or 1.250-(2*.125)-(2*.020)= .960. GCG will then ask you for the start and finish depth. For the first cut the end mill is zeroed on top of the material so the start depth is 0 and the end depth is .495 or almost all the way through the material. Note that the material is held above the milling machines table. You can always zero the end mill on top of the material with a shim if you add that amount to the depth. You shouldn't cut all the way through, because the plug might fly across the room. If you leave .005-.010 in aluminum, you can push the plug out and then remove the burr with a deburring tool. Remember that I will not be held responsible if you fling a plug across the garage and break the Porsche's windshield.

With that info, GCG will create a whole mess of G-code and write it to the file you opened when starting the program. You will then be returned to the main menu for the second cut. This cut has a start diameter of 1.240, just to make sure it doesn't leave a thin web and an end diameter of 1.656. The start depth is 0 and the end depth is .400.

The Add diameter feature is a *finished size* feature, i.e. you enter the finished size and GCG does the math for the toolpath. Some other GCG features are toolpath features where you enter tool path info directly.

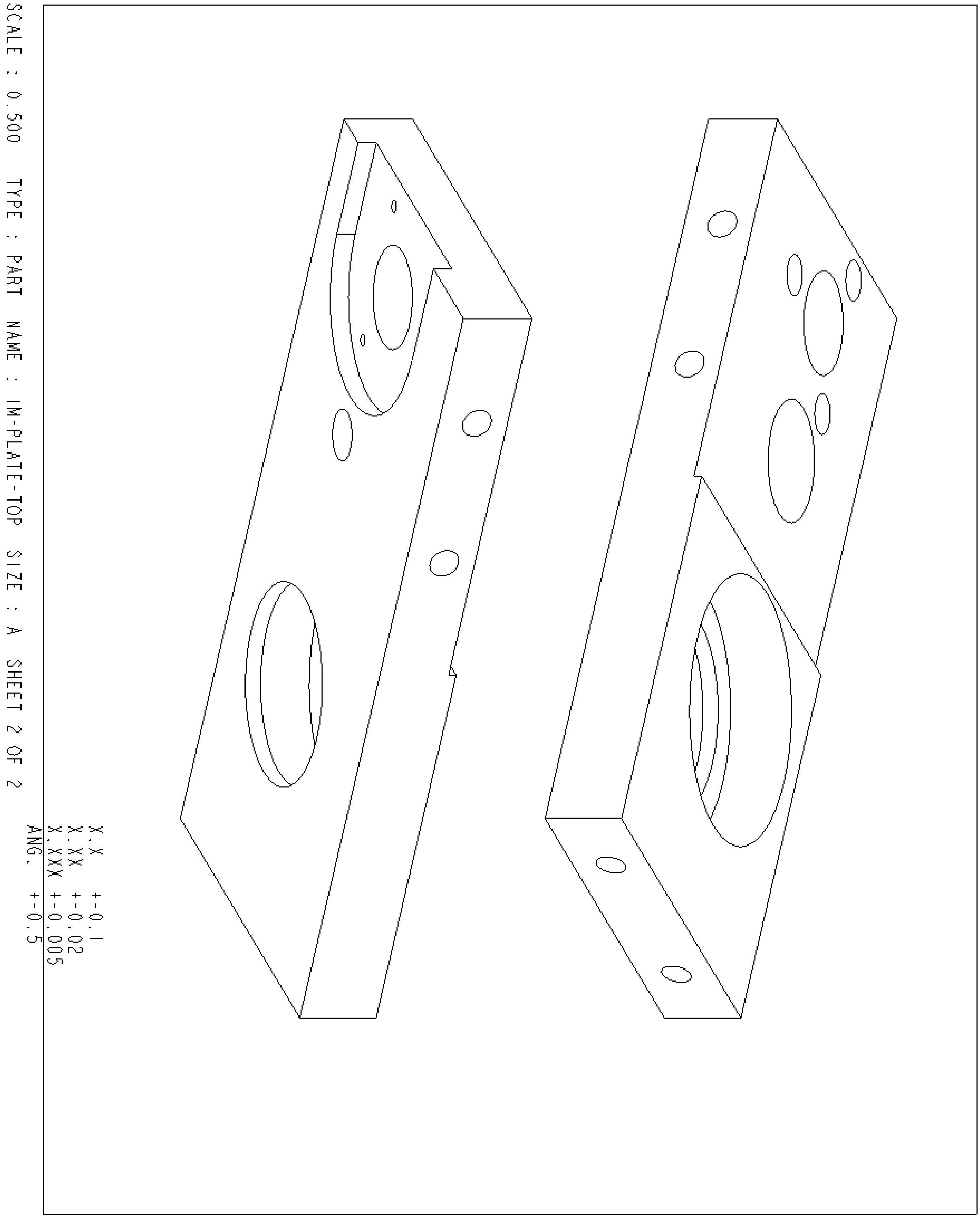

SCALE : 0.500 TYPE : PART NAME : IM-PLATE-TOP SIZE : A SHEET 2 OF 2
X.X +-0.1
X.XX +-0.02
X.XXX +-0.005
ANG. +-0.5

Figure 2

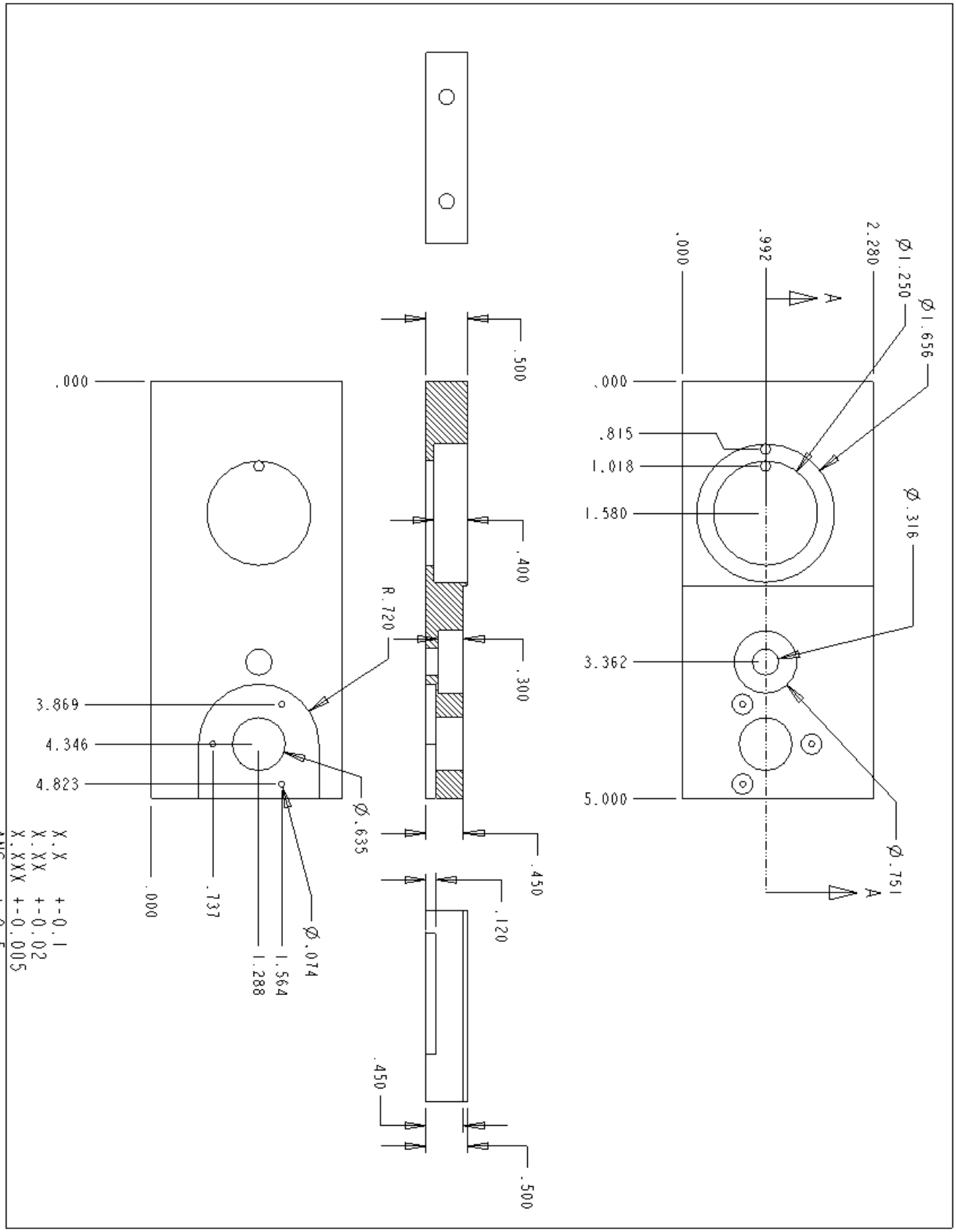

SCALE : 0.500 TYPE : PART NAME : IM-PLATE-TOP SIZE : A SHEET 1 OF 2
X.X +-0.1
X.XX +-0.02
X.XXX +-0.005
ANG. +-0.5
.000
2.280
Ø1.250
Ø1.656
.992
A
.000
.815
1.018
1.580
Ø.316
3.362
5.000
Ø.751
A
.500
.400
R.720
.300
.450
.120
.450
.500
.000
3.869
4.346
4.823
Ø.635
.737
.000
Ø.074
1.288
1.564

Figure 3

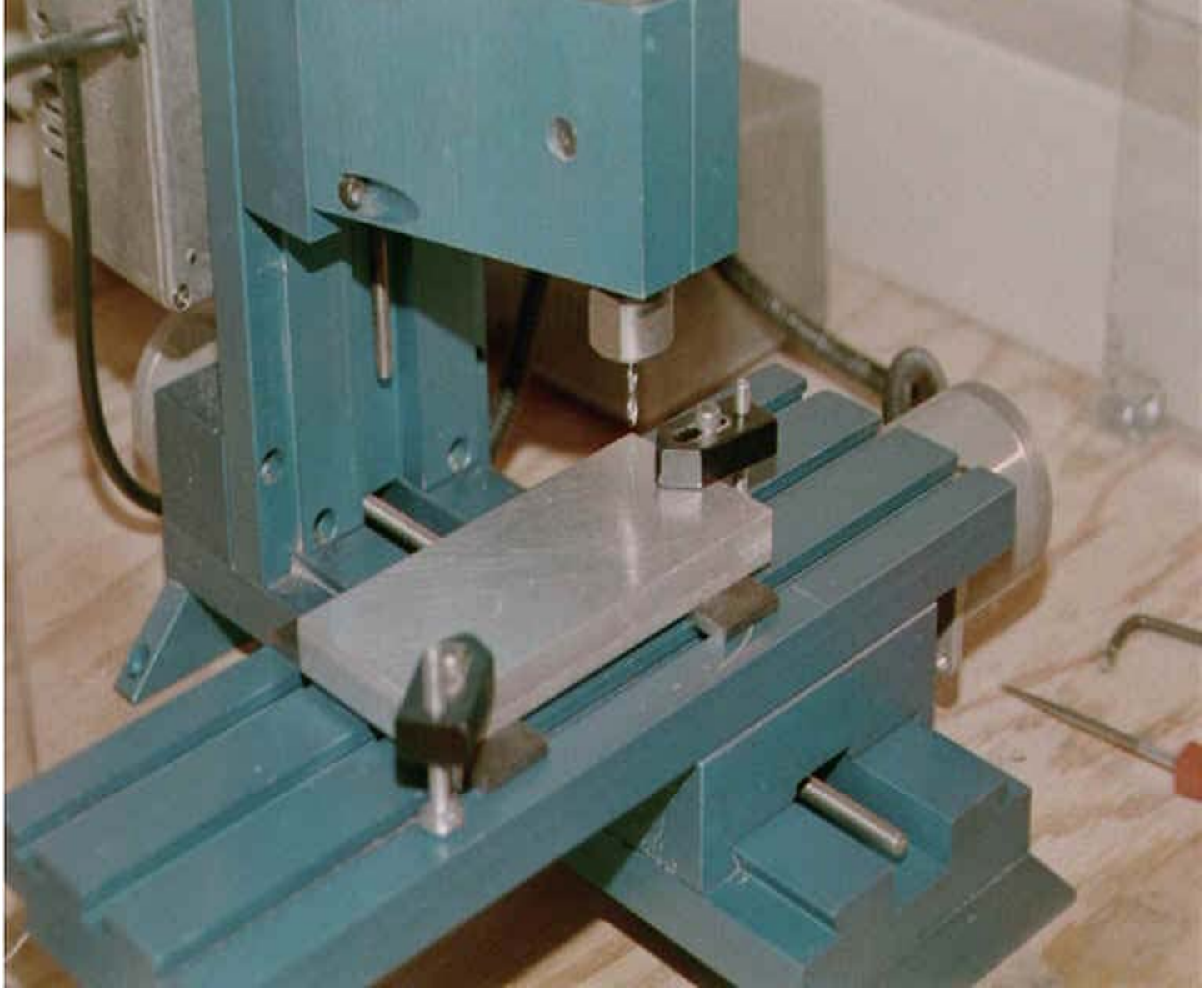


Photo 1

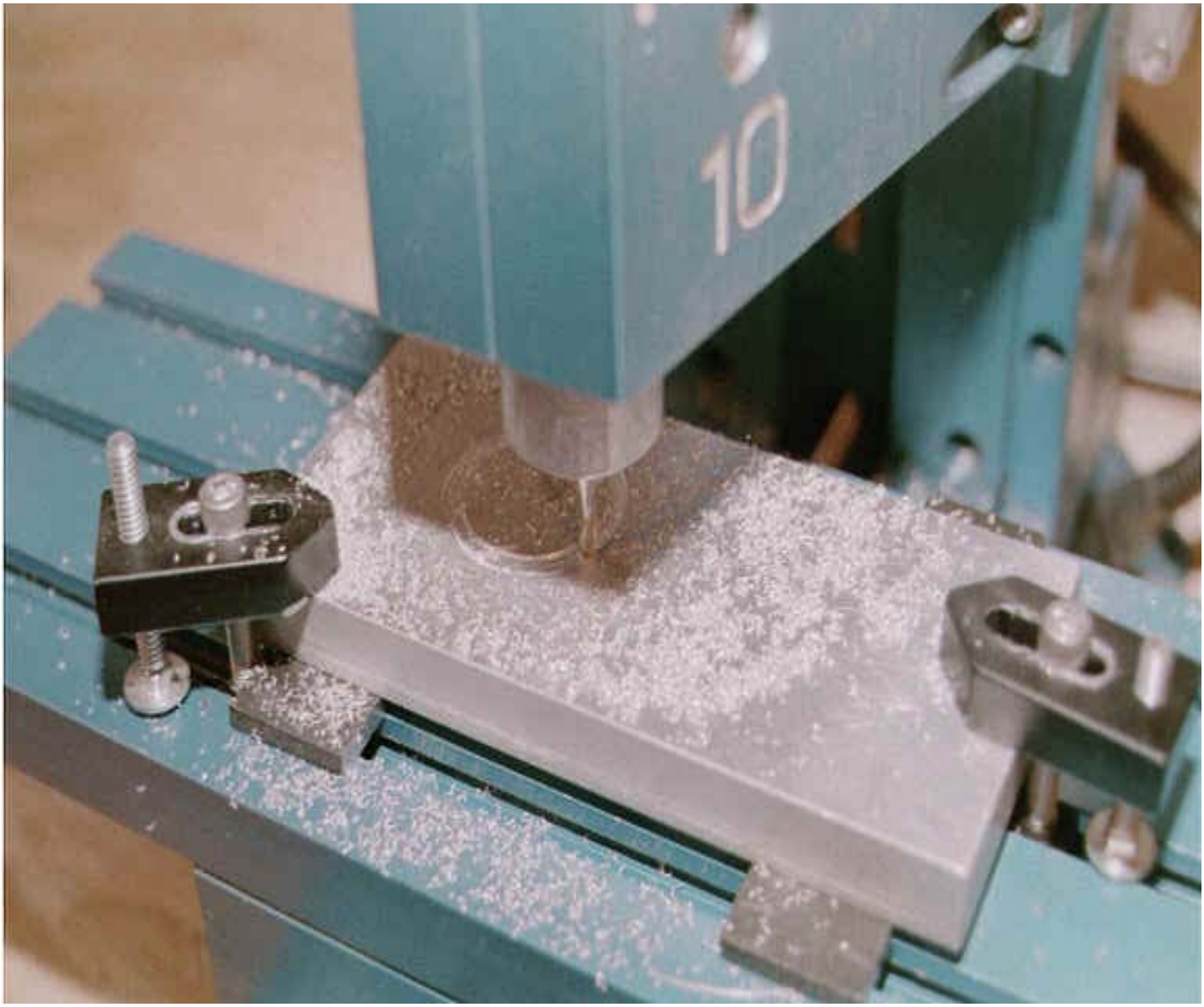


Photo 2

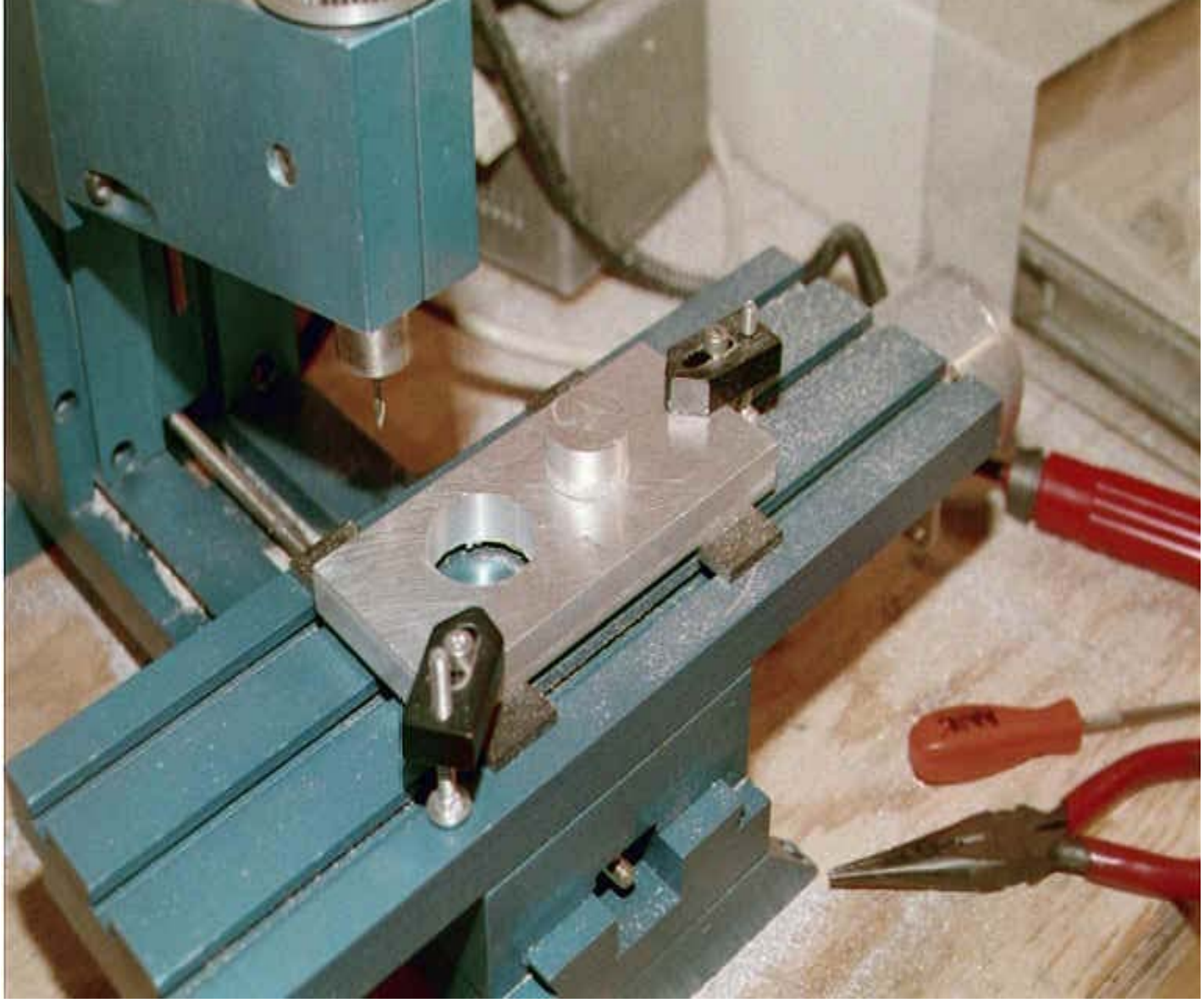


Photo 3

Section 2 Add Horizontal Line

This one is really easy. GCG asks you for the start position's X and Y and the end positions X. Y stays the same because the line is horizontal. You'll then enter the start and end depth. Piece of Cake. Use this to add a slot or trim off the end of some stock. Look at the following drawing. There is a horizontal line that begins at X=2.625 Y=2.250 and ends at X= 4.250 Y=2.250. Be very careful when you set the incremental depth because GCG generates a command to plunge the cutter in at speed F1. If you're incrementing the cut outside the material this isn't a problem. If you are inside the material, like all of the cuts in the following drawing, limit the vertical increment to about .020" for aluminum and .050" for plastics. Also make sure you have a center cutting cutter. Remember you have plenty of time and not enough mousepower or money to take the big cuts. If you have tons of money, go buy a Bridgeport. See Appendix D for step-by-step instructions on how to make the plate in figure 4.

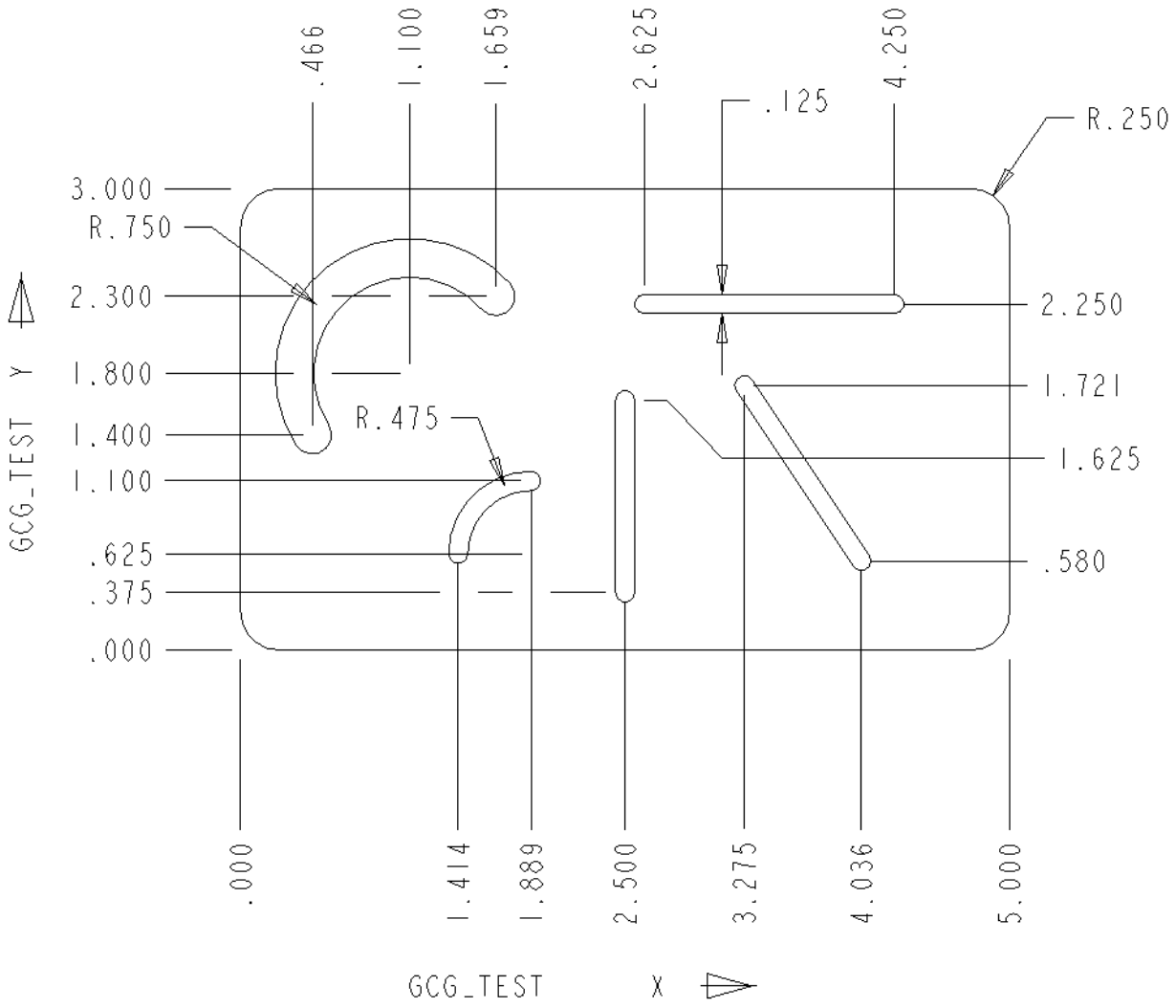


Figure 4

Section 3 Add Vertical Line

Just as easy as a horizontal line. GCG asks you for the start position's X and Y and the end positions Y. X stays the same because the line is vertical. You'll then enter the start and end depth. Another piece of cake. Use it the same way as a horizontal line. On the above drawing the vertical line begins at X=2.500 Y=. 375 and ends at X=2.500 Y= 1.625

Section 4 Add Point to Point Line

Almost as easy as the last two! GCG asks for the start and end position's X and Y and the depth parameters. Plug'em in and away you go. Use it just like the previous two and it will scramble your eggs for you! (Just checking). In the above drawing the line starts at X=3.275 Y=1.721 and ends at X=4.036 Y=.580. Note that you can start at the end and end at the start on any of these. You should also notice that the ends of the cuts are round, just like the endmill. The cuts are as wide as the endmill, of course, but notice that features 1-5 are toolpath features so if you are trying to trim an edge, remember to add or subtract ½ the width of the endmill.

Section 5 Add Arc

Ok you guessed it, just plug in the numbers and away you go. Look again at the drawing, we'll tackle the easy one first. It is the 90 degree arc that starts at X=1.414 Y=. 625 and ends at X=1.889 Y=1.100 with a .475 radius. GCG needs X, Y of the center first, then X, Y of the start and X, Y of the end. The center of the arc is at X=1.889, Y=. 625. The start point is 1.414, .625. For arcs, GCG always cuts Clockwise first and then Counterclockwise and repeats until the depth is reached. The end point is 1.889, 1.100. Next you'll be asked for the start and finish depth. When calculating the points for an arc, make sure they make geometric sense as GCG will put out code for whatever garbage you put in. It is up to you to test the code on a screen plotter or a scrap piece of plastic before you cut the unobtanium slab. The second arc's center is at 1.1, 1.8, starts at 0.466,1.400 and finishes at 1.659, 2.300.

Section 6 Add Path

The other features above are just window dressing, except for diameters, they're pretty neat. Where GCG really shines is with the path feature. You can add a complex tool path consisting of lines and arcs and cut all sorts of stuff. For an example of a *path* feature, look at the drawings of the of the test block. There are 9 points that the tool must go through, 1 through 8 as shown in the drawing and then from 8 to 1 to cut the last corner and close the path.

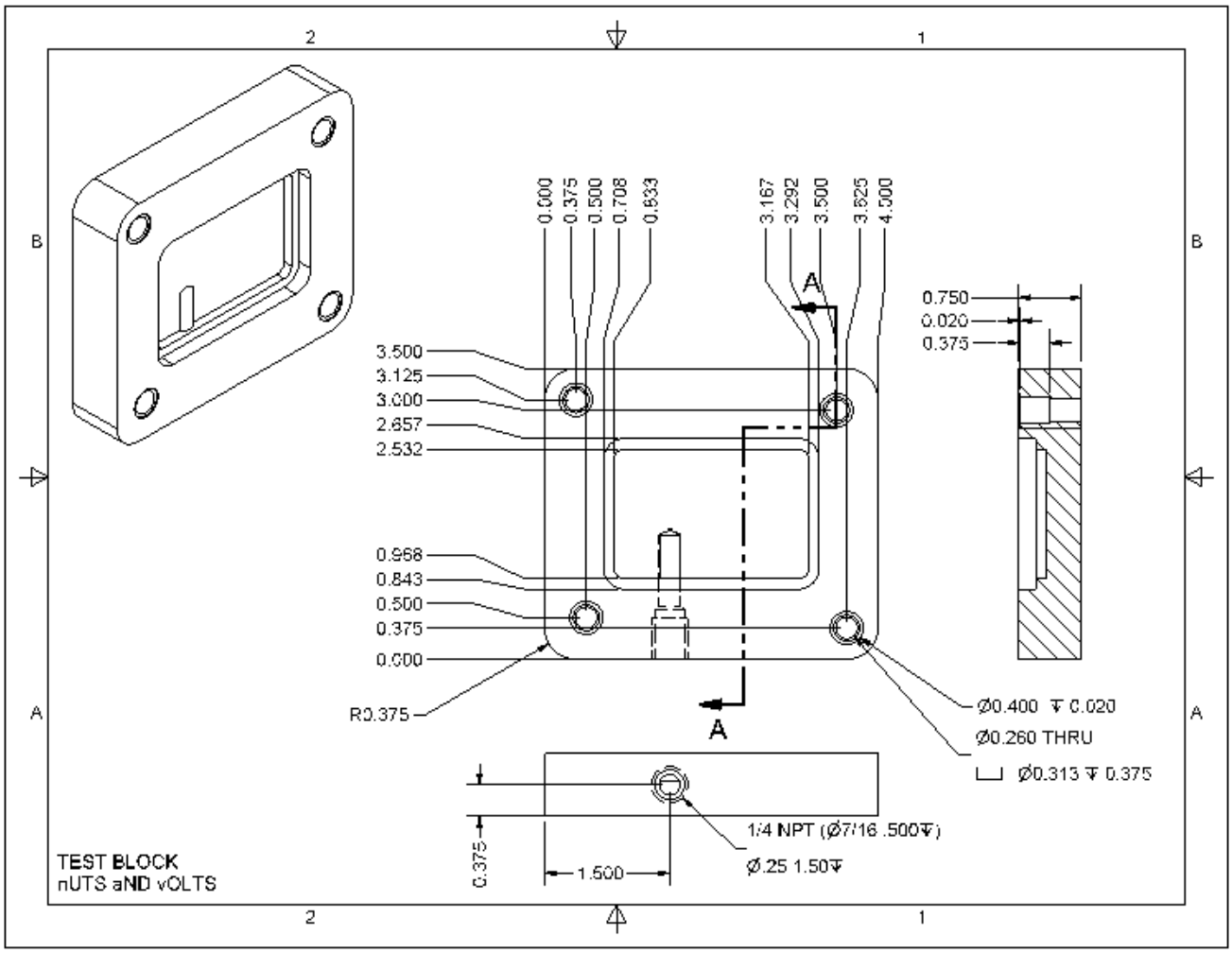

2
1
B
A
0.000
0.375
0.500
0.708
0.833
3.167
3.292
3.500
3.625
4.500
3.500
3.125
3.000
2.657
2.532
0.968
0.843
0.500
0.375
0.000
R0.375
A
A
0.750
0.020
0.375
Ø0.400 ↧ 0.020
Ø0.260 THRU
⌴ Ø0.313 ↧ 0.375
1/4 NPT (Ø7/16 .500↧)
Ø.25 1.50↧
0.375
1.500
TEST BLOCK
nUTS aND vOLTS

Figure 5

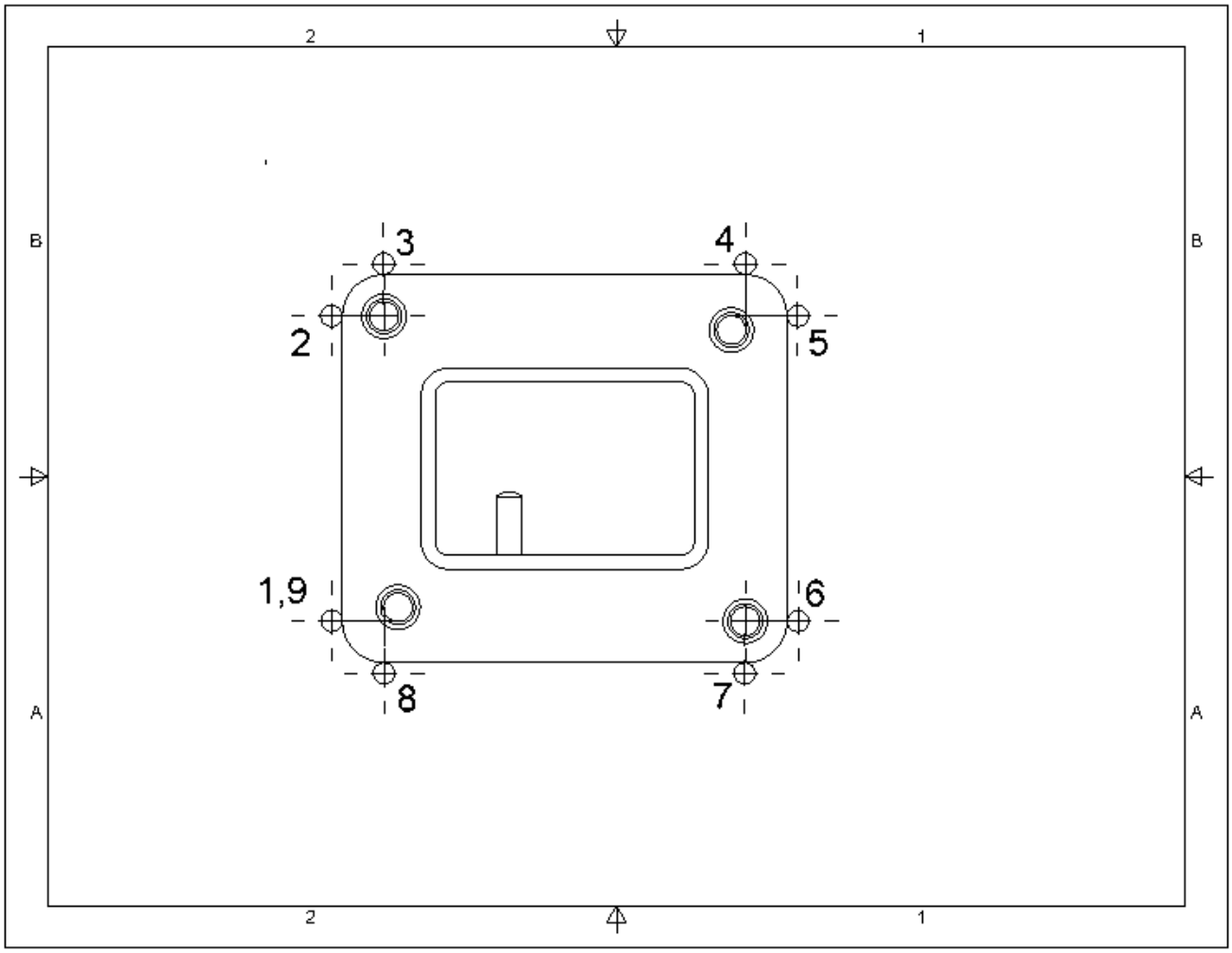


Figure 6

This will be done with the 'read points' option of the path feature, where the points are read in from a text file. The properly formatted text file appears below. This file assumes a .25 diameter cutter. Remember that this is a toolpath feature; in this case the toolpath or centerline of the tool is always ½ the tool diameter or .125 outside of the finished edge.

9,0, -0.73,0.05
1, -0.125,0.25,0,0
1, -0.125,3.25,0,0
2,0.25,3.625,500,0
1,3.75,3.625,0,0
2,4.125,3.25,0, -.500
1,4.125,0.25,0,0
2,3.75, -0.125, -.500,0
1,0.25, -0.125,0,0
2, -0.125,0.25,0,500

The first line contains the number of points, the start depth, end depth and the depth increment. Remember to keep the depth increment small because we have plenty of time and not enough mousepower! This is the only feature that does not use the default value for vertical increments. The second line moves the tool to the starting point. The third through N+1 lines, where N is the number of points, starts with the segment type (i.e. G-Code)

X location, Y location, I value and J. value. For segment type 1, a line, the values for I and J are 0 and are not used to create the G-Code line describing a line. However they are still needed by the 'read points' option so that GCG stays in the right place while reading the text file. GCG requires that the tool be at the starting depth when you start the path feature. There are probably neater, more programmerish type ways of doing this, but it works. Notice that the last point is directly on top of the first point, but that the segment information is different. The following is a portion of the G-code text file generated by this input file.

```
G01 Z-.050 F3
G01 X -.125 Y 3.250
G02 X 0.250 Y 3.625 I  0.375 J  0.000
G01 X 3.750 Y 3.625
G02 X 4.125 Y 3.250 I  0.000 J -0.375
G01 X 4.125 Y 0.250
G02 X 3.750 Y -.125 I -0.375 J  0.000
G01 X 0.250 Y -.125
G02 X -.125 Y 0.250 I  0.000 J  0.375
```

Although this is a very simple example of the path feature, you'd have to type in the above code 14 more times to cut the .750 depth of the outside of the block. See the screen capture and photos below for the before and after shots.

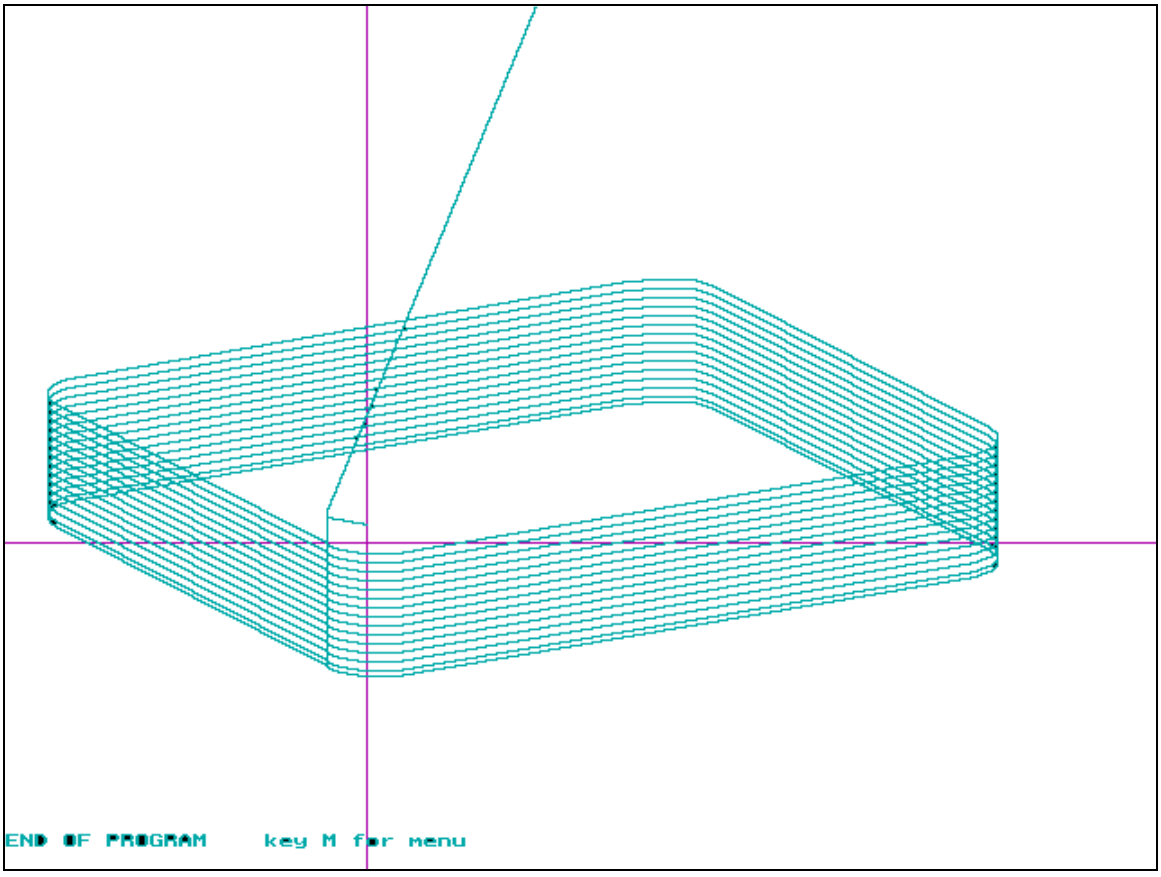


Figure 7

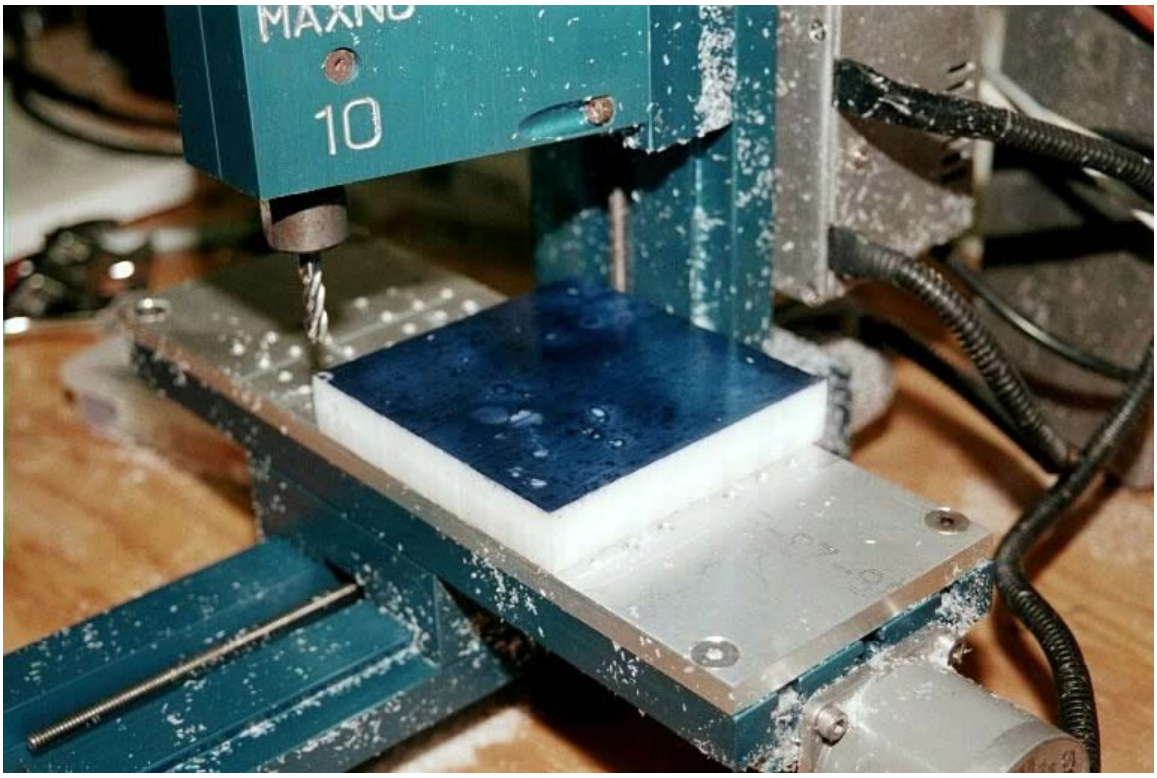


Photo 4

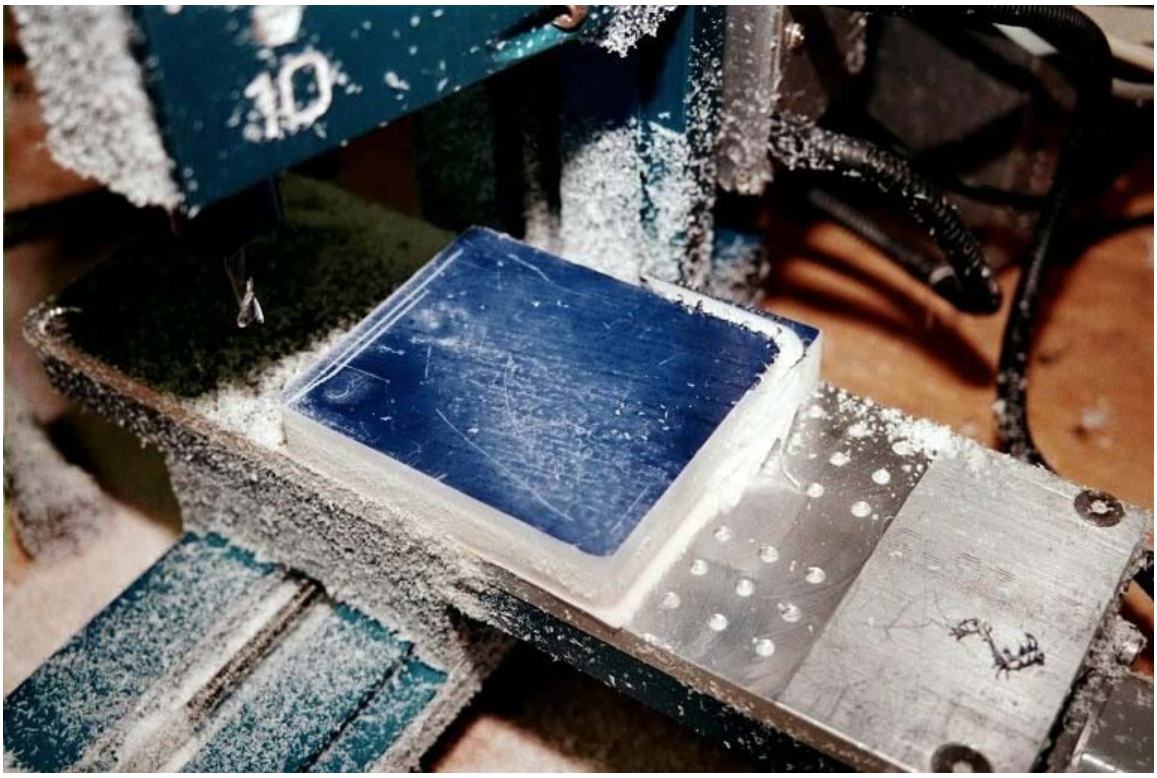


Photo 5

The casually observant reader will catch that the above two photo's are not of the same block. I had to cut several of them and some of the pictures didn't come out.

Any number of points can be in the text file for any combination of lines and arcs. Remember that we are using GCG to generate the large amount of data needed to cut on a small machine with limited mousepower. As always, check the code with a screen plotter before cutting on a machine.

The second path option is for the user to enter points manually. The number of points is limited to five. If you path has more than five points, you'll just have to write the small text file required by option one. This feature is mainly to cover any small features you forgot to enter with the first path feature. Note that this feature uses the default values for vertical increment, unlike the first feature

Section 7 Rectangular Pocket

For a rectangular pocket look again at the drawing of the rectangular block. There are two rectangular pockets cut out of the middle, one .200 deep and one .325 deep. All you do with GCG is input the lower left corner, the upper right corner and the depth. The program does the rest. You can use the default values for the vertical and horizontal feed or change them from the main menu. The corners for the .200 deep rectangle are X_1=.708 Y_1=.843 and X_2=3.293 Y_2=2.658. The corners for the .325 deep rectangle are X_1= .833 Y_1= .968 and X_2=3.168 Y_2= 2.533. The result of these few entries is a 3 KB file. Since the file for these rectangles is so big, I'll change the values of the horizontal and vertical feeds to create some code that is much more picture friendly. The default values for the vertical and horizontal feeds are both .050 inches. The picture friendly values are .1 for the vertical and .23 for the horizontal. I wouldn't use these values on my mill because the z-axis hamster's union would go on strike due to being overloaded. See the screen capture below of GCG's main menu and the option to change the default parameters. Note the current values shown in the upper portion. The next figure below shows a screen capture of the data you need to input for the first of the two rectangles. Note that the current values have changed to those entered above.

```
 GCG VERSION 2
CURRENT TOOL DIAMETER IS   .25
CURRENT FEED RATE IS   5
CURRENT VERTICAL INCREMENT IS   .05
CURRENT HORIZONTAL (RADIAL) INCREMENT .05

AVAILABLE FEATURES

0   QUIT
1   ADD DIAMETER
2   ADD HORIZONTAL LINE
3   ADD VERTICAL LINE
4   ADD POINT TO POINT LINE
5   ADD ARC
6   ADD PATH
7   RECTANGULAR POCKET
8   DRILL POINT
9   CHANGE TOOL PARAMETERS

CHOOSE 0-9? 9

CHANGE DEFAULT TOOL PARAMETERS

0 RETURN TO MAIN MENU
1 CURRENT TOOL DIAMETER IS   .25
2 CURRENT FEED RATE IS   5
3 CURRENT VERTICAL INCREMENT IS   .05
4 CURRENT HORIZONTAL (RADIAL) INCREMENT IS   .05

ENTER 0-4? 4
NEW HORIZANTAL INCREMENT ? .23

CHANGE DEFAULT TOOL PARAMETERS

0 RETURN TO MAIN MENU
1 CURRENT TOOL DIAMETER IS   .25
2 CURRENT FEED RATE IS   5
3 CURRENT VERTICAL INCREMENT IS   .05
4 CURRENT HORIZONTAL (RADIAL) INCREMENT IS   .23

ENTER 0-4?
```

Figure 8

```
 GCG VERSION 2
CURRENT TOOL DIAMETER IS   .25
CURRENT FEED RATE IS   5
CURRENT VERTICAL INCREMENT IS   .1
CURRENT HORIZONTAL (RADIAL) INCREMENT  .23
AVAILABLE FEATURES
0   QUIT
1   ADD DIAMETER
2   ADD HORIZONTAL LINE
3   ADD VERTICAL LINE
4   ADD POINT TO POINT LINE
5   ADD ARC
6   ADD PATH
7   RECTANGULAR POCKET
8   DRILL POINT
9   CHANGE TOOL PARAMETERS
CHOOSE 0-9? 7
RECTANGULAR POCKET
TOOL DIAMETER IS   .25
ENTER LOWER LEFT POINT X,Y? .708,.843
ENTER UPPER RIGHT POINT X,Y? 3.293,2.658
Y INCREMENT IS   .23
DEPTH INCREMENT IS   .1
ENTER START DEPTH ? 0
ENTER FINISH DEPTH ? .2
```

Figure 9

The code generated by GCG for these features is in appendix A, the screen plot below shows the toolpaths and the finished features are shown in the photo below that.

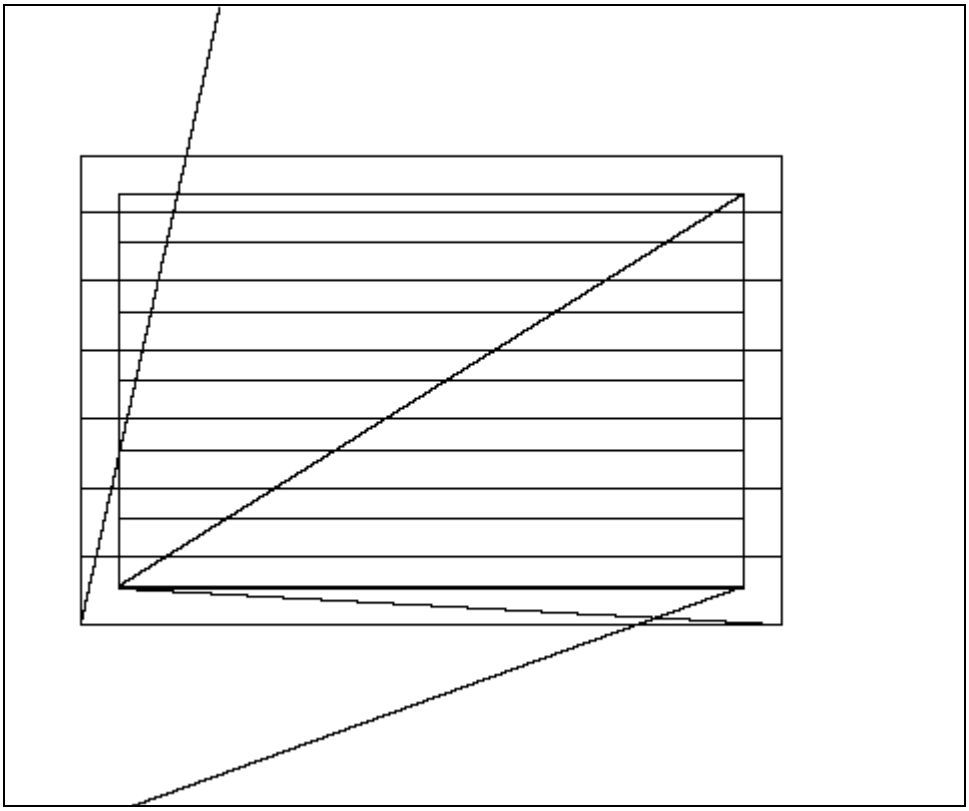


Figure 10

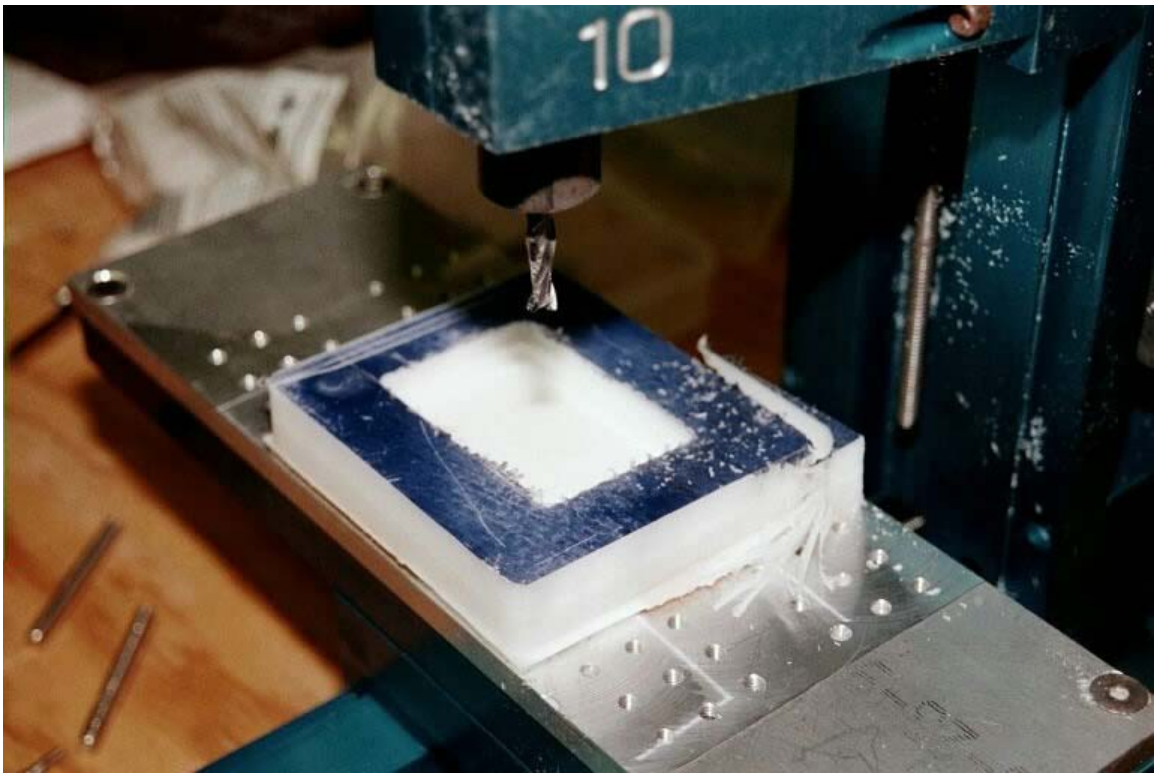


Photo 6

Section 8 Drill Point

This feature is used to mark the center of a hole for drilling later. Use a center drill. Enter the X and Y location of the holes. The default depth is .05, you can change it with feature #9, menu pick 3, Vertical increment.

Section 9 Change Tool Parameters

This is where you can modify the default values GCG uses and save them to the parameter file from which GCG "gets" them when you start. You can change the tool diameter, vertical feed rate, vertical increment, horizontal feedrate, horizontal increment and the home position. You also set where GCG moves the tool at the end of the file to either the home position or 0,0. The home position is any x,y location choosen so that you can remove the part when complete. The feedrates merely add an F XX to the end of a g-code line. See the screen capture below.

Appendix A Sample GCG output

```
(02-22-2002)
(THIS IS FEATURE #  1  )
(THIS FEATURE IS A RECTANGULAR POCKET)
(LOWER LEFT CORNER IS  X,Y  .708   .843 )
(UPPER RIGHT CORNER IS X,Y  3.293   2.658 )
(START DEPTH  0 )
(END DEPTH -.2
(Y INCREMENT  .1 )
(DEPTH INCREMENT .1 )
(MILL DIAMETER  .25 )
G01 X0.833 Y 0.968 F10
M3
G01 Z-.100 F3
G01 X 3.168 F 5
G01 Y 1.068
G01 X 0.833
G01 Y 1.168
G01 X 3.168 F 5
G01 Y 1.268
G01 X 0.833
G01 Y 1.368
G01 X 3.168 F 5
G01 Y 1.468
G01 X 0.833
G01 Y 1.568
G01 X 3.168 F 5
G01 Y 1.668
G01 X 0.833
G01 Y 1.768
G01 X 3.168 F 5
G01 Y 1.868
G01 X 0.833
G01 Y 1.968
G01 X 3.168 F 5
G01 Y 2.068
G01 X 0.833
G01 Y 2.168
G01 X 3.168 F 5
G01 Y 2.268
G01 X 0.833
G01 Y 2.368
G01 X 3.168 F 5
G01 Y 2.468
G01 X 0.833
G01 Y 2.533
G01 X 3.168 F 5
G01 Z .1
M5
G01 X0.833 Y 0.968 F10
M3
G01 Z-.100 F3
G01 Y 2.533
G01 Z .1
M5
G01 X 3.168
M3
G01 Z-.100 F3
G01 Y 0.968
G01 Z .1
M5
G01 X0.833 Y 0.968 F10
M3
G01 Z-.200 F3
G01 X 3.168 F 5
G01 Y 1.068
G01 X 0.833
G01 Y 1.168
G01 X 3.168 F 5
G01 Y 1.268
G01 X 0.833
G01 Y 1.368
G01 X 3.168 F 5
G01 Y 1.468
G01 X 0.833
G01 Y 1.568
G01 X 3.168 F 5
G01 Y 1.668
G01 X 0.833
G01 Y 1.768
G01 X 3.168 F 5
G01 Y 1.868
G01 X 0.833
G01 Y 1.968
G01 X 3.168 F 5
G01 Y 2.068
G01 X 0.833
G01 Y 2.168
G01 X 3.168 F 5
G01 Y 2.268
G01 X 0.833
G01 Y 2.368
G01 X 3.168 F 5
G01 Y 2.468
G01 X 0.833
G01 Y 2.533
G01 X 3.168 F 5
G01 Z .1
M5
G01 X0.833 Y 0.968 F10
M3
G01 Z-.200 F3
G01 Y 2.533
G01 Z .1
M5
G01 X 3.168
M3
G01 Z-.200 F3
G01 Y 0.968
G01 Z .1
M5
(END OF FEATURE # 1 )
(THIS IS FEATURE #  2  )
(THIS FEATURE IS A RECTANGULAR POCKET)
(LOWER LEFT CORNER IS  X,Y  .833   .968 )
(UPPER RIGHT CORNER IS X,Y  3.293   2.533 )
(START DEPTH -.2 )
(END DEPTH -.325
(Y INCREMENT  .1 )
(DEPTH INCREMENT .1 )
(MILL DIAMETER  .25 )
G01 X0.958 Y 1.093 F10
M3
G01 Z-.300 F3
G01 X 3.168 F 5
G01 Y 1.193
G01 X 0.958
G01 Y 1.293
G01 X 3.168 F 5
G01 Y 1.393
G01 X 0.958
G01 Y 1.493
G01 X 3.168 F 5
G01 Y 1.593
G01 X 0.958
G01 Y 1.693
G01 X 3.168 F 5
G01 Y 1.793
```

```
G01 X 0.958
G01 Y 1.893
G01 X 3.168 F 5
G01 Y 1.993
G01 X 0.958
G01 Y 2.093
G01 X 3.168 F 5
G01 Y 2.193
G01 X 0.958
G01 Y 2.293
G01 X 3.168 F 5
G01 Y 2.393
G01 X 0.958
G01 Y 2.408
G01 X 3.168 F 5
G01 Z .1
M5
G01 X0.958 Y 1.093 F10
M3
G01 Z-.300 F3
G01 Y 2.408
G01 Z .1
M5
G01 X 3.168
M3
G01 Z-.300 F3
G01 Y 1.093
G01 Z .1
M5
G01 X0.958 Y 1.093 F10
M3
G01 Z-.325 F3
G01 X 3.168 F 5
G01 Y 1.193
G01 X 0.958
G01 Y 1.293
G01 X 3.168 F 5
G01 Y 1.393
G01 X 0.958
G01 Y 1.493
```

```
G01 X 3.168 F 5
G01 Y 1.593
G01 X 0.958
G01 Y 1.693
G01 X 3.168 F 5
G01 Y 1.793
G01 X 0.958
G01 Y 1.893
G01 X 3.168 F 5
G01 Y 1.993
G01 X 0.958
G01 Y 2.093
G01 X 3.168 F 5
G01 Y 2.193
G01 X 0.958
G01 Y 2.293
G01 X 3.168 F 5
G01 Y 2.393
G01 X 0.958
G01 Y 2.408
G01 X 3.168 F 5
G01 Z .1
M5
G01 X0.958 Y 1.093 F10
M3
G01 Z-.325 F3
G01 Y 2.408
G01 Z .1
M5
G01 X 3.168
M3
G01 Z-.325 F3
G01 Y 1.093
G01 Z .1
M5
(END OF FEATURE # 2 )
G01 Z .1
G01 X0 Y0
M5
M30
```

Appendix B Frequently Asked Questions

Well…go ahead and ask! If I think it is worth it. I'll add it in the next revision.

1. This program is so awesome! How can I encourage the author to continue adding features and improve the existing ones?

1A. you can send him money, chocolate and beer and convince 5 of your friends to do the same.

2. How will I know when more wonderful and exciting features are added to GCG.

2A. Pay me $5 through paypal and I will add you to the e-mail list. You will get notices or maybe even the entire updated program and manual zipped in your inbox. If you want to pay more, then go right ahead!

3 What is mousepower?

3A. Mousepower is a term for describing the power output of small machines, such as mills, lathes etc. It is needed by the hobby machining community to impress their wives and girlfriends. Example: "My mill has a 86mp AC Motor". Sounds Much Better than 1/12 HP. 1mp = .001HP or .746 watts means 1 mousepower = 1/1000 horsepower.

Appendix C Future improvements

Listed in no important order, not how important I think they are. Get on the e-mail list to find out when these will be released.

1. Anti-plunge: An option to remove the plunge from the current depth to the next depth at one location and spread it out over a certain distance. This will make the code output by GCG a little longer, but you will be able to make larger vertical increments, thus making the code smaller. Go Figure.
2. Scaled Text: Not engraving, but actual machined text 26 letters and 0-9. I designed the font for easy programming, not to look good. It will scale from ¼ inch with a 1/16 end mill to 1 inch with a ¼ inch end mill.
3. Canned cut outs for standard size switches and LEDS etc for electronic panels
4. Support for TurboCNC a neat program that will run your homespun mill from GCG generated G-code. I will also add features to support other drives such as Gecko and Linistepper as time permits.
5. Variable z clearance (already done!)
6. Set Home option (already done this too!)
7. Goto home option (guess what, I did this too.)
8. Gcg Parameter File (damn, I am good!)
9. Verbose Output file

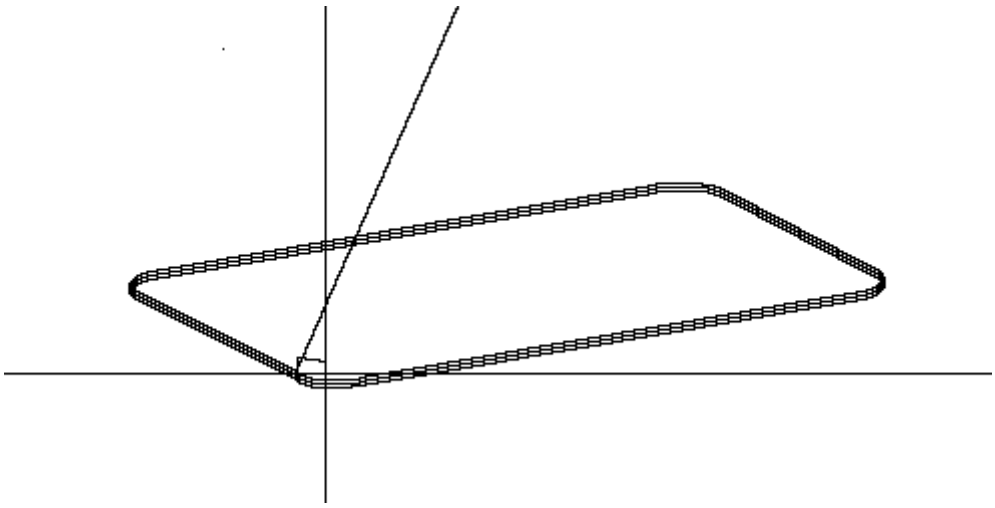
10. Ask me for anything else you might want, maybe I'll put it in.

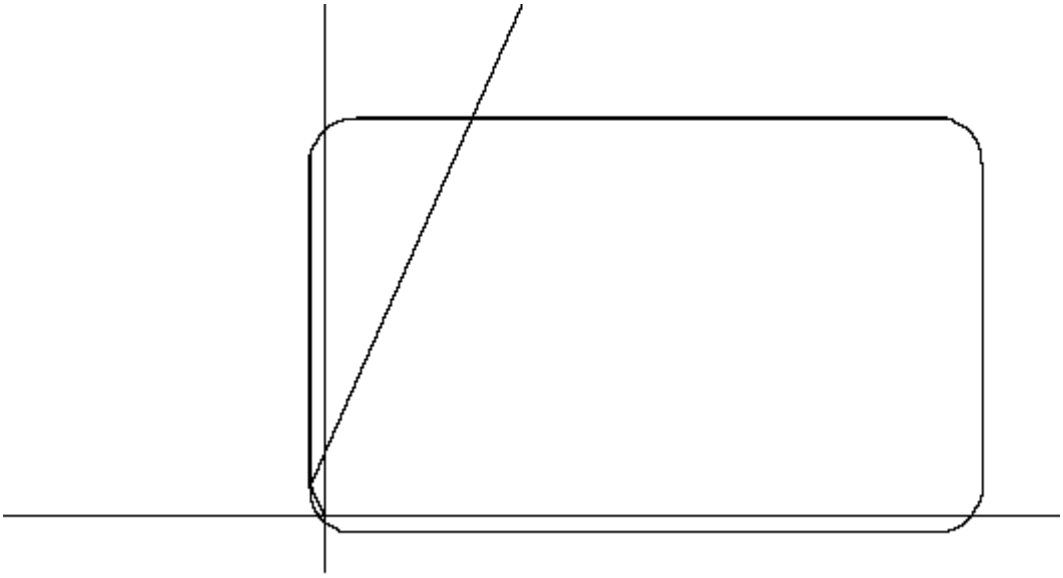
Appendix D Step by Step method to create the plate in figure 4

1. Mark and rough cut a slab of aluminum. I cut mine on an old craftsman bandsaw, but hey a tough guy like you could probably rip it with your bare hands or use a hacksaw. Try to keep it flat, it is aluminum. Make it big enough that it won't matter if it doesn't get mounted perfectly in line with X and Y.
2. Tape the bottom with double sided sticky tape, mine came from McMaster-Carr (www.mcmaster.com). You might want to clean both the slab and your machine with alcohol (no, the rubbing type!) to increase adhesion.
3. Trim the edges of the tape.
4. Mount the slab to the machine
5. Use GCG, Feature 6, Option 1 and the text file f4outside.txt to create the code for cutting the outside.
6. Cut the outside. Here is the code:

```
(F4OUT.TXT)
(04-26-2003)
G01 X-.125 Y 0.250 F10
M3
G01 Z 0 F10
G01 Z-.030 F1
G01 X -.125 Y 2.750 F5.000
G02 X 0.250 Y 3.125 I  0.375 J  0.000 F5.000
G01 X 4.750 Y 3.125 F5.000
G02 X 5.125 Y 2.750 I  0.000 J -0.375 F5.000
G01 X 5.125 Y 0.250 F5.000
G02 X 4.750 Y -.125 I -0.375 J  0.000 F5.000
G01 X 0.250 Y -.125 F5.000
G02 X -.125 Y 0.250 I  0.000 J  0.375 F5.000
(FULL DEPTH INCREMENT)
(NEW DEPTH IS-.06  )
G01 Z -.060 F1
G01 X -.125 Y 2.750 F5.000
G02 X 0.250 Y 3.125 I  0.375 J  0.000 F5.000
G01 X 4.750 Y 3.125 F5.000
G02 X 5.125 Y 2.750 I  0.000 J -0.375 F5.000
G01 X 5.125 Y 0.250 F5.000
G02 X 4.750 Y -.125 I -0.375 J  0.000 F5.000
G01 X 0.250 Y -.125 F5.000
G02 X -.125 Y 0.250 I  0.000 J  0.375 F5.000
(FULL DEPTH INCREMENT)
(NEW DEPTH IS-.09  )
G01 Z -.090 F1
G01 X -.125 Y 2.750 F5.000
G02 X 0.250 Y 3.125 I  0.375 J  0.000 F5.000
G01 X 4.750 Y 3.125 F5.000
G02 X 5.125 Y 2.750 I  0.000 J -0.375 F5.000
G01 X 5.125 Y 0.250 F5.000
G02 X 4.750 Y -.125 I -0.375 J  0.000 F5.000
G01 X 0.250 Y -.125 F5.000
G02 X -.125 Y 0.250 I  0.000 J  0.375 F5.000
G01 Z .1 F10
M5
(END OF FEATURE # 1 )
G01 X0 Y0 F10
M5
M30
```

Here is the screen plots:





Here is the photo:

Opps- I lost the film, will add the photos for the section in the next Revision

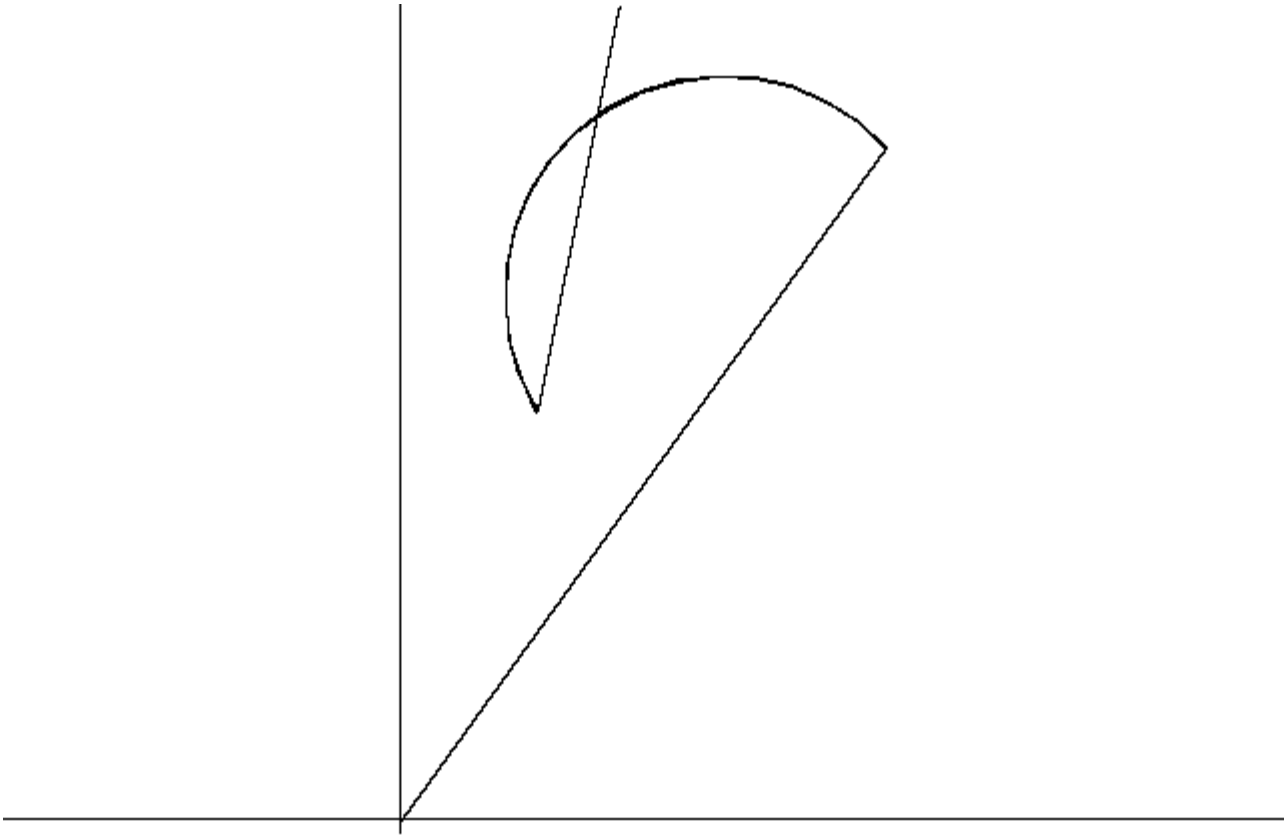
7. Now we'll do the larger ¼" width arc

The code:

```
(F4_ARC1.TXT)
(04-26-2003)
(THIS FEATURE IS AN ARC)
(CENTER AT  1.1              1.8  )
(STARTING AT X,Y = 0.466,1.400 )
(FINISHING AT X,Y = 1.659,2.300 )
```

```
(START DEPTH = 0.000 )
(FINISH DEPTH =-.050 )
(DEPTH INCREMENT =  .02  )
G01 X0.466 Y 1.400 F10
M3
G01 Z 0 F10
G01 Z-.020 F1
G02 X  1.659 Y  2.300 I  0.634 J  0.400 F5.0
(FULL DEPTH INCREMENT)
(NEW DEPTH IS-.04  )
G01 Z -.040 F1
G03 X  0.466 Y  1.400 I -0.559 J -0.500 F5.0
(PARTIAL DEPTH INCREMENT)
(NEW DEPTH =-.05  )
G01 Z -.050 F1
G02 X  1.659 Y  2.300 I  0.634 J  0.400 F5.0
G01 Z .1 F10
M5
(END OF FEATURE # 1 )
G01 X0 Y0 F10
M5
M30
```

The screen plot:



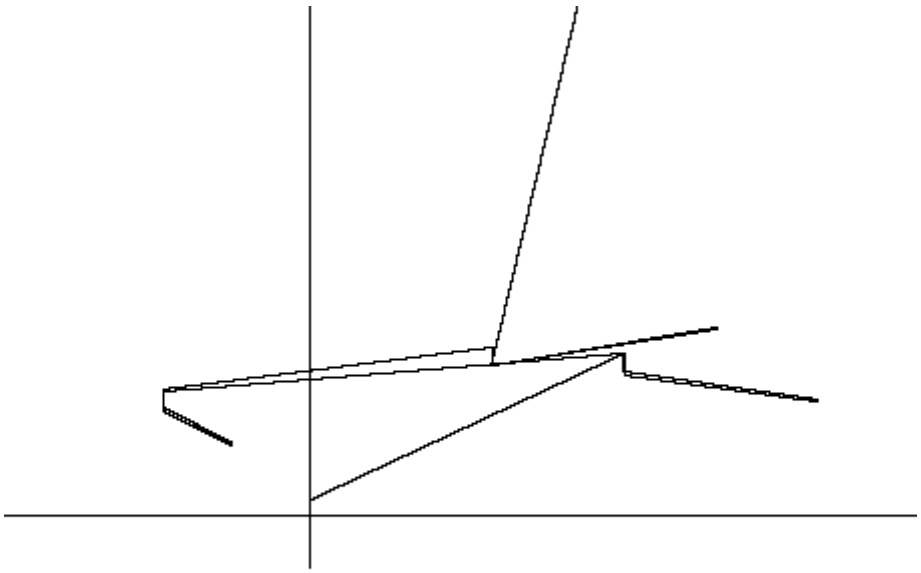
The Photo:

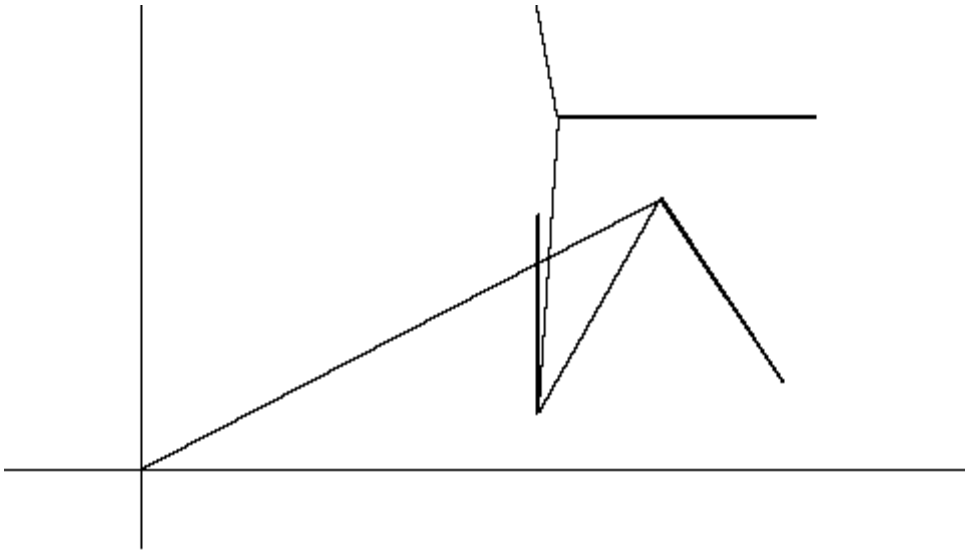
8. Next are all three lines:

The code:

```
The code:(F4_LINES.TXT)
(04-26-2003)
(THIS IS FEATURE #  1 )
(THIS FEATURE IS A HORIZONTAL LINE)
(STARTING AT X,Y = 2.625,2.250)
(FINISHING AT X,Y = 4.250,2.250)
(START DEPTH = 0.000)
(FINISH DEPTH =-.030)
(DEPTH INCREMENT =  .02 )
G01 X2.625 Y 2.250 F10
M3
G01 Z 0 F10
G01 Z-.020 F1
G01 X 4.250 Y 2.250 F5.000
(PARTIAL DEPTH INCREMENT)
(NEW DEPTH =-.03  )
G01 Z -.030 F1
G01 X 2.625 Y 2.250 F5.000
G01 Z .1 F10
M5
(END OF FEATURE # 1 )
(THIS IS FEATURE #  2 )
(THIS FEATURE IS A VERTICAL LINE)
(STARTING AT X,Y = 0.375,2.500)
(FINISHING AT X,Y = 0.375,1.625)
(START DEPTH = 0.000)
(FINISH DEPTH =-.030)
(DEPTH INCREMENT =  .02 )
G01 X0.375 Y 2.500 F10
M3
G01 Z 0 F10
G01 Z-.020 F1
G01 X 0.375 Y 1.625 F5.000
(PARTIAL DEPTH INCREMENT)
(NEW DEPTH =-.03  )
G01 Z -.030 F1
G01 X 0.375 Y 2.500 F5.000
G01 Z .1 F10
M5
(END OF FEATURE # 2 )
(THIS IS FEATURE #  3 )
(THIS FEATURE IS A POINT TO POINT LINE)
(STARTING AT X,Y = 3.275,1.721)
(FINISHING AT X,Y = 4.036,0.580)
(START DEPTH = 0.000)
(FINISH DEPTH =-.030)
(DEPTH INCREMENT =  .02 )
G01 X3.275 Y 1.721 F10
M3
G01 Z 0 F10
G01 Z-.020 F1
G01 X 4.036 Y 0.580 F5.000
(PARTIAL DEPTH INCREMENT)
(NEW DEPTH =-.03  )
G01 Z -.030 F1
G01 X 3.275 Y 1.721 F5.000
G01 Z .1 F10
M5
(END OF FEATURE # 3 )
G01 X0 Y0 F10
M5
M30
```

The screen plots:





The Picture:

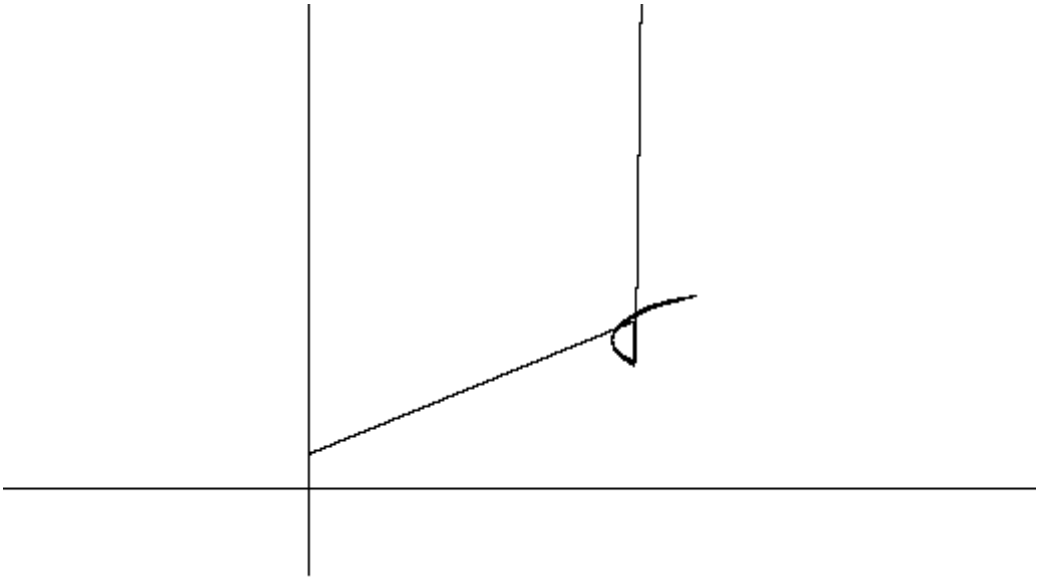
9. Last (maybe) is the small arc.

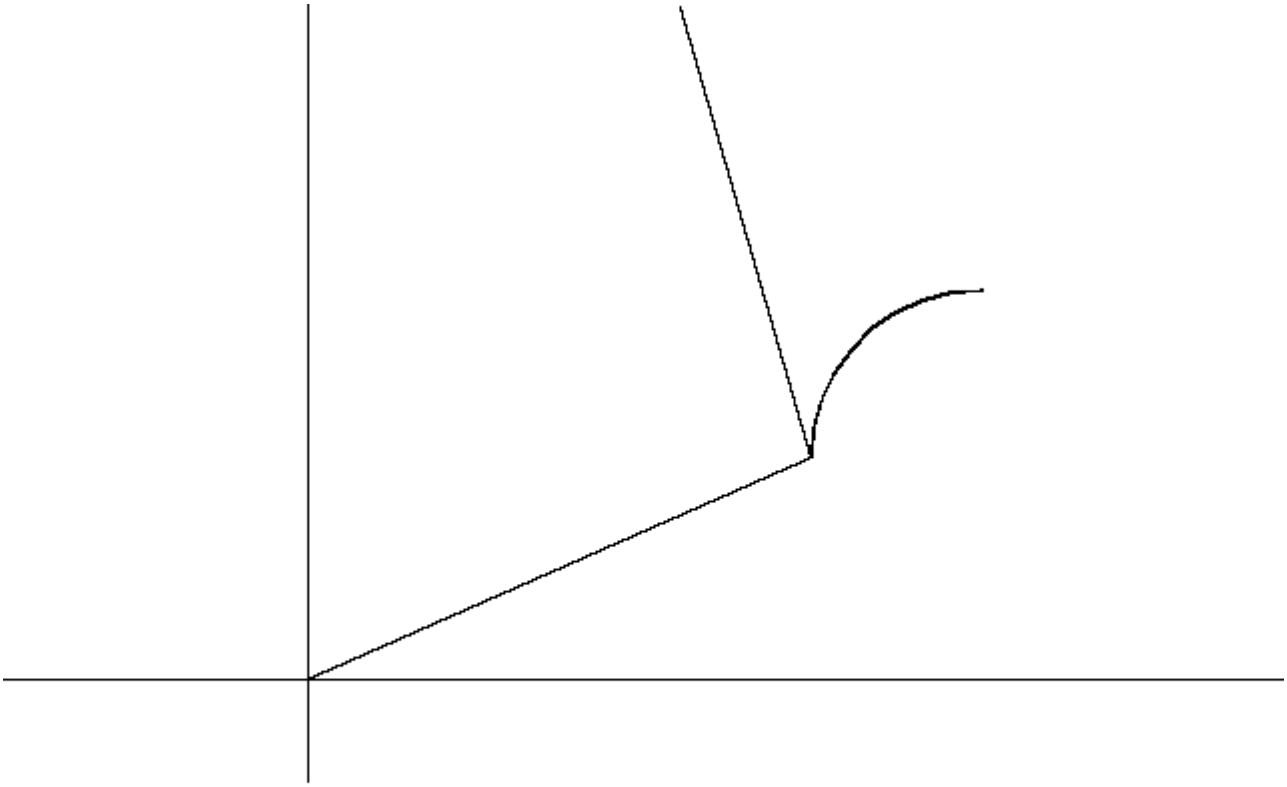
Here is the Code:

```
(F4_ARC2.TXT)
(04-26-2003)
(THIS FEATURE IS AN ARC)
(CENTER AT  1.889            .625  )
(STARTING AT X,Y = 1.414,0.625 )
(FINISHING AT X,Y = 1.889,1.100 )
(START DEPTH = 0.000 )
(FINISH DEPTH =-.030 )
(DEPTH INCREMENT =  .02  )
G01 X1.414 Y 0.625 F10
```

```
M3
G01 Z 0 F10
G01 Z-.020 F1
G02 X  1.889 Y  1.100 I  0.475 J  0.000 F5.0
(PARTIAL DEPTH INCREMENT)
(NEW DEPTH =-.03  )
G01 Z -.030 F1
G03 X  1.414 Y  0.625 I  0.000 J -0.475 F5.0
G01 Z .1 F10
M5
(END OF FEATURE # 1 )
G01 X0 Y0 F10
M5
M30
```

Here are the screen plots:





10. Lets add a diameter or two, just for fun! Naw, I'll wait until the next revision of this manual, cause I lost the film!

Appendix E Notes on G-code

GCG uses the following codes

G0 rapid positioning
G1 linear interpolation
G2 circular/helical interpolation (clockwise)
G3 circular/helical interpolation (c-clockwise)
M3 Spindle on
M5 Spindle off
M30 End Program